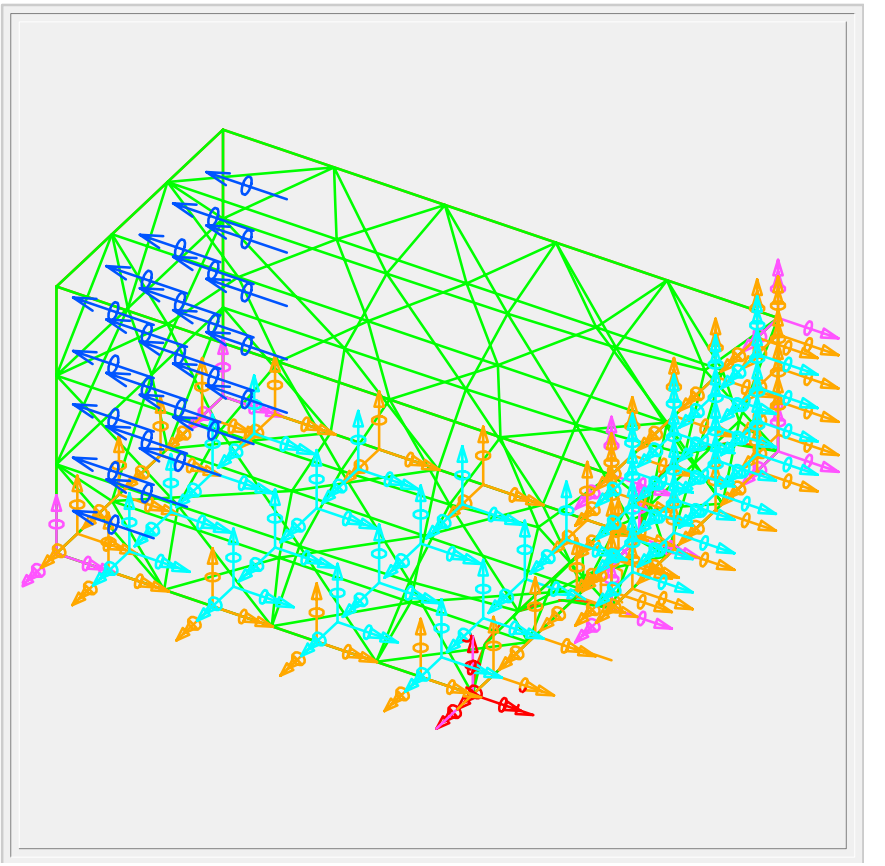


Boundary Condition Symmetry

I-DEAS™ Tutorials: Fundamental Skills

This tutorial shows the use of symmetry and anti-symmetry restraints. These methods will produce faster solve times and smaller file sizes.

Learn how to:

- use symmetry restraints
- use anti-symmetry restraints

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
- Boundary Condition Surface Loads

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

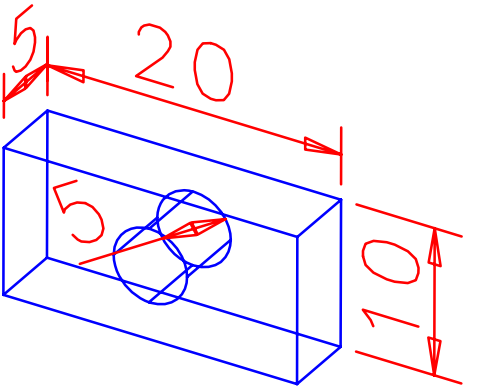
Set your units to mm.

Options
Units


mm (*milli newton*)

What: Create the part shown.

Hint



What: Name the part.

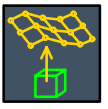
Hint

☐ *Name*: Link Part

What: Create an FE model named “FEM4” associated with the part.

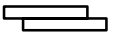
Hint


Boundary Conditions




FE Model Name: FEM4

Save your model file.


File
Save

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


No

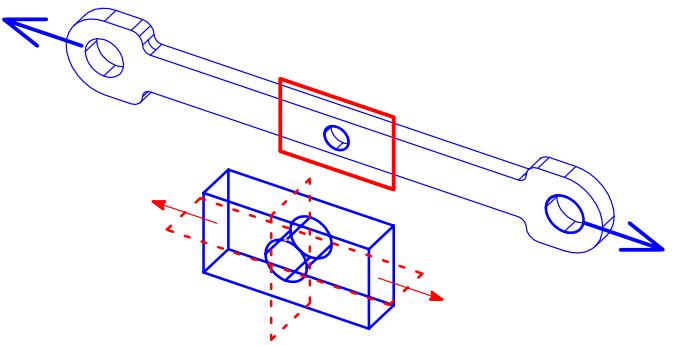
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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.

If a part is symmetric and the loading is symmetric, you can analyze just one half of the part and get the same results as if you had analyzed the entire part. If there is more than one symmetry plane, you can even cut the model more than once.

This section shows you how to analyze the stress concentration on the pin hole in the center of a linkage, when the linkage is being pulled at both ends. This is an example of having symmetry on two planes.



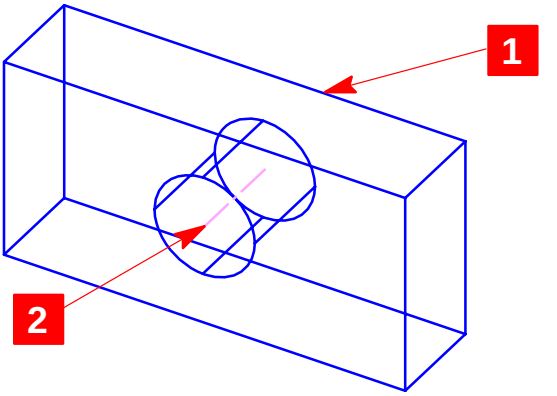
To use symmetry, you must provide the correct restraints. Along the plane of symmetry where the model is cut, nodes cannot move perpendicular to the plane or rotate in the plane.

For example, if the symmetry plane is the plane where X is a constant, all nodes on this plane must be restrained in X translation, Y rotation, and Z rotation.

What: Cut the part on the symmetry plane.

How:


Master Modeler



1

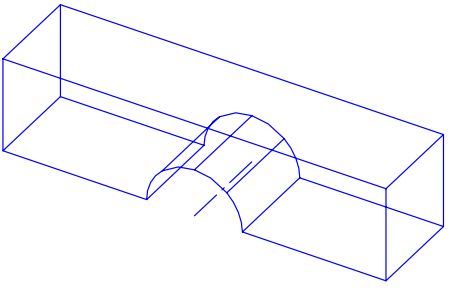

Axis Planes


ZX Plane


Pick point

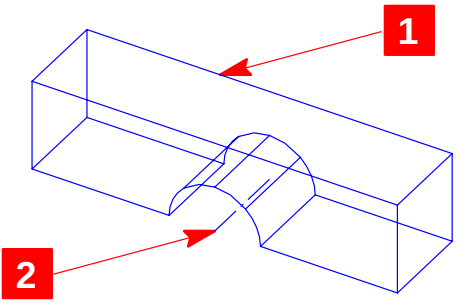
2 pick circle center point


Keep Positive Side



What: Cut the part again on the other symmetry plane.

How:



1

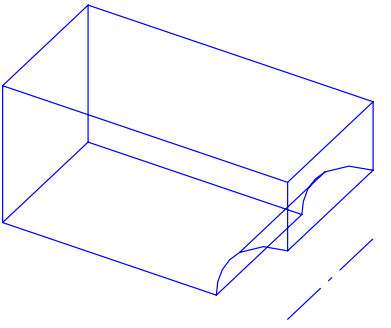
Axis Planes

YZ Plane

Pick point

2 pick circle center point

Keep Negative Side



Recovery Point

File
Save

What: Create a new restraint set.

Hint


Boundary Conditions


Set: Symmetry


Restraint


Create

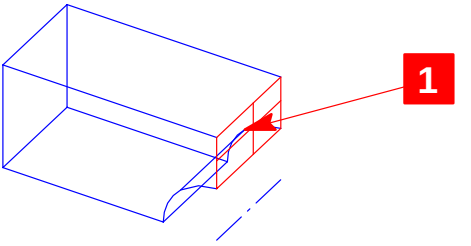

Make Current


Dismiss

What: Restrain the cut faces to maintain symmetry.

Why: To properly represent the entire part, you must prevent movement along X for the vertical face and along Y for the horizontal face.

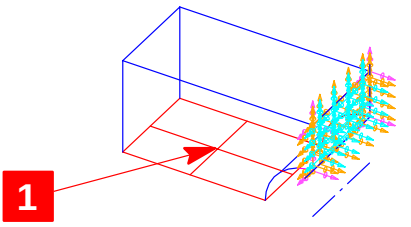
Hint



Continued on next page...

Displacement Restraint on Surface form

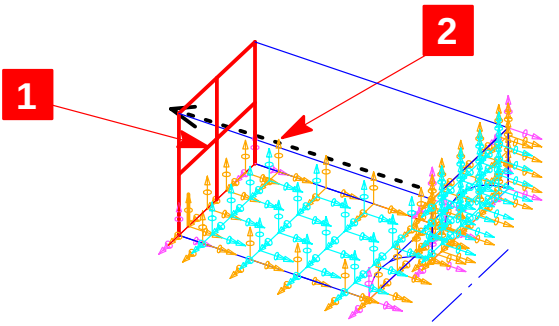
- *X Translation*: *constant*
- *Y Translation*: *free*
- *Z Translation*: *free*
- *X Rotation*: *free*
- *Y Rotation*: *constant*
- *Z Rotation*: *constant*
- *OK*



- *X Translation*: *free*
- *Y Translation*: *constant*
- *Z Translation*: *free*
- *X Rotation*: *constant*
- *Y Rotation*: *free*
- *Z Rotation*: *constant*
- *OK*

What: Apply a force to simulate pulling.

Hint



1 pick surface

Load Set: Symmetry Force

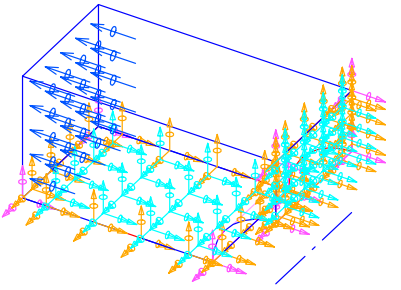
Vector

2 pick edge

Yes (when arrow is as shown above)

Total Force

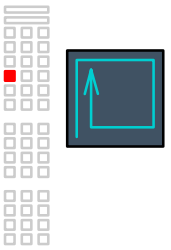
Traction: 500

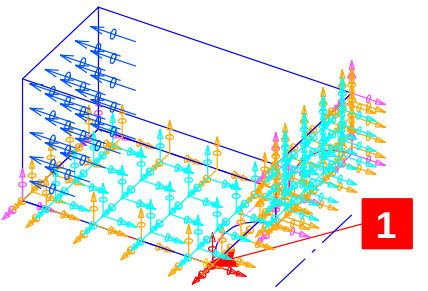


What: Modify one restraint to restrict rigid body movement.

Why: Even though the model looks fully restrained, there is still a possibility of movement in the Z direction. The model may still solve correctly without the additional restraint, but you will get a warning in the *I-DEAS List* window.

How:





1 pick corner restraint

Displacement Restraint on ... form

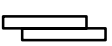
 *Specified*

 *Specify Restraint...*

 *Z Translation*: *constant*

 *OK* (all forms)

Recovery Point

 *File*
Save

What: Create a boundary condition set.

Hint

Symmetry Pull

Restraint Set

Symmetry

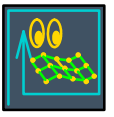
Load Sets: Symmetry Force

What: Create a solid mesh.

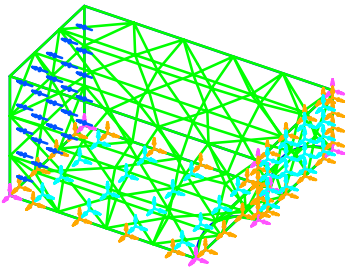
Hint

Meshing

Element Length: 2



Modify Mesh Preview form



Keep Mesh

What: Solve the model.

Hint

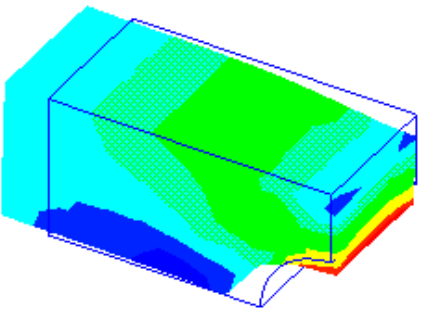

Model Solution


Create

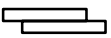
What: Display the results.

Hint


Post Processing



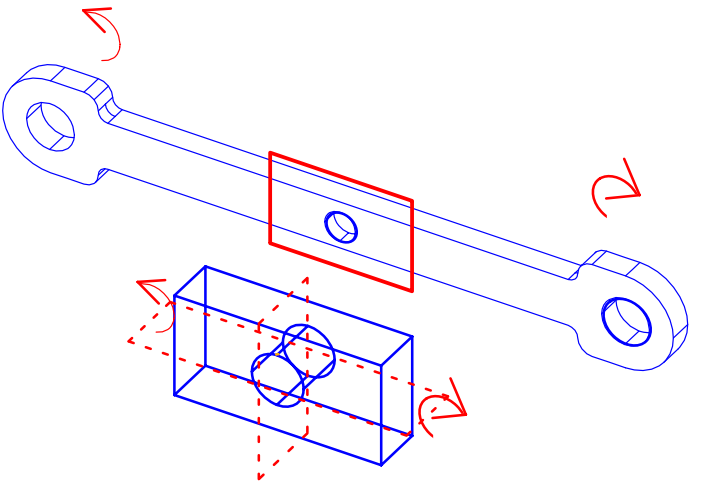
Recovery Point


File
Save

Use anti-symmetry restraints 1 of 7

Another symmetry condition is anti-symmetry. This is where the part is symmetric, but the load is reversed across the plane of symmetry.

In the bending case shown below, the vertical cutting plane is symmetric, but the horizontal plane is anti–symmetric. On the horizontal plane, where Y is a constant, you'll restrain the X translation, Z translation, and Y rotation.



What: Create a new restraint set.

Hint

Boundary Conditions

Restraint

Set: Anti-Symmetry

Create

Make Current

Load

Set: Bending Force

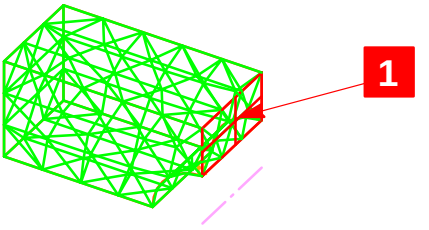
Create

Make Current

Dismiss

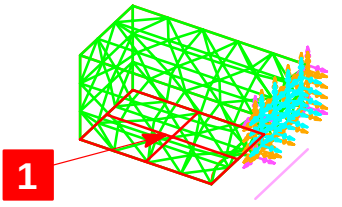
What: Restrain the cut faces.

Hint



Continued on next page...

- *X Translation*: *constant*
- *Y Translation*: *free*
- *Z Translation*: *free*
- *X Rotation*: *free*
- *Y Rotation*: *constant*
- *Z Rotation*: *constant*
- *OK*

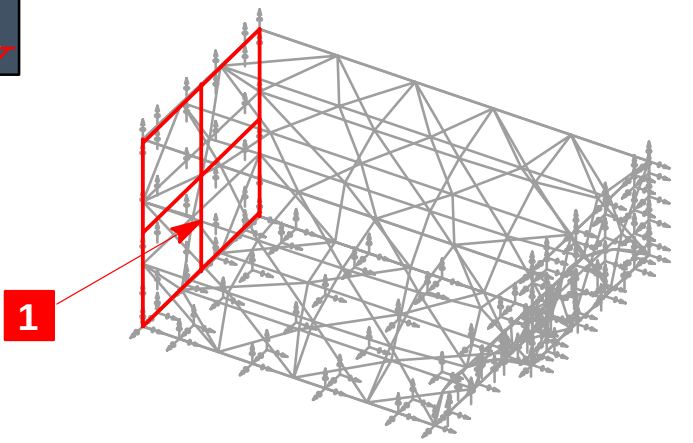


- *X Translation*: *constant*
- *Y Translation*: *free*
- *Z Translation*: *constant*
- *X Rotation*: *free*
- *Y Rotation*: *constant*
- *Z Rotation*: *free*
- *OK*

Why: The vertical face is symmetric, the same as the last case. The horizontal face is anti–symmetric; therefore, all the restraints switch the opposite way from the last case.

What: Restrain the left end in the Y direction.

Hint



1 pick surface

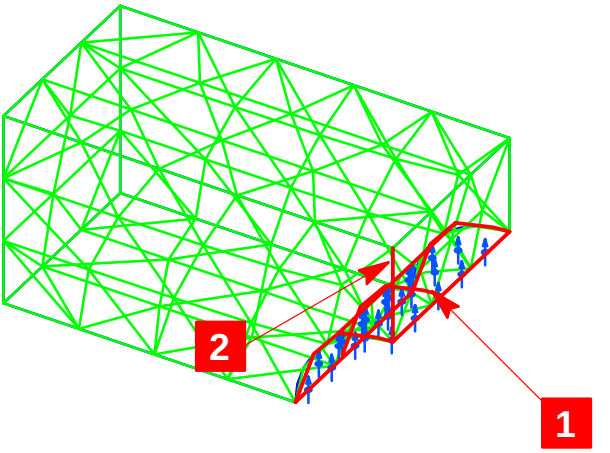
Y Translation: *constant*

(all other directions): *free*

OK

What: Apply a Y force to the cylindrical surface.

Hint



1 pick cylindrical surface

Vector

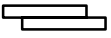
2 pick edge

Total Force

Traction: 500

OK

Recovery Point



File
Save

What: Create a boundary condition set.

Hint

Bending

Restraint Set:

ANTI–SYMMETRY

Load Sets: Bending Force

What: Solve the model.

Hint

Model Solution

Create

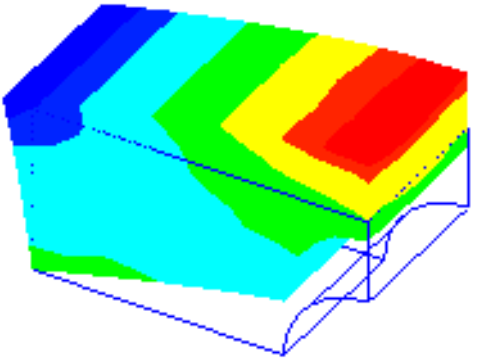
Solution Set form

Boundary Condition Set: Bending

What: Select and display the bending stress and deflection.

Hint

Post Processing



You may want to turn off some of the boundary conditions to clarify the display.

Hint

FE Models...

Tutorial wrap-up

You have completed the Boundary Condition Symmetry tutorial.

What: Delete the FE model, then the part. This part is not used in any other tutorial.

Hint

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 (symmetry)
- Applying Boundary Conditions
  - Applying Restraints
  - Applying Structural Loads

What's next?

After completing the Fundamental Skills tutorials on boundary conditions, you should try the tutorials on displaying results.

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