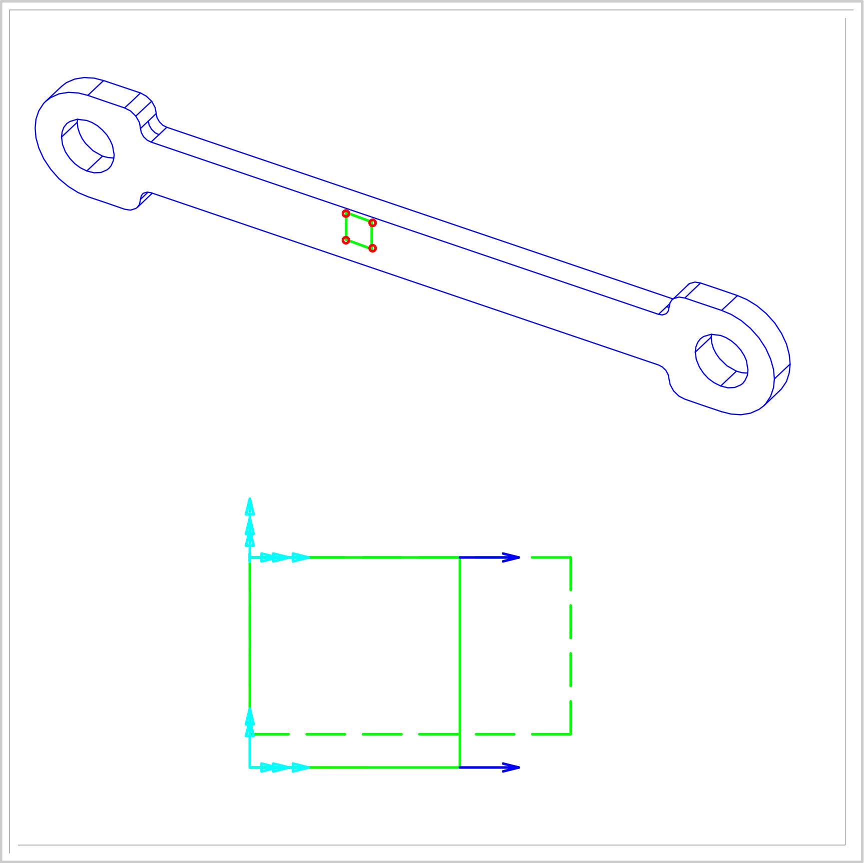


What Is Finite Element Modeling?

I-DEAS™ Tutorials: Fundamental Skills

Although typically you'd use many elements, in this tutorial you'll simulate what happens to a single thin-shell element in the middle of a machine linkage, under a tensile load.

Learn how to:

- create a finite element model
- create material properties
- create physical properties
- create nodes
- create an element
- apply boundary conditions
- solve the model
- interpret the results

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

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

This tutorial assumes you have a background in the concepts of stress as taught in a typical college course in strength of materials.

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*)

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.

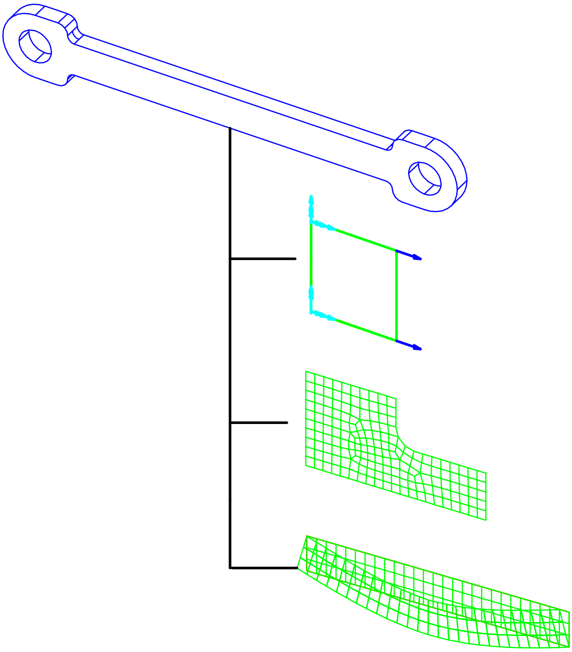
Before you start an analysis, you should think about the type of solution you want. Which solution type you choose depends on loading and other factors. Key factors include:

- how loads vary with time, or the duration of the load relative to the period of natural frequency
- whether the expected result is linear

In this example, you'll use a linear static solution.

You start by creating a finite element (FE) model. An FE model is always associated with a part.

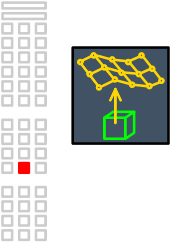
In the model file, there may be more than one finite element model associated with the same part, with different meshes.



What: Switch to the *Meshing* task and create an FE model.

How:

Meshing



FE Model Create form

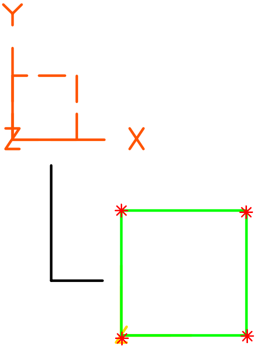
Part or Assembly: (any unique name)

FE Model Name: Tensile Model

OK

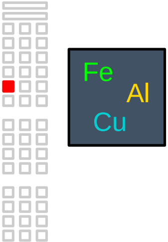
Things to notice

Ignore the warning message if you get one. When you created the FE model above and entered the name of a nonexisting part, a null part was created. A null part contains no surfaces. It is displayed in Simulation as a part coordinate system.



What: Create a material for the analysis and define the modulus of elasticity and Poisson’s ratio.

How:



Materials form

Quick Create...

Quick Create form

Material Label & Name: Link Material (enter name only, in second field)

Property: Modulus of Elasticity

2E8

Modify Value

Continued on next page...

Property: Poisson’s Ratio

0.3

Modify Value

OK

Do not exit from the Materials form yet.

Things to notice

The value for Poisson’s ratio of 0.3 means that if the material is stretched one unit in the X-direction, it will shrink 30% in the Y- and Z-directions. Later in this tutorial you’ll check the displacements to see if this holds true.

Create material properties

What: Examine the new material to make sure you entered the correct values.

How:

Materials form

Link Material

Examine...

Examine form

Review the property values in the table.

LINK MATERIAL
MODULUS OF ELASTICITY
2.0000E+08
POISSONS RATIO
3.L0000E–01

Dismiss

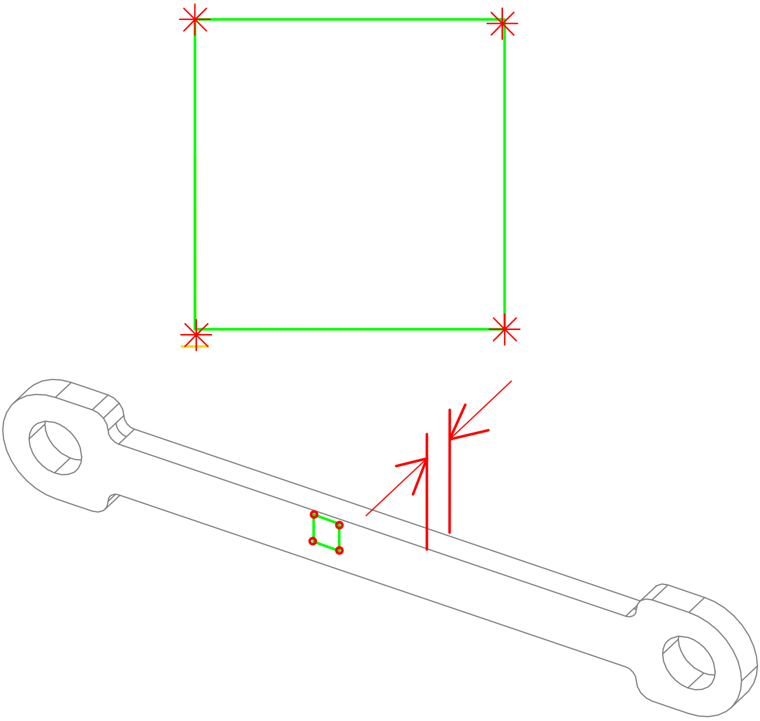
Materials form

OK

Things to notice

This material definition is stored in your model file and can be referenced by any FE model you create in the same model file.

The thin shell element you'll create will be defined by four nodes at its corners. The physical thickness of thin shell elements is defined by a separate physical property table.

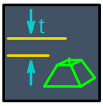


Things to notice

The link part is shown here only to indicate the thickness of the element. Since you are creating nodes and elements manually, you won't have a part displayed.

What: Create a physical property table that defines a shell thickness of 5mm.

How:



Thin Shell

Check *I-DEAS Prompt*.

physical property name: Link Thickness

No

Directory

Continued on next page...

TK THICKNESS [4V]

There are four values of thickness (4V)—one value at each corner.

Check *I-DEAS Prompt*.

1st value for thickness: 5

<Return> – for rest of the prompts

Warning!

If you enter more than one value, all four values must be non-zero. To enter a uniform thickness at each corner, you need to enter only the first value.

Done

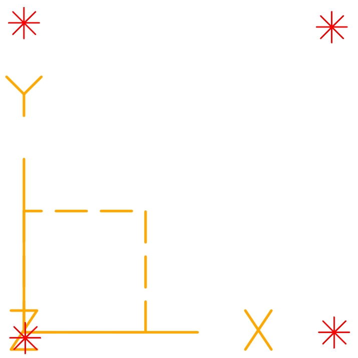
Recovery Point

File

Save

In this example, you'll create four nodes in the finite element model.

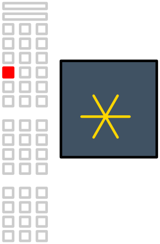
The displacements of nodes are the variables for which the finite element model is solved. Each node has three potential displacements and three potential rotations, or a total of six degrees of freedom (DOF).



The four nodes in the model contain 24 DOF to be solved.

What: Create four nodes in a 10mm square on the XY plane.

How:



Node form

OK

Check *I-DEAS Prompt*.

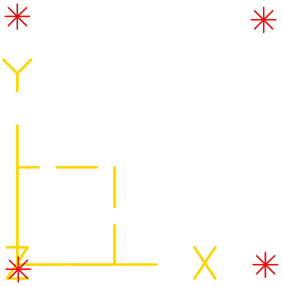
Node 1 location: 0 0

Node 2 location: 10 0

Node 3 location: 0 10

Node 4 location: 10 10

Done

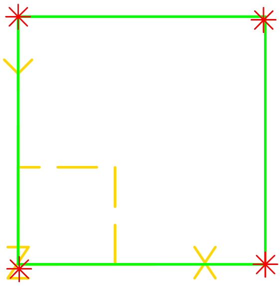


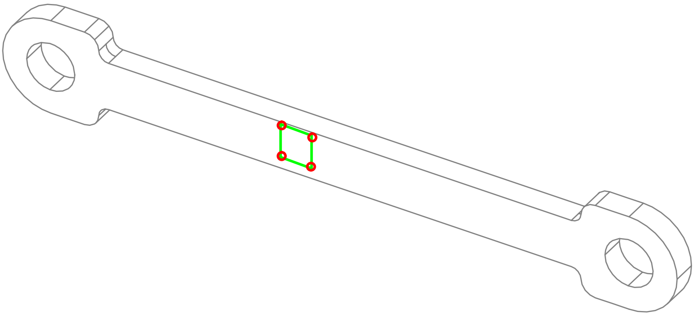
Adjust the display to get a better view of the nodes. You could use *Zoom All* and F2.

Recovery Point

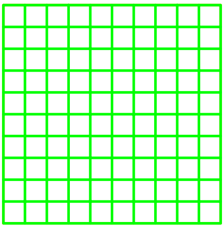
File
Save

Elements are displayed by lines connecting the nodes.





Why: In this example you'll use only one element to understand how nodes and elements behave. The computed displacements and stresses will be correct only for simple uniform loads. In most cases, you'll have to create many smaller elements to converge on a correct result.

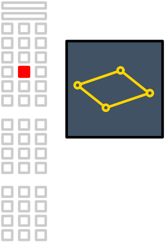


To create elements you'll:

- specify the type of elements (1D, 2D, 3D)
- define material and physical properties
- pick the nodes

What: Create one thin-shell element to connect the four nodes.

How:



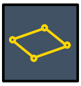
Element form

Make sure the following are selected on the form:

 *2D*

 *Element Family: Thin Shell*

 *Element Type:*



 Don’t close the Element form yet.

What: Assign material and physical properties to the elements.

How:

Element form

Material Selection...

Materials form

Link Material

OK

Element form

Physical Property:

?

Physical Property form

Link Thickness

OK

Element form

OK

Why: The default is to use the material stored with the part that's associated with the finite element model. Since you're not using a meaningful part here, you manually selected the material table you created earlier.

What: Pick the four nodes with which to create the thin-shell element. In a typical problem, you would use more automatic methods to create elements.

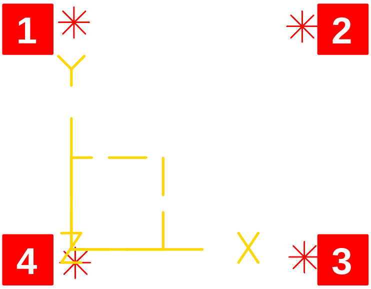
How:

1

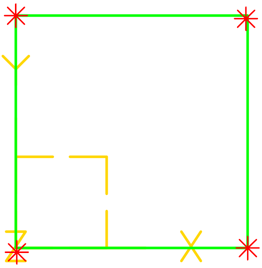
2 shift-pick

3 shift-pick

4 shift-pick



(Done)

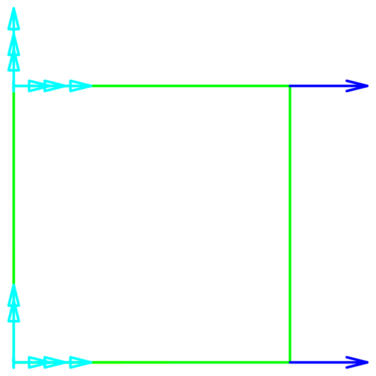


If the element looks wrong, you've picked the nodes in the wrong order. Press Control-Z and repeat this step before saving your model file.

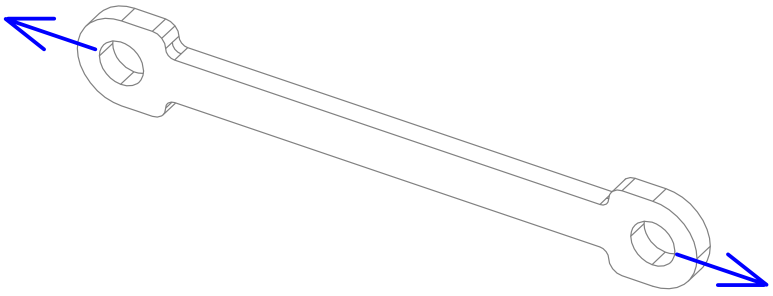
Recovery Point

File
Save

At each degree of freedom (DOF), you can either apply restraints to the degree of freedom, or you can apply a force.



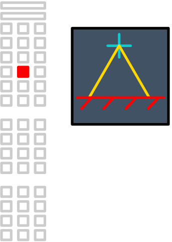
You must apply enough restraints to hold the model as a rigid body in space. Even though the linkage has equal and opposite forces on the two ends, you will restrain one end, and apply a force on the other.

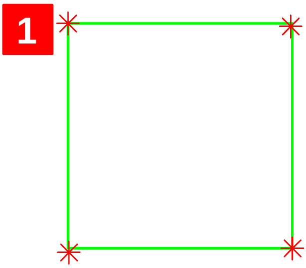


What: Restrain all six degrees of freedom of the node indicated.

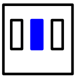
How:

 *Boundary Conditions*





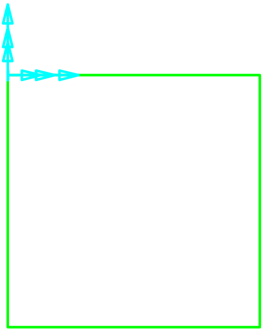
1 pick node

 *Done*

Displacement Restraint on Node form

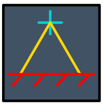
 *Clamp*

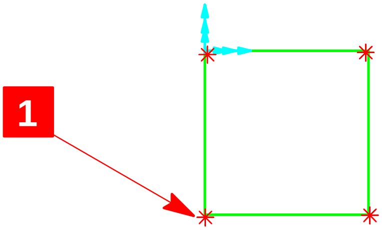
 *OK*



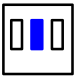
What: Restrain the indicated node, and leave only the Y translation free.

How:





1



Displacement Restraint on Node form

 *Specified*

 *Specify Restraint...*

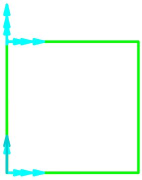
Specified Restraint on Node form

 *Y Translation: Free*

 *OK*

Displacement Restraint on Node form

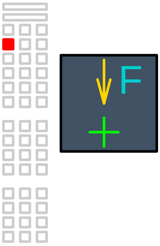
 *OK*

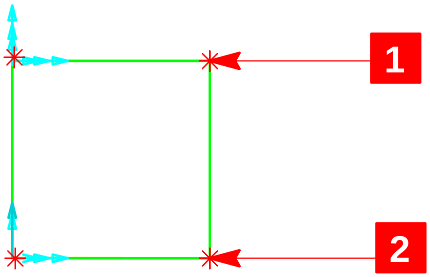


Why: The Y-displacement is free so the element can contract in the Y-direction as it stretches in the X-direction.

What: Apply a force in the X-direction at two nodes.

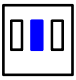
How:





1

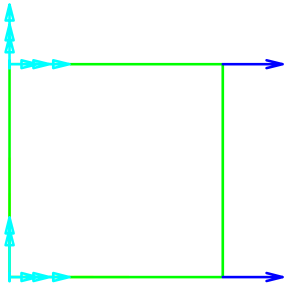
2 shift-pick



Force on Node form

X Force: 100000

OK



Recovery Point

File

Save

Things to notice

The total force on the element is the sum of the two nodal forces.

Solve the model

The most common type of solution is linear statics, in which the software simulates the response of the model to a steady-state static load.

The software will solve the matrix equation:

{F} = [K] * {d}

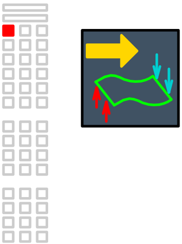
At each degree of freedom, either a displacement is calculated at the given forces, or reaction forces can be calculated where displacements are restrained.

What: Create a solution set to define:

- the type of solution to perform
- the boundary condition set to use
- the types of output to calculate and store
- other options that affect the solution

How:

Model Solution



Manage Solution Sets form

Create...

Solution Set form

Name: Uniform Static Load

Don't close the form; continue on the next page.

What: Request output types of displacement and stress. List and store the output during the solve.

How:

Solution Set form

Output Selection...

Output Selection form

Output Type: *Displacements*

Store/List: *Store and List*

Output Type: *Stress*

Store/List: *Store and List*

OK

Solution Set form

OK

Manage Solution Sets form

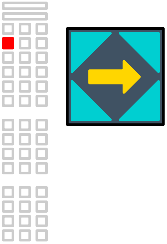
Dismiss

Recovery Point

File
Save

What: Define the location where temporary files and output files are stored. Then solve the model.

How:



Solve form

Output file: results1.lis

Solve

Things to notice

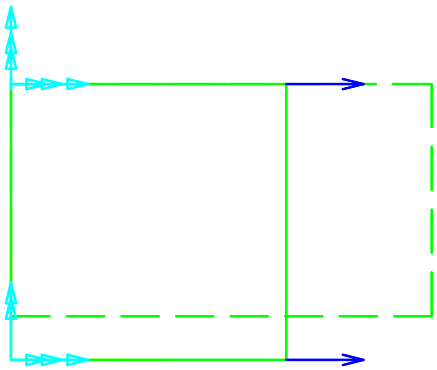
The windows will temporarily disappear when the solver starts. As the solve progresses, status messages are displayed in the *I-DEAS List* window. After the solve, any errors or warnings are listed in the *I-DEAS List* window.

During the solve, the file results1.lis is created to store details of the analysis. You may want to examine this file later, then delete it.

When reviewing results you must ask:

- Are the results from the model correct?
- What do the results mean?

Some simple hand calculations can increase your confidence that the results are correct.



What: Select the deformation results to examine.

How:

Post Processing

Results Selection form

Display Results: *Clear*

Displacement_1 (select)

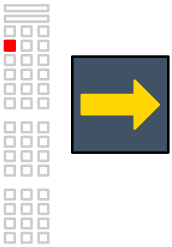
Deformation Results

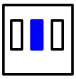
OK

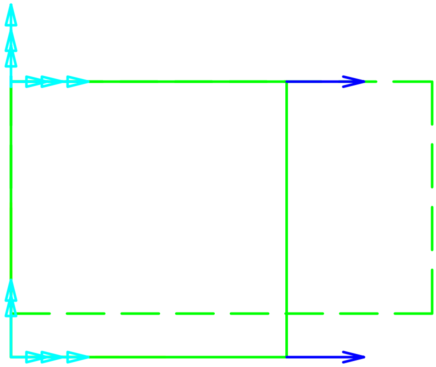
Why: Deformation results will be shown as displaced nodes. Display results are shown in color. By clearing the display results, you can see the displacements more clearly.

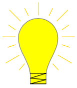
What: Display the deformation.

How:



 (to select all elements)





It’s good practice to look at displacements first because they are more intuitive than stresses. Make sure the displacements look reasonable before you try to interpret stresses.

Interpret the results

What: Perform a hand calculation to check the displacements.

How:

The displacement in the X-direction should be:

X = PL/AE

P = 100,000 * 2
L = 10
A = 10 * 5
E = 2E8

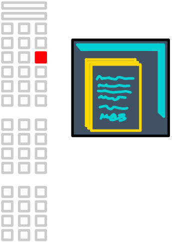
X = 0.0002

Things to notice
This displacement value is small, verifying that the assumption of using linear static analysis is valid.

Most workstations have a desktop calculator you can use to try this calculation.

What: Generate a report of the displacements to check the values.

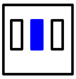
How:



Report Writer form

 *Output Similar To*: *Deformed Display*

 *Generate Report*



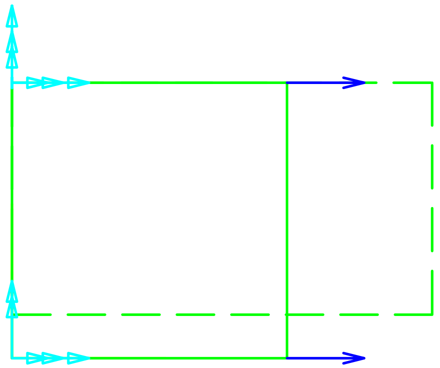
Check *I-DEAS List*.
Scroll back in the *I-DEAS List* window.
X-displacements at nodes 2 and 4 should match your hand calculation. If they don't match, you may have made a mistake in entering boundary conditions, material properties, or physical properties.

What: Check Poisson's ratio.

How:
Using the values shown in the *I-DEAS List* window, divide the Y-translation by the X-translation at node 2:

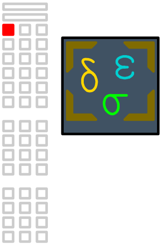
Y-translation / X-translation = 0.3

Things to notice
In the plot of deflection, the element shrinks in the Y-direction as it stretches in the X-direction. Under a uniaxial load, this value should match the value you entered for Poisson's ratio (0.3).

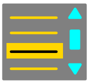


What: Display the stress results.

How:



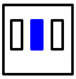
Results Selection form

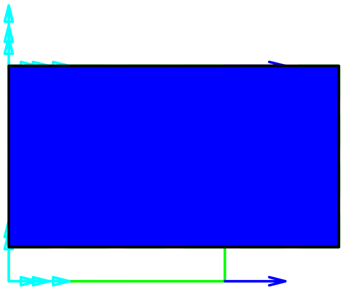
 Stress_2 (select)

 *Display Results*

 *Component: Maximum Principal*

 *OK*





Things to notice

Due to uniform loading, all values for the element are the same. The fact that the stresses are less than the yield stress verifies that using linear static analysis is a good assumption.

What: Perform a hand calculation to check the stress.

How:

Stress = P/A

P = 100,000 * 2
A = 10 * 5

Stress = 4000

Things to notice

If 4000 is not the value on your plot, you may have incorrectly entered the nodal coordinates, physical properties, or forces.

In this simple case, one element gives the correct answer since the stress is uniform. In most models where the stress is not uniform, the elements can only approximate the true values, so you'll require more elements to obtain a correct answer.

You may want to examine the output.lis file on the operating system. It will list the same results you displayed.

Tutorial wrap-up

You have completed the What Is Finite Element Modeling tutorial.

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 Overview
- Using Simulation Tools

Simulation: Model Solution/Optimization User's Guide
- Using the Solvers
  - Using Linear Statics Analysis

What's next?

You know that one element is not enough to model most problems, but how many should you use? Here we used thin-shell elements, but when should you use other types of elements? These questions and others will be discussed in other tutorials.

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