Boundary Condition Surface Loads

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
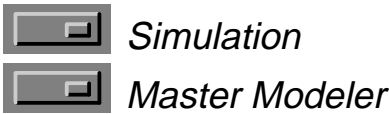- Free Meshing
- Boundary Condition Sets

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.**

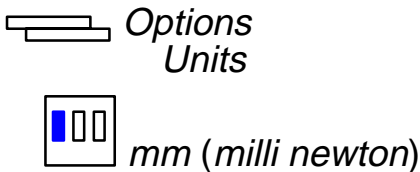Otherwise, open a new model file and give it a unique name.

Hint

File
       Open

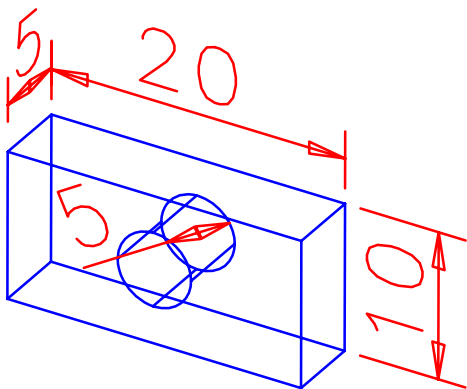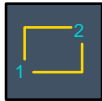Make sure you're in the following application and task:

Simulation

Master Modeler

Set your units to mm.

Options
       Units

mm (*milli newton*)

What: Create the part shown.

Hint

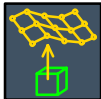**What:** Name the part.

Hint

Name: Link Part

What: Create an FE model named "FEM3" associated with the part.
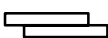
Hint

 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

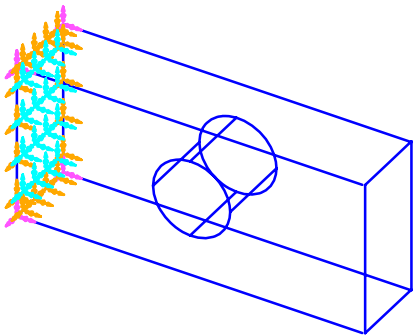Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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*
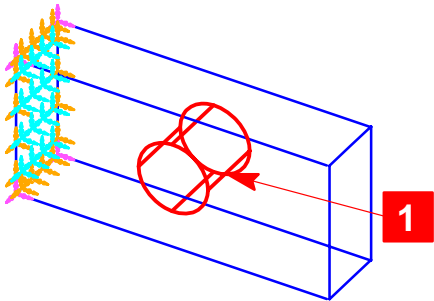

Recovery Point
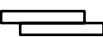
▭ *File*
   Save

What: Create a force on the cylindrical surface.

1

Done

Traction on Surface form

Load Set: Pin Force

◆ *Vector*

2
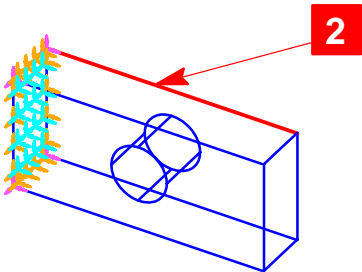
 No or *Yes* (to switch vector in +X direction)

Traction on Surface form

◆ *Total Force*

Traction: 1000

OK
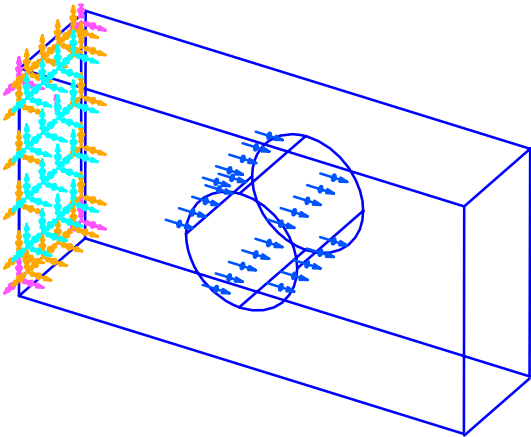
Recovery Point

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
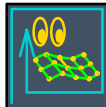
What: Solve the model.

Hint

 Model Solution

 Create

What: Display the results.
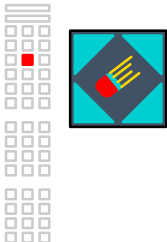
Hint

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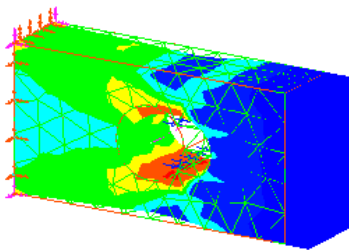
What: Before continuing with the tutorial, delete the results from the solve.

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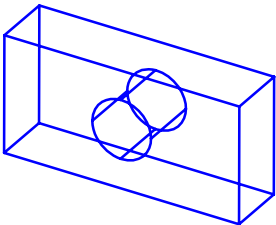
Why: When you modify a part, the FE model is also automatically updated if it's on the workbench. By putting it away while you make changes, you avoid updating the model after each change.

Recovery Point

 File
    Save

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

Hint

 Master Modeler

Sketch two lines.
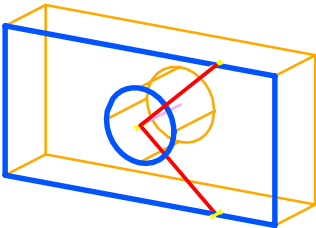
Extrude and partition.

 Section Options...

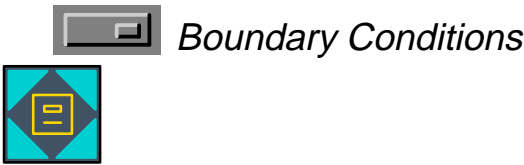 *Stop at intersections* (off)

 Partition

1
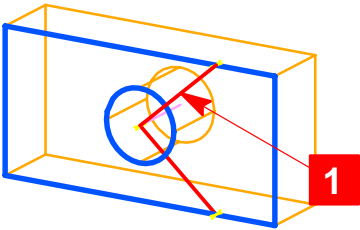
 OK

Result

Recovery Point

⬜ *File*
   Save
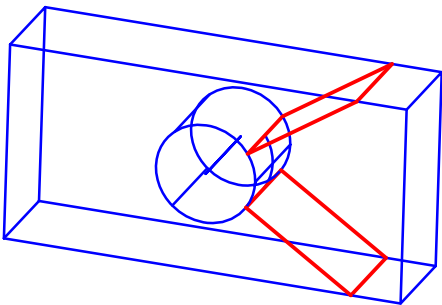
What: Get the FE model "FEM3" back onto the workbench.

Hint

⬜ *Boundary Conditions*

⬜ FEM3
⬜ *Get*

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
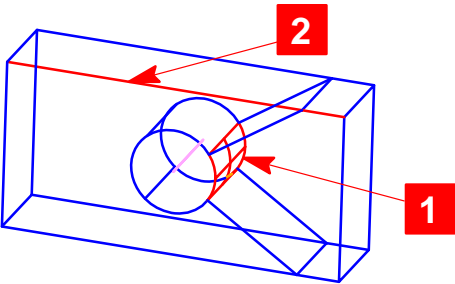
 Done

___

 Load Set: Pin Force

2 pick edge

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

◆ *Total Force*

Traction: 1000

▭ *OK*

___

Result
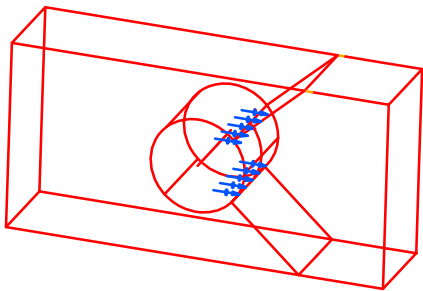
What: Check the boundary condition set.

Hint
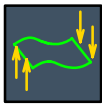
Boundary Condition Set Management form

 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

 File
      Save

What: Solve the model.

Hint

 Model Solution

What: Display the results.

Hint

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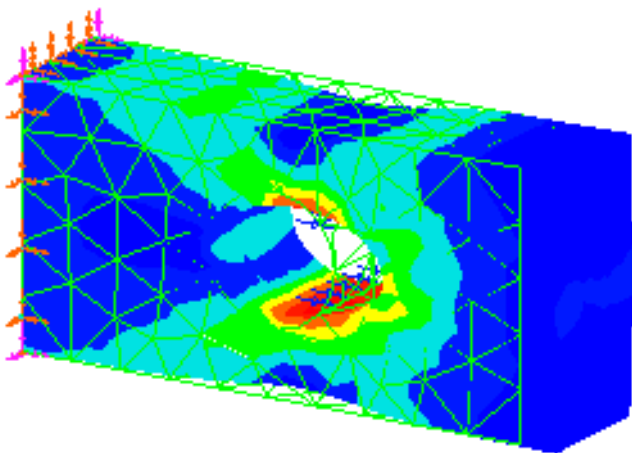
One way to model the condition of rotation around a pin is to use a local cylindrical coordinate system (R, Theta, Z) in the center of the hole, to restrain each node on the edge of the hole in the R direction.

What: Delete the results.

Hint

 All

 Yes

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

 Master Modeler
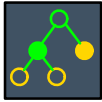
1
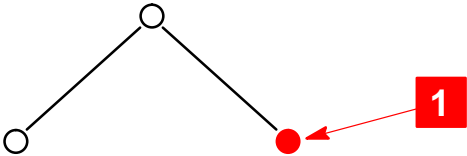
 Boundary Conditions

2
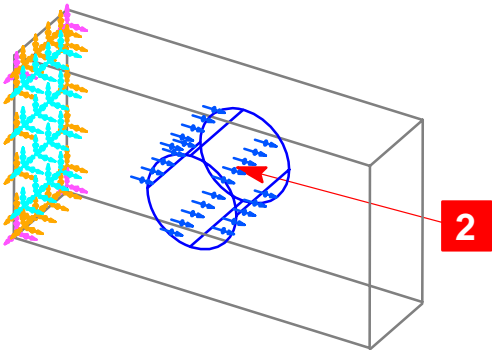
What: Create a cylindrical coordinate system at the hole center.

Hint

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

How:

1
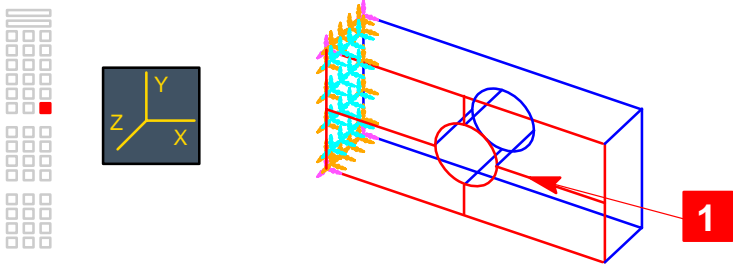
🔲 *Origin*

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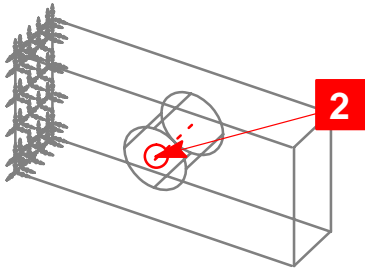
Why: The coordinate system will be created in the center without the *Origin* selection, but it may not remain there if you move or size the part. Selecting *Origin* ensures the coordinate system stays in the center.

🔲 *Options*...

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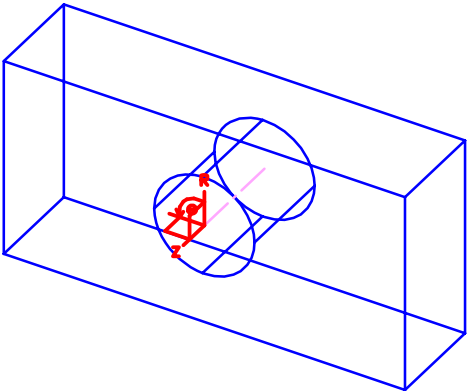
💡 You can restrain one node against translation and
still allow rotation. However, when you want to allow
a **group** of nodes to be free to rotate as a unit but
not translate, don't restrain each node against
translation. Restraining multiple nodes against
translation in effect restrains overall rotation.

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
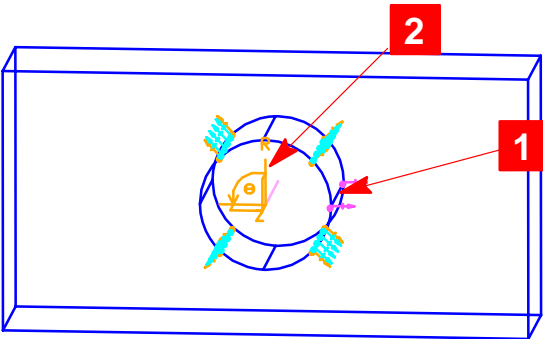
What: Solve the model and display results.

Hint

Model Solution

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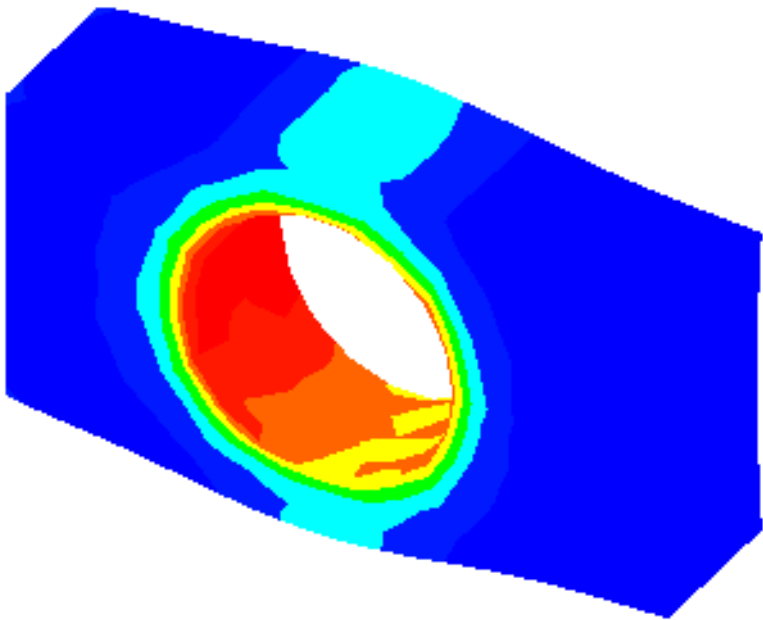
You could also use contact elements to simulate a press fit.

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.

    File
        Save

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
    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:

File
   Exit

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