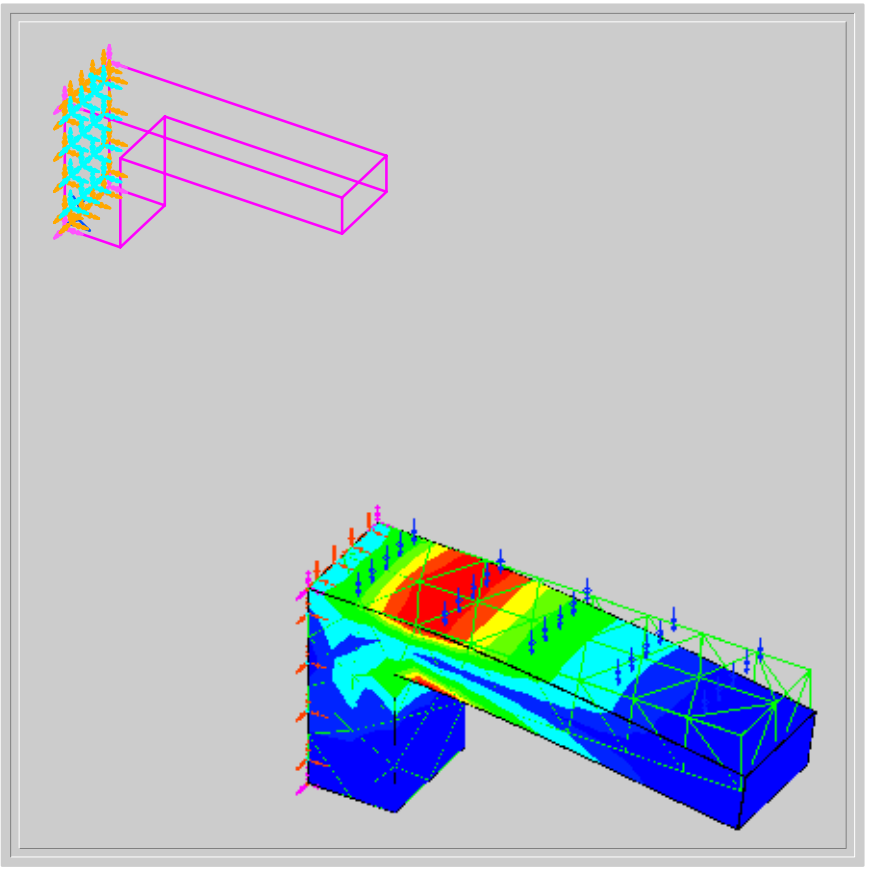


Introduction to Simulation

I-DEAS™ Tutorials: Fundamental Skills

Simulation involves three major steps: Pre-processing (modeling, applying boundary conditions, meshing); solving the model; and post-processing (displaying the results).

Learn how to:

- create a finite element model
- apply boundary conditions
- mesh the FE model
- solve the FE model
- display the results

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

  Quick Tips to Using I-DEAS
  –and–
  Creating Parts

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

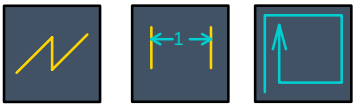
Set your units to mm.

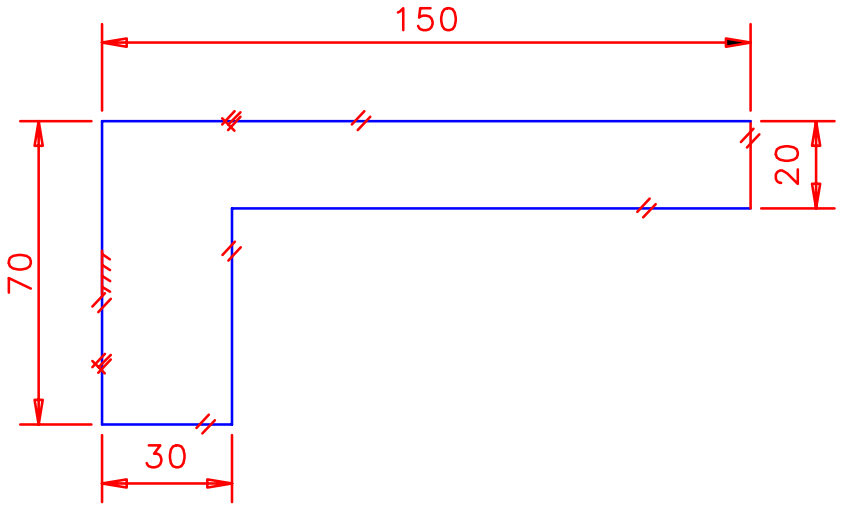
Options
Units

mm (*milli newton*)

What: Sketch this closed shape to the dimensions shown.

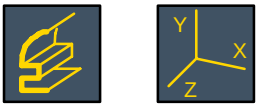
Hint

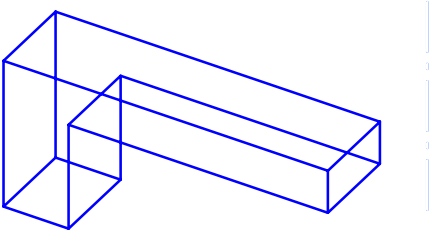




What: Extrude the wireframe a distance of 40mm.

Hint





What: Name the part.

Hint

Name form

Name: Bracket

OK

Save the 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

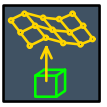
To reopen your model file to the previous save, press Control-Z.

Create a finite element model 1 of 1

What: Create a finite element (FE) model to associate with the part. An FE model is always associated directly to a part.

How: Change the task to *Boundary Conditions*.

Boundary Conditions



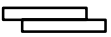
FE Model Create form

FE Model Name: Static Analysis

Geometry Based Analysis Only

OK

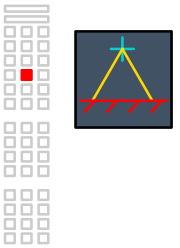
Recovery Point

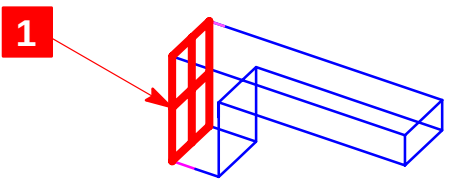


File
Save

What: Fully restrain the rear vertical surface.

How:



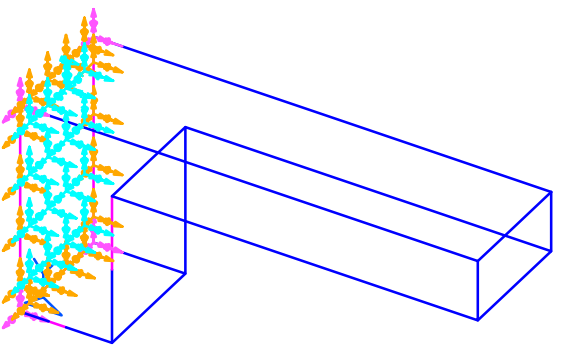


1 pick surface

Done

Displacement Restraint on Surface form

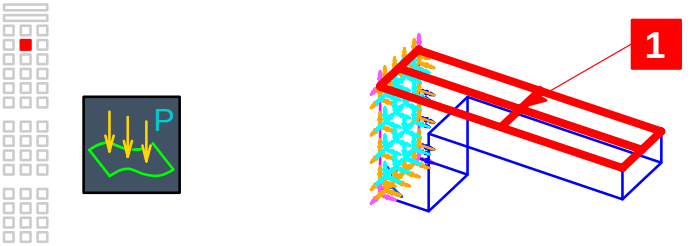
OK



Apply boundary conditions 2 of 3

What: Create a pressure on the top surface.

How:



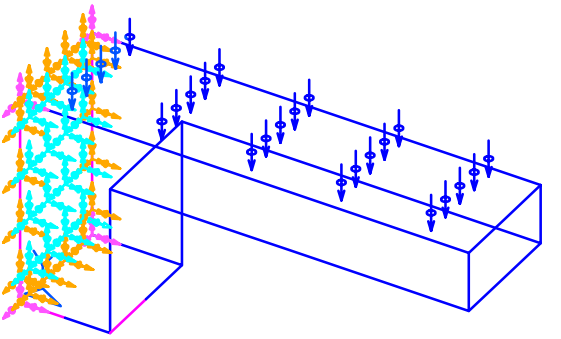
1 pick top surface

Done

Pressure on Surface form

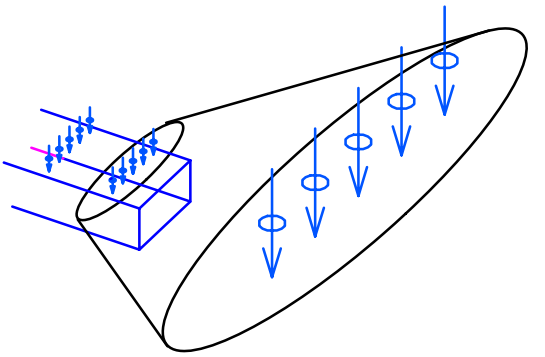
Pressure: 500

OK

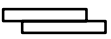


Things to notice

Circles around the arrows mean the pressure is applied to the part geometry.



Recovery Point



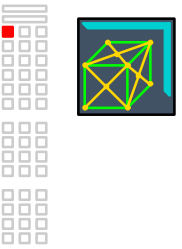
File
Save

I-DEAS Simulation includes manual and automatic meshing tools. Next, a mesh is automatically generated on the entire part.

What: Define the overall element mesh length for the volume.

How: Change the task to *Meshing*.

 *Meshing*



 pick anywhere on part

 *Done*

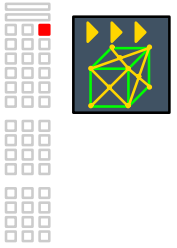
Define Mesh form

 *Element Length*: 20

 *OK*

What: Generate the solid mesh.

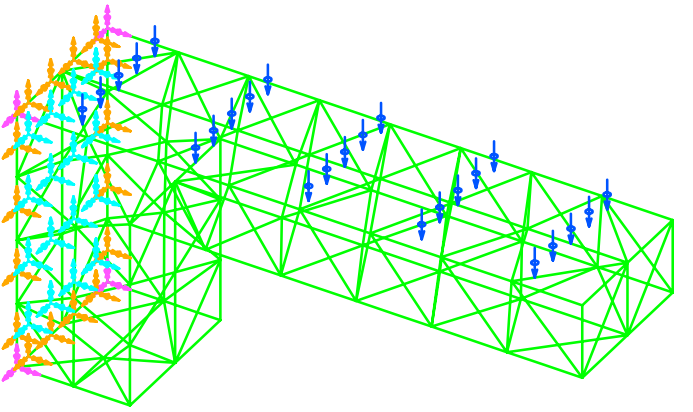
How:



pick anywhere on part

Done

Yes



Things to notice

The elements are parabolic tetrahedral elements. These are more accurate than linear tetrahedral elements for structural analysis.

Recovery Point

File
Save

What: Create a solution set (accepting all defaults) and solve the model using linear statics for deflection and stress.

How:

Model Solution

Manage Solution Sets form

Create...

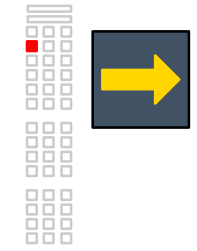
Solution Set form

OK

Dismiss

What: Solve the model.

How:



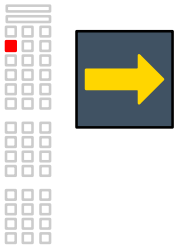
Check *I-DEAS List*.
When the solve is finished, the *I-DEAS List* region displays any warnings or errors.

When the solver finishes, change the task to *Post Processing*.

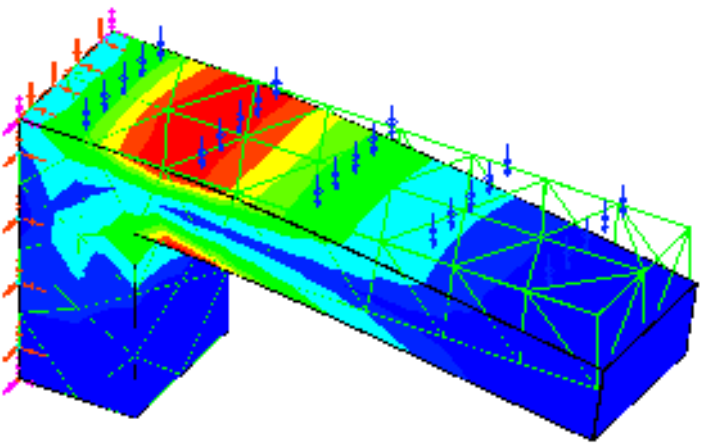
 *Post Processing*

What: Display the default set of displacement and stress results on the deformed geometry.

How:



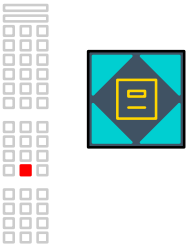
 (the default displays all elements)



You have completed the Introduction to Simulation tutorial.

Delete the part. It is not used in any other tutorial. To delete the part, first delete the finite element model.

How:



Manage form

 Bracket... (double-click)

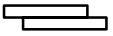
 Static Analysis (select)

 Bracket (select)

 *Dismiss*

See also...

For additional information on many of the concepts covered in this tutorial, see the following in the I-DEAS *Help* facility:

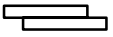

Help, Manuals, Table of Contents

Simulation: Finite Element Modeling User's Guide
- Simulation Overview
- Using Simulation Tools
  - Overview
  - Creating an FE Model on a Part
  - Creating an FE Model from an Assembly
  - Managing Models in Simulation

What's next?

After exiting, choose the Fundamental Skills tutorial that is next in the learning path you are following.

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