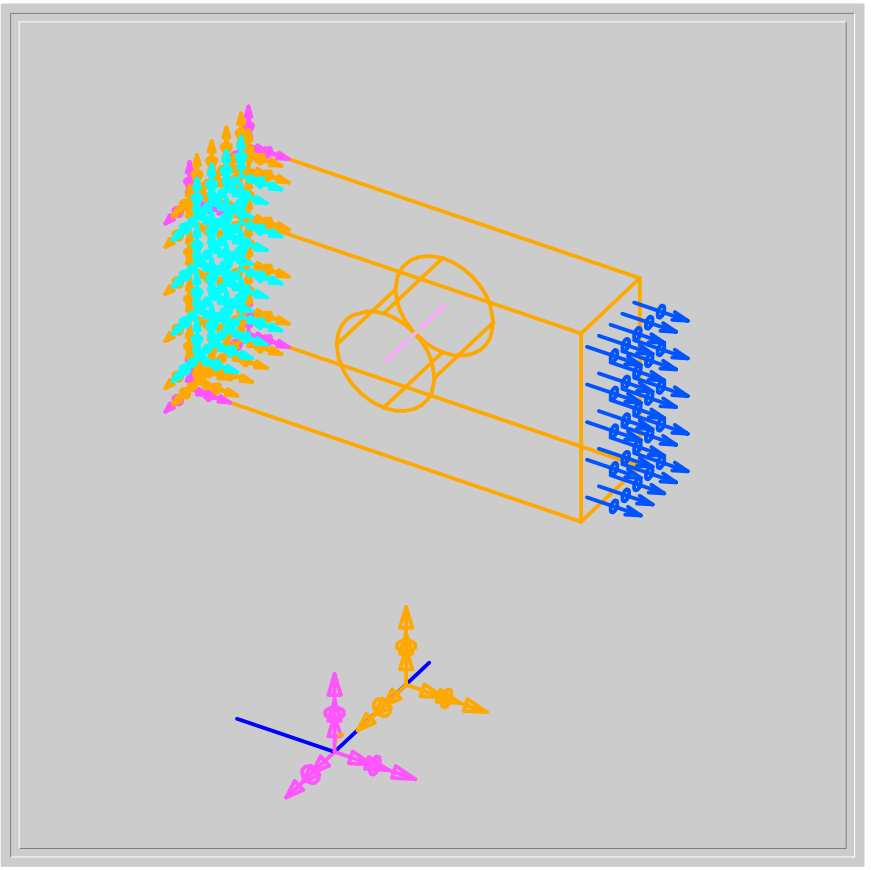


Boundary Condition Sets

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.

File
Open

Open Model File form

Model File name: any unique name
OK

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

Simulation
Master Modeler

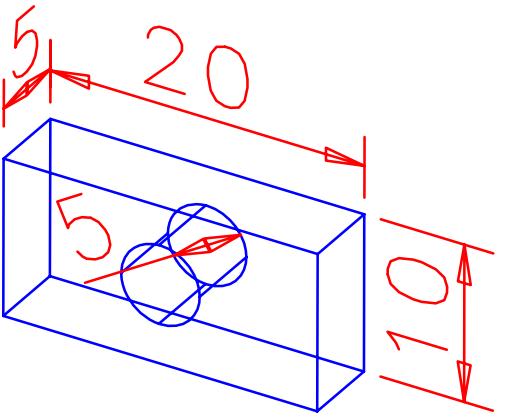
Set your units to mm.

Options
Units

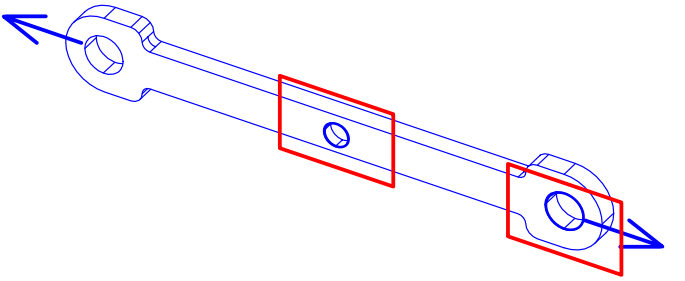
mm (*milli newton*)

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.

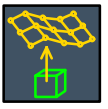
Hint

Name: Link Part

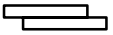
What: Create an FE model named “FEM1” associated with the part.

Hint


Boundary Conditions



Save your model file.


File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:


No

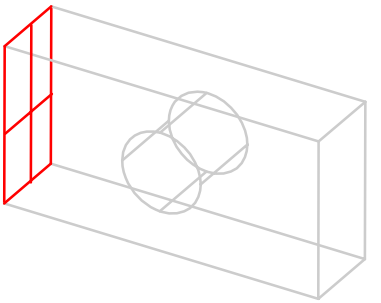
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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.

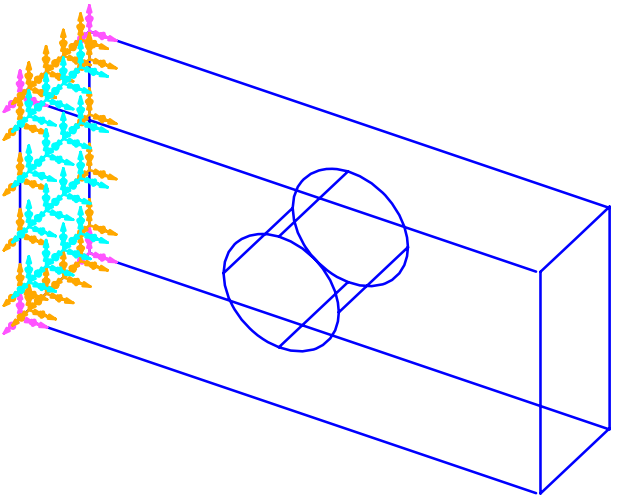
What: Restrain the face shown.

Hint



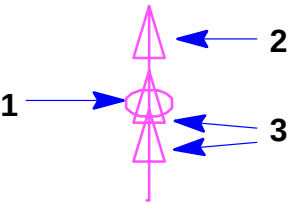
Displacement Restraint on Surface form

OK



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

Restraint

Rename...

Rename form

Clamped End Restraint

OK

Set Management form

Dismiss

Recovery Point

File

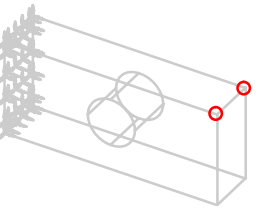
Save

You can create forces on:

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

What: Create forces on the two vertices shown.

How:



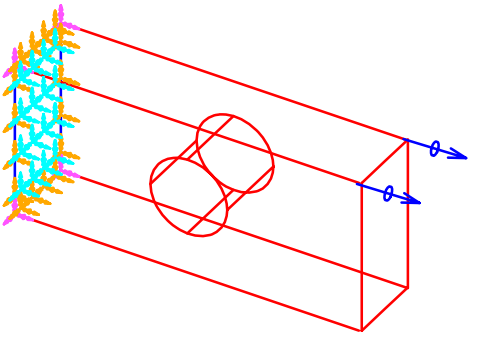
shift-pick two vertices

Done

Force on Vertex/Location on Geometry form

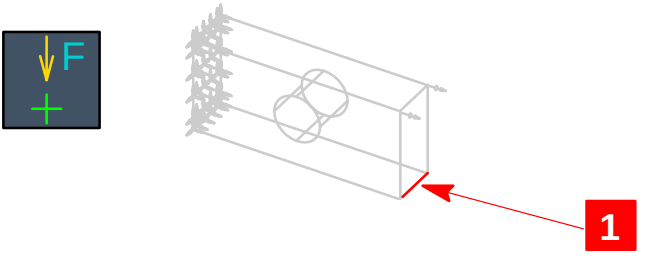
X Force: 250

OK



What: Create a force on the bottom edge.

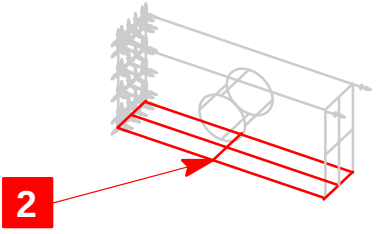
How:



1

Done

2

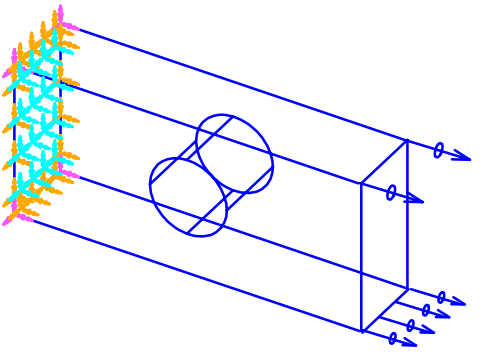


Force on Edge form

Total Force

In Plane Force: –500

OK



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

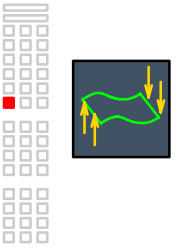
Save

Create a boundary condition set 1 of 1

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)
- *OK*

Recovery Point

File
Save

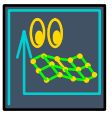
What: Create a solid mesh.

Hint

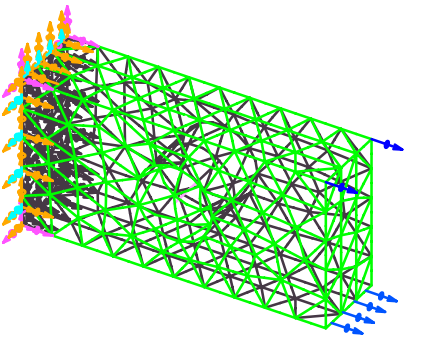
 *Meshing*

 pick part

 *Element Length*: 2



Modify Mesh Preview form



 *Keep Mesh*

What: Solve the model.

Hint


Model Solution


Create


OK


Dismiss

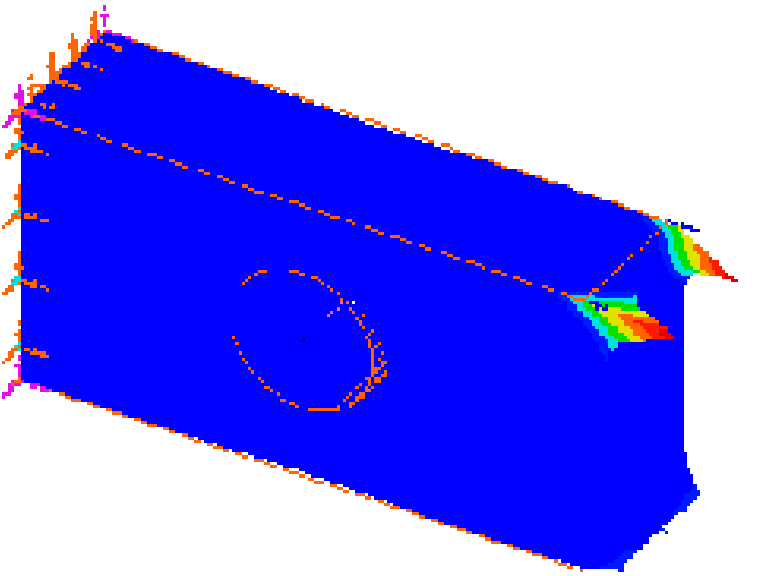

Check *I-DEAS List*.
Look for any warning or error messages.

What: Display results with all elements selected.

Hint

 *Post Processing*

 (default is all)



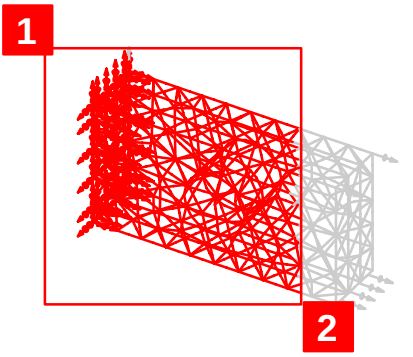
Things to notice

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.

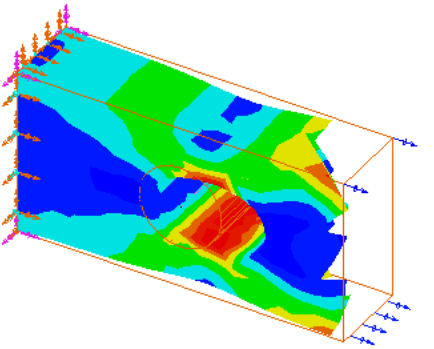
What: Display results with elements nearer the hole (away from the forces) selected.

Hint

 use area–select to pick most of the elements.



 *Done*

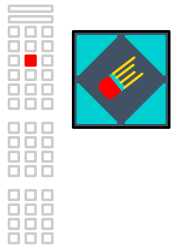


Things to notice

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:



All

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

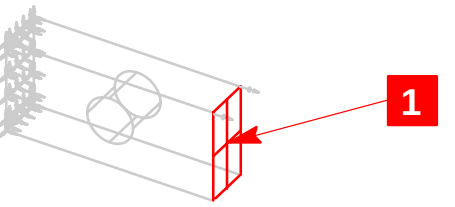
Recovery Point

File
Save

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

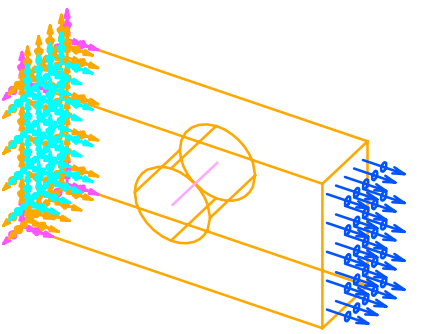

Load Set: Face Pressure


Total Force


Force: –1000


OK

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



Create a pressure 2 of 2

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

Hint

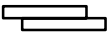
 Tension – Face Load

 *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

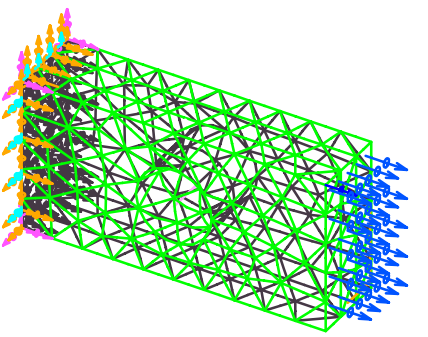

Model Solution


Create


Boundary Condition Set:
Tension – Face Load


OK


Dismiss



What: Display the results.

How:

 *Post Processing*

Results Selection form

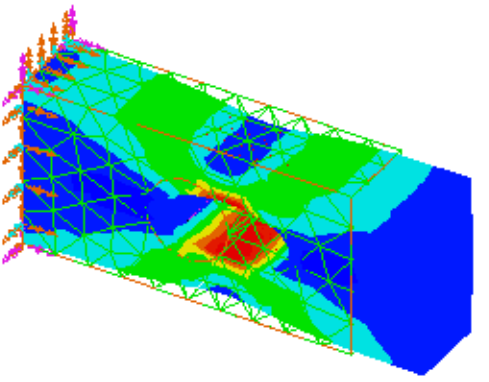
 B.C. 2, STRESS...

 *Display Results*

 B.C. 2, DISPLACEMENT...

 *Deformation Results*

 *OK*

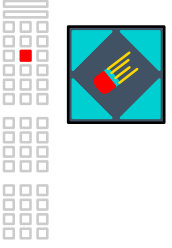


Things to notice

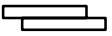
In both this case and in the preceding example, the stresses around the hole were valid. But in this example, you have avoided singularity where the load is applied.

What: Delete the results.

Hint



Recovery Point



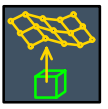
File
Save

Compare boundary condition types 1 of 5

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

Hint


Boundary Conditions




Do not turn on the toggle:


Geometry Based Analysis Only

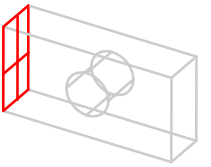
Why: With this toggle on, you can't create FE-based boundary conditions.

- 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.

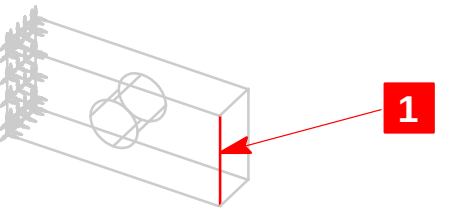
What: Restrain the face shown.

Hint



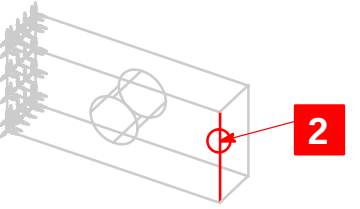
What: Create a force on the edge shown.

Hint



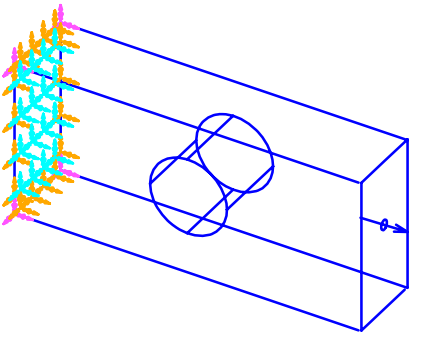
1

Locations on Edge



2

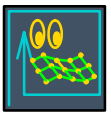
X Force: 500

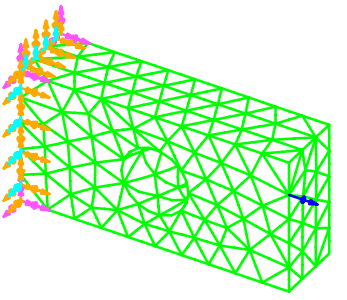


What: Create a solid mesh.

Hint

Meshing

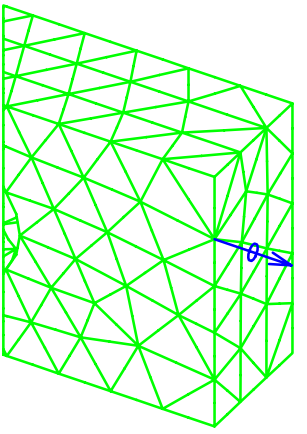




Keep Mesh

Things to notice

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

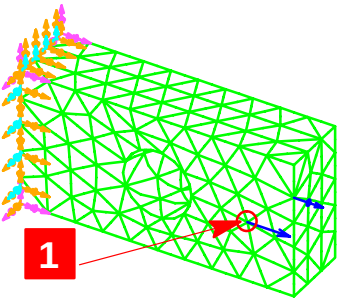


What: Create a force on a node as shown.

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)

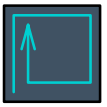
 FEM1 (select)

Compare boundary condition types 5 of 5

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

Hint

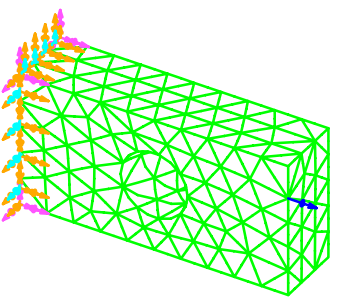
 *Master Modeler*



 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.

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
- 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:

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

Structural Dynamics Research Corporation
2000 Eastman Drive
Milford, Ohio 45150
(513) 576−2400