

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

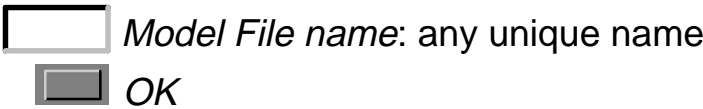
- Introduction to Simulation

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

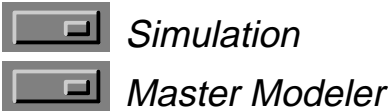
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



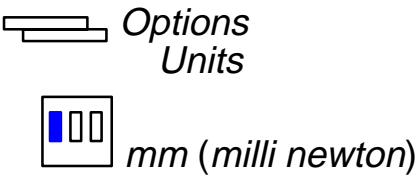
Open Model File form



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



Set your units to mm.



Save your model file.



Warning!

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



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.

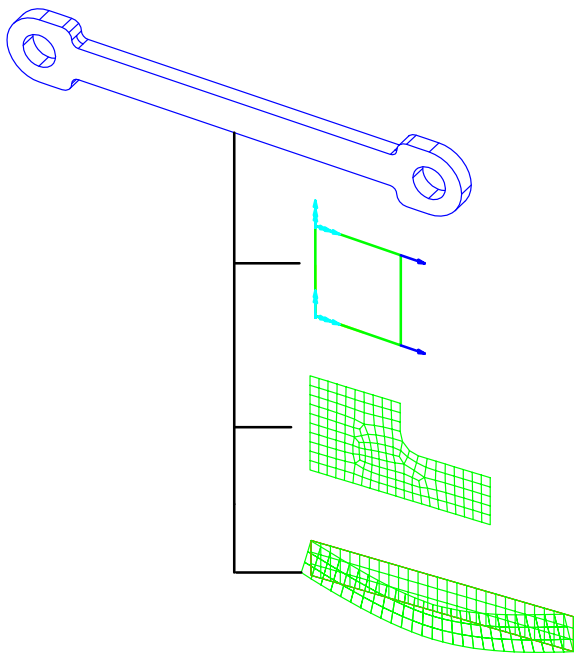
Before you start an analysis, you should think about the type of solution you want. Which solution type you choose depends on loading and other factors. Key factors include:

- 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.

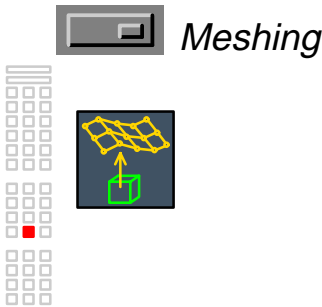
You start by creating a finite element (FE) model. An FE model is always associated with a part.

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:



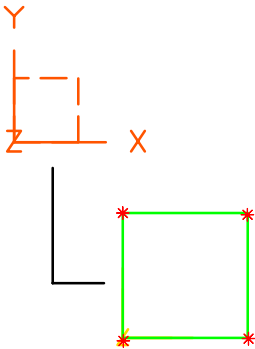
FE Model Create form

Part or Assembly: (any unique name)

FE Model Name: Tensile Model

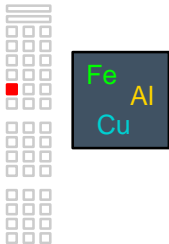
Things to notice

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 form

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



Property: Modulus of Elasticity

2E8



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.

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
3.0000E-01



Dismiss

Materials form

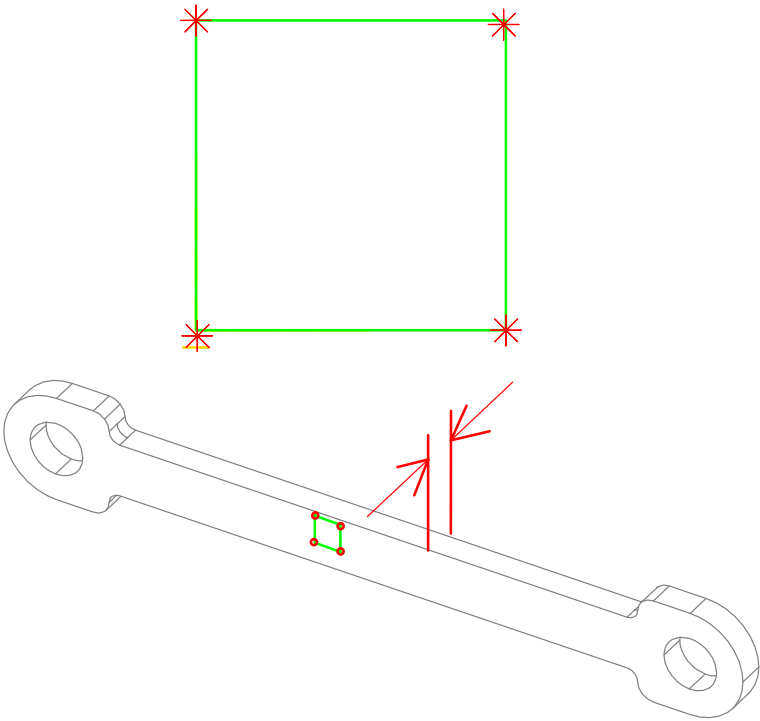


OK

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.

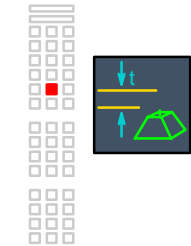


Things to notice

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

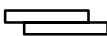
Warning!

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

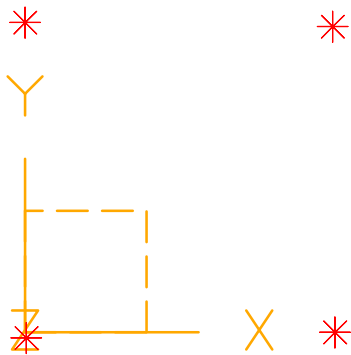


File

Save

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

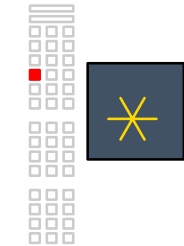
The displacements of nodes are the variables for which the finite element model is solved. Each node has three potential displacements and three potential rotations, or a total of six degrees of freedom (DOF).



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



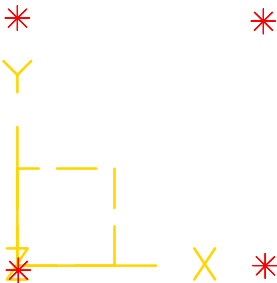
 **Check I-DEAS Prompt.**

Node 1 location: 0 0

Node 2 location: 10 0

Node 3 location: 0 10

Node 4 location: 10 10

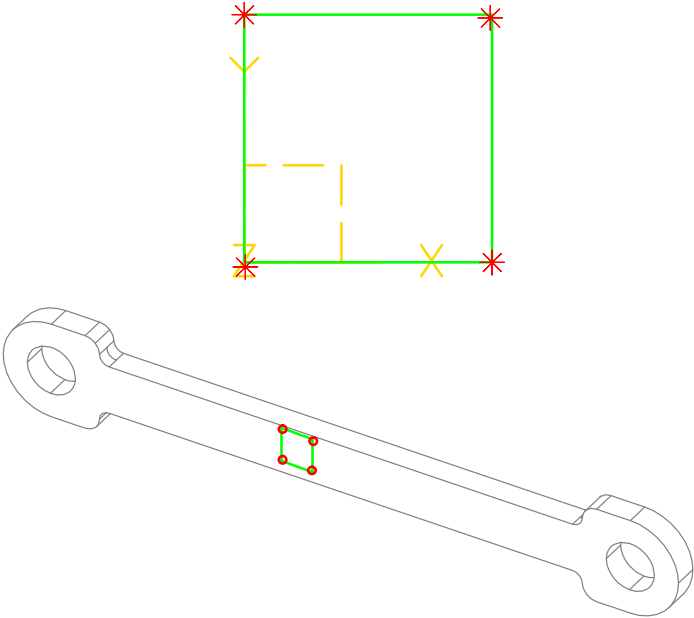


Adjust the display to get a better view of the nodes. You could use *Zoom All* and F2.

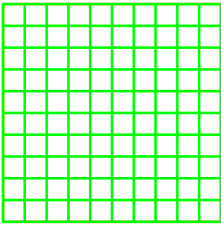
Recovery Point



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.

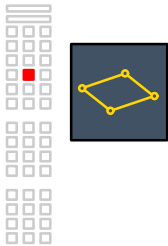


To create elements you'll:

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

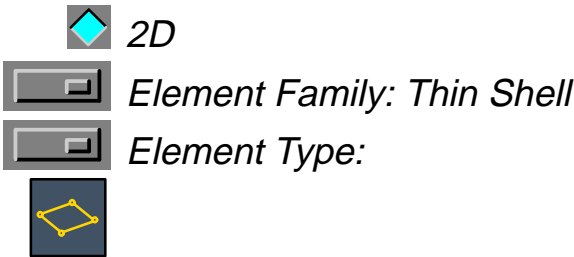
What: Create one thin-shell element to connect the four nodes.

How:



Element form

Make sure the following are selected on the form:



 Don't close the Element form yet.

What: Assign material and physical properties to the elements.

How:

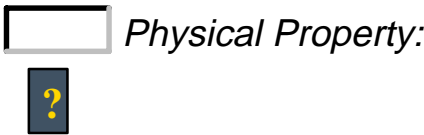
Element form



Materials form



Element form



Physical Property form



Element form

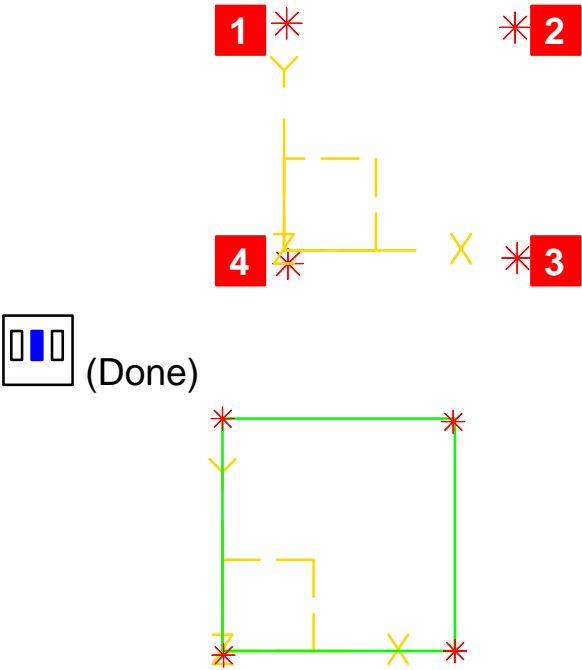



Why: The default is to use the material stored with the part that's associated with the finite element model. Since you're not using a meaningful part here, you manually selected the material table you created earlier.

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

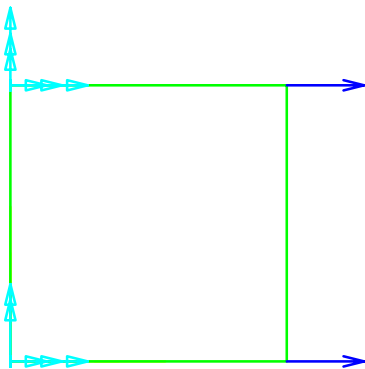


 If the element looks wrong, you've picked the nodes in the wrong order. Press Control-Z and repeat this step before saving your model file.

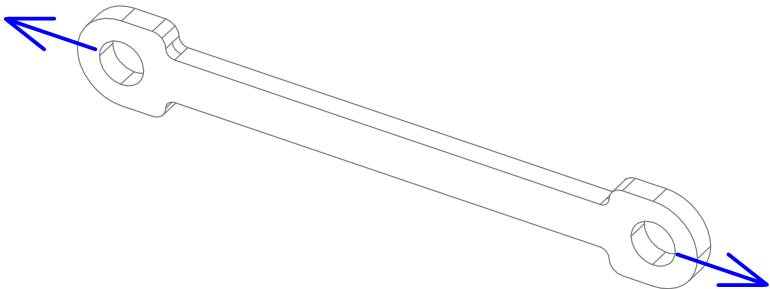
Recovery Point

 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.

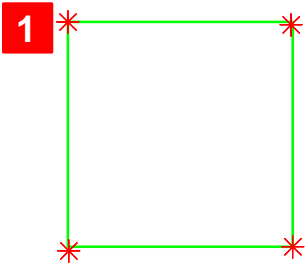


What: Restrain all six degrees of freedom of the node indicated.

How:



Boundary Conditions



1 pick node



Done

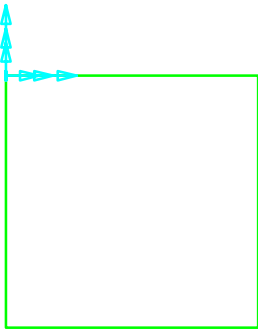
Displacement Restraint on Node form



Clamp

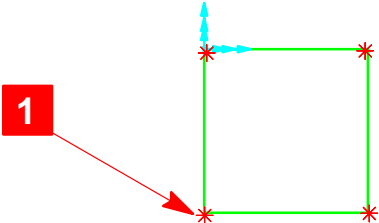
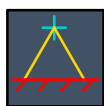


OK



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

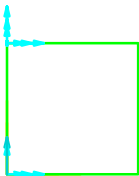


OK

Displacement Restraint on Node form



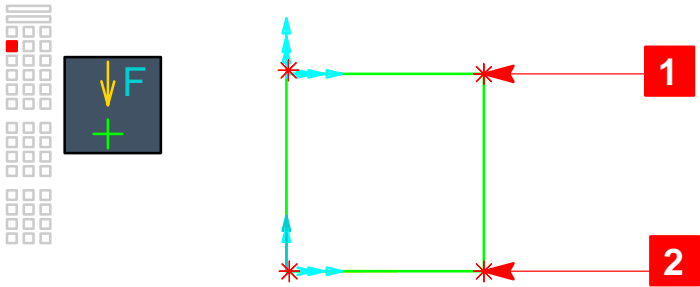
OK



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:



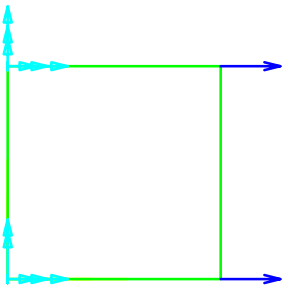
1

2 shift-pick



Force on Node form

X Force: 100000



Recovery Point

Things to notice

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

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

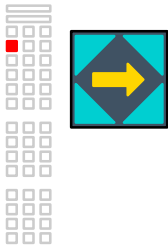
 *Dismiss*

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

Things to notice

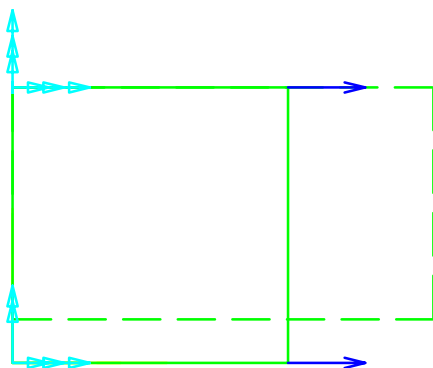
The windows will temporarily disappear when the solver starts. As the solve progresses, status messages are displayed in the *I-DEAS List* window. After the solve, any errors or warnings are listed in the *I-DEAS List* window.

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:

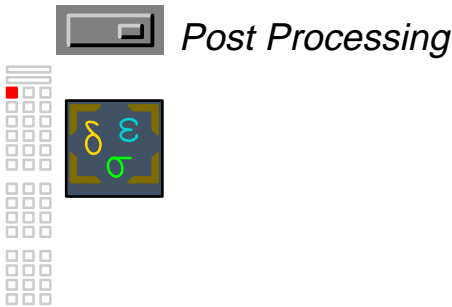
- Are the results from the model correct?
- What do the results mean?

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

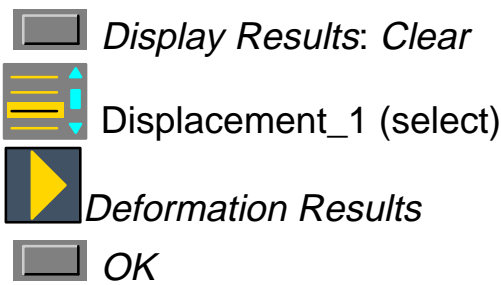


What: Select the deformation results to examine.

How:



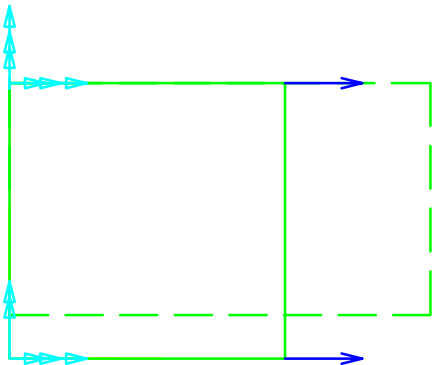
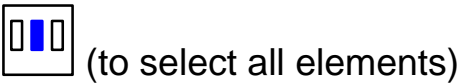
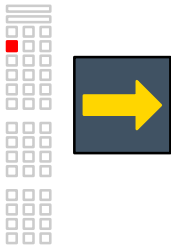
Results Selection form




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:



 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.

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

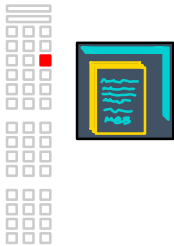
This displacement value is small, verifying that the assumption of using linear static analysis is valid.



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.

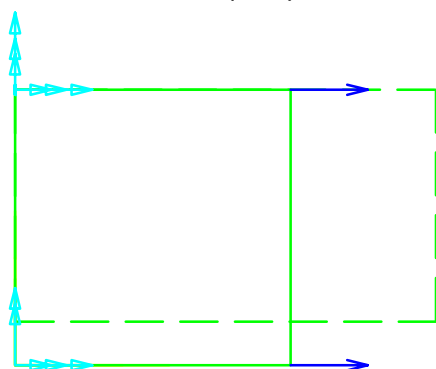
How:

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

$$\text{Y-translation} / \text{X-translation} = 0.3$$

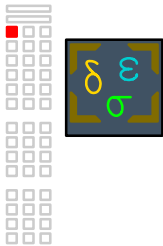
Things to notice

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

This online information content, is licensed to the user for the period set forth in the applicable license agreement, subject to termination of the license by Structural Dynamics Research Corporation (SDRC®) at any time, and at all times remains the intellectual property of SDRC. The information contained herein is confidential to SDRC and shall not be copied or reproduced in any form whatsoever, nor be disclosed to anyone other than an authorized representative of the user's employer who is contractually obligated not to disclose same, without express prior written consent of SDRC. The user of this tutorial and the computer program(s) referred to herein retains full control over and is solely responsible for the mechanical design of the user's equipment, machinery, systems, and products. SDRC makes no warranties of any kind, including the warranty of merchantability or fitness for a particular purpose in respect to the equipment, machinery, systems, and products derived or resulting hereunder, and the user assumes all risks and liability for results obtained by the manufacturing, use or implementation of the computer program(s) described herein, whether used singly or in combination with other designs or products. SDRC shall not be liable for any special or consequential damages. SDRC makes no warranty that the equipment, machinery, systems, and products derived or resulting hereunder will not infringe the claims of domestic or foreign patents and further does not warrant against infringement by reason of the use thereof in combination with other design, products, or materials or in the operation of any process. Users shall protect, indemnify and hold harmless SDRC of and from any loss, cost, damage or expense arising from any claim that is in any way associated with the computer program(s) described in this tutorial. Data presented in examples do not necessarily reflect actual test results and should not be used as design criteria.

By acceptance of I-DEAS Master Series, the user agrees to the above conditions and further agrees that this intellectual property will not be exported (or reexported from a country of installation), directly or indirectly, separately or as part of a system, without user or user's employer, at its own cost, first obtaining all licenses from the United States Department of Commerce and any other appropriate agency of the United States government as may be required by law.

© Structural Dynamics Research Corporation 1979, 1980, 1983, 1984, 1986, 1988, 1990, 1991, 1992, 1993, 1994, 1995, 1996, 1997, 1998

© Maya Heat Transfer 1990, 1991, 1992, 1993, 1994, 1995, 1996, 1997, 1998

All rights reserved. No part of this work may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, or by any information storage or retrieval system without permission in writing from SDRC.

Federal Acquisitions: Commercial Computer Software
Use governed by terms of SDRC's Software License and Service Agreement.

SDRC has worked to verify the accuracy of the information contained in this manual as of its publication date; however, such information is subject to change without notice and SDRC is not responsible for any errors that may occur in this document.

This software is a Licensed Product of and distributed by SDRC and may only be used according to the terms of that license on the system identified in the License Agreement.

SDRC and SDRC I-DEAS are registered trademarks of Structural Dynamics Research Corporation.

The following are trademarks of Structural Dynamics Research Corporation

I-DEAS, I-DEAS Master Series

All other trademarks or registered trademarks belong to their respective holders. All questions or requests should be addressed to:

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