

Which Element Type Should I Use?

I-DEAS™ Tutorials: Fundamental Skills

In this tutorial you'll investigate the differences among element types and learn why you might use one element type rather than another.

You will use four different models to analyze a machine linkage under a simple bending load.

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

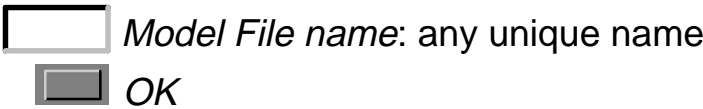
Creating Parts

- Introduction to Simulation
- What is Finite Element Modeling?
- Free Meshing

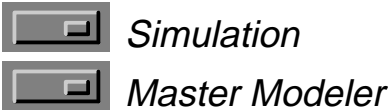
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



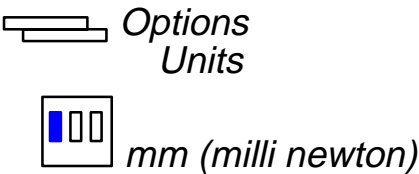
Open Model File form



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

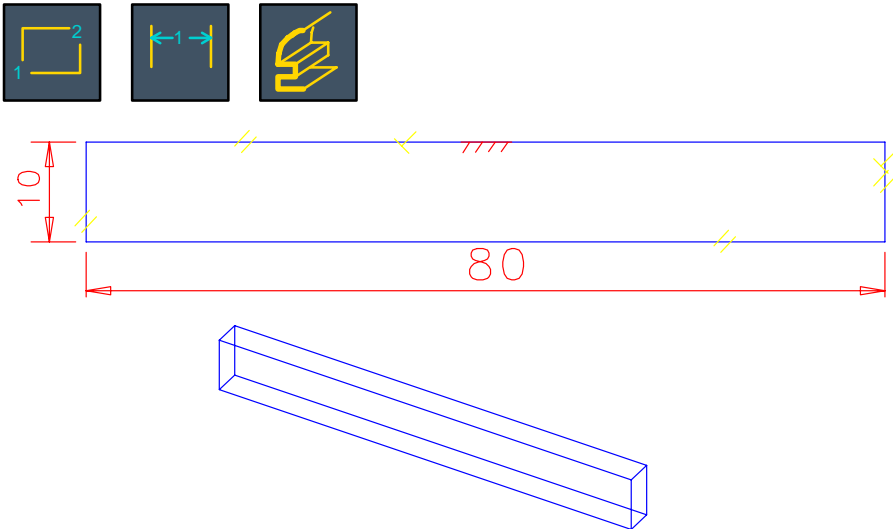


Set your units to millimeters.



What: Sketch a rectangle to the dimensions shown and extrude to 5mm.

Hint



What: Name the part.

Hint



Name: Four Beams

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:

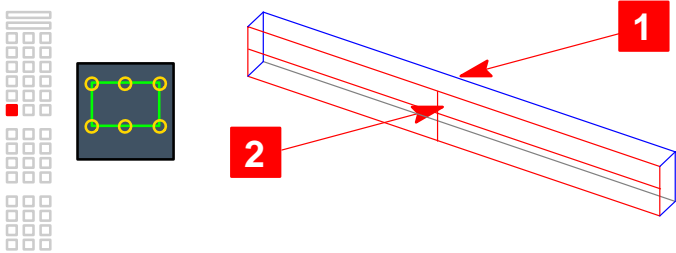
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

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

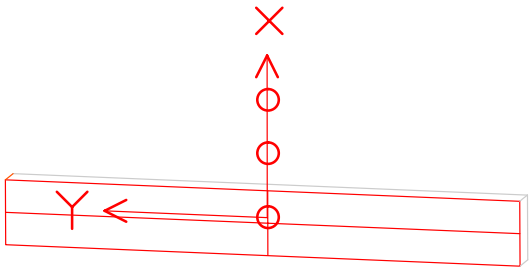
What: Make a pattern of four occurrences of the part.

How:



1

2 pick the front face
(Accept if necessary)



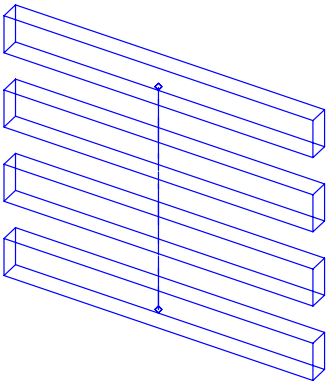
Rectangular Pattern form

Number along X: 4

Distance between: 20

Number along Y: 1

Result



Things to notice

If your part doesn't look like the part shown here, use Control-Z to go back to the original beam and try again. Switch the values in the X and Y fields if your pattern orientation is different.

What: Create an FE model associated with the part.

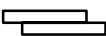
Hint



Meshing



Recovery Point

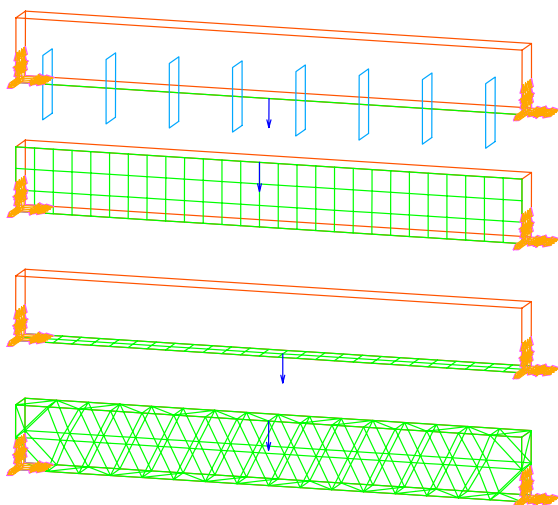


*File
Save*

Before you can select an element type, you must decide which type of analysis you will perform. In this tutorial, you will use linear statics to analyze a bending load.

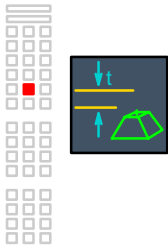
For this analysis, beam elements, thin-shell elements, or solid elements all could be used. Which element type should you use?

This tutorial shows the use of different element types in one model to let you compare the results of each type.



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

How:



Thin Shell

physical prop name: Shell 5 mm



No



Directory



TK THICKNESS[4V]

1st value for thickness: 5

<Return> (for remaining prompts)



Done

Remember

Accepting the defaults sets each thickness equal to the first thickness value entered.

What: Create a second physical property table with a thickness of 10mm.

Hint



Thin Shell

physical prop name: Shell 10 mm



No



Directory



TK THICKNESS[4V]

1st value for thickness: 10

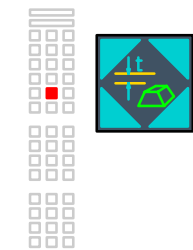
<Return> (for remaining prompts)



Done

What: Verify that you correctly created both physical property tables defining shell element thickness of 5mm and 10mm.

How:



Physical Property Tables form



Shell 5 mm (select)
Shell 10 mm (shift-pick)



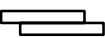
Check *I-DEAS List*.

Scroll back in the *I-DEAS List* window to find the two tables. Note the 4 thickness values for each, the first of which should be 5 or 10mm, the rest zero.



Dismiss

Recovery Point



File
Save

What: Restrain the lower right edge of each volume, allowing Z rotation.

How:



Boundary Conditions



Filter...

Selection Filter form



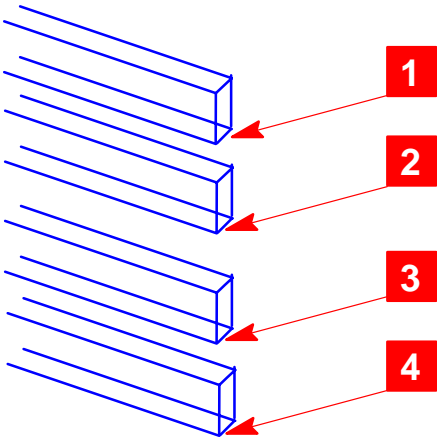
Edge...



Pick Only



shift-pick the 4 edges



Done

Displacement Restraint on Edge form



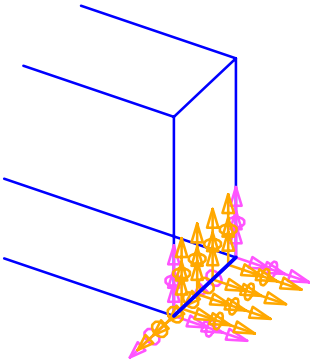
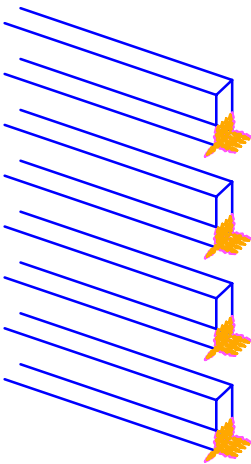
Z Rotation: Free



OK

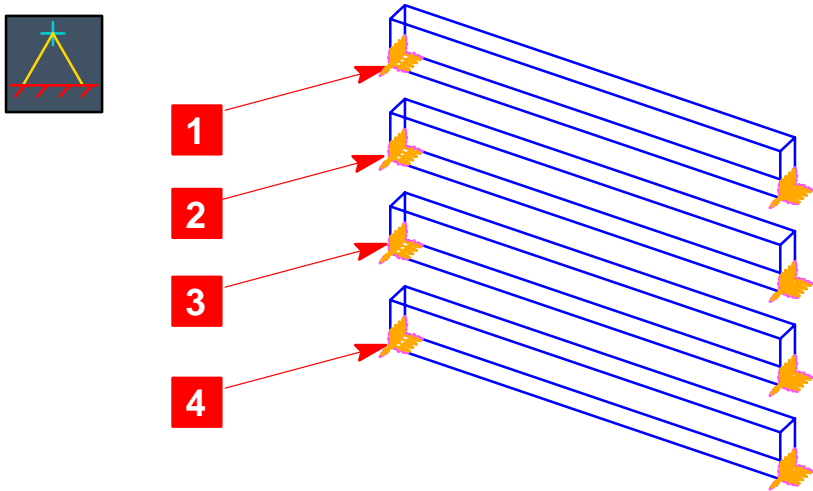


Result



What: Restrain the lower left edge of each volume, allowing X translation and Z rotation.

Hint



Displacement Restraint on Edge form

- ☐ X Translation: Free
- ☐ Z Rotation: Free
- ☐ OK

Why: There is some deflection in the X direction when bending occurs. If you don't allow X sliding translation on one end, you over-restrain the model and the comparisons are not valid.

Recovery Point

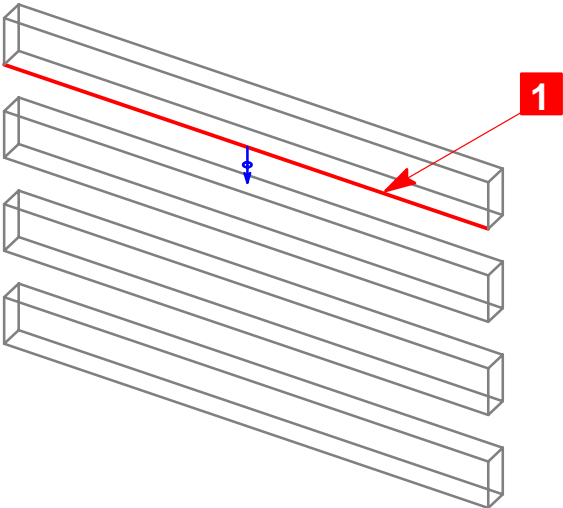


What: Apply a 1000mN force in the $-Y$ direction at the center of the bottom-front edge of the top beam.

How:



Deselect All



1



Locations on Edge



Key In



Done

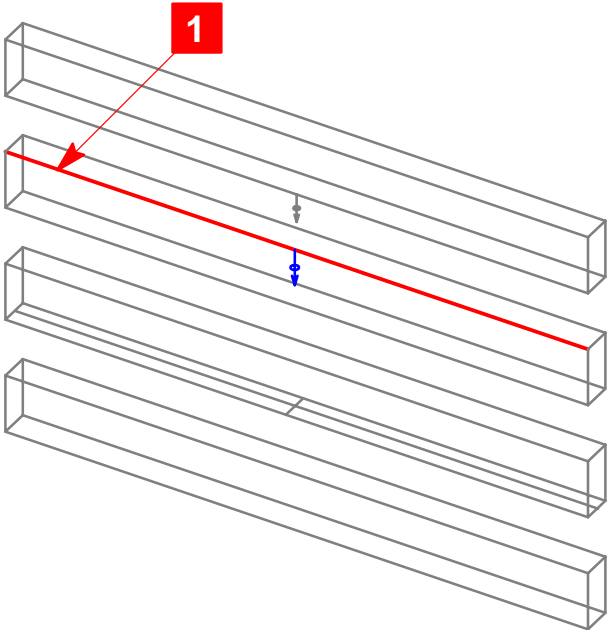
Force on Vertex/Location on Geometry form

Y Force: -1000

OK

What: Apply a 1000mN force in the $-Y$ direction at the center of the top-front edge of the second beam.

Hint



1



Locations on Edge



Key In



Done

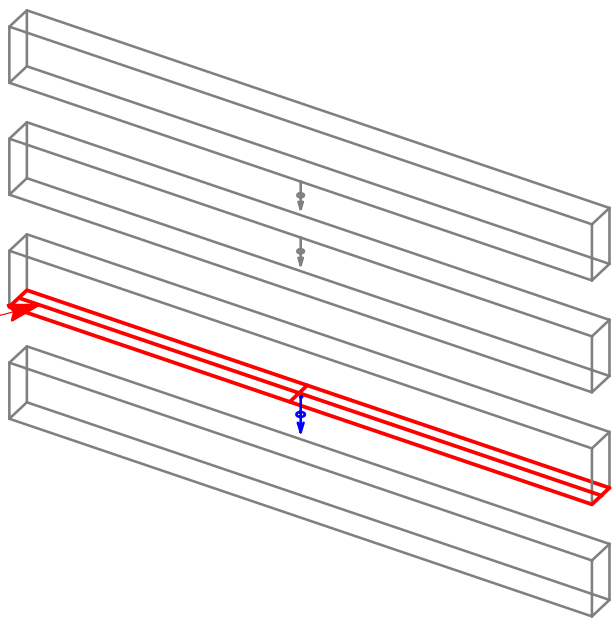
Y Force: -1000



OK

What: Apply a 1000mN force in the $-Y$ direction at the center of the bottom face of the third beam.

Hint



1



Locations on Surface



Key In



Done

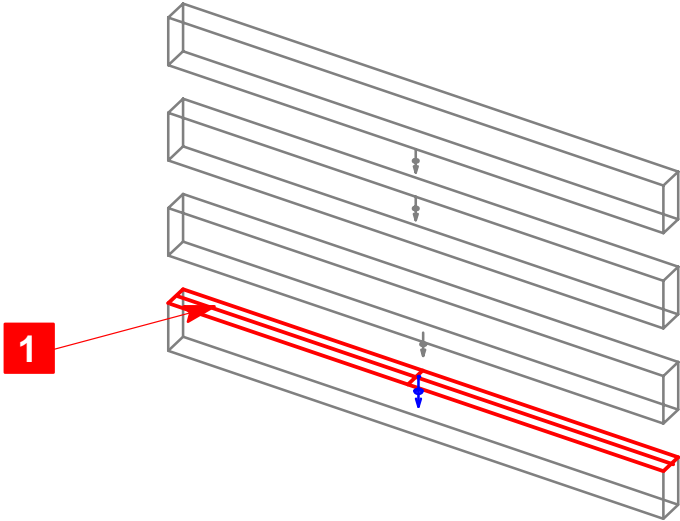
Y Force: -1000



OK

What: Apply a 1000mN force in the $-Y$ direction at the center of the top face of the bottom beam.

Hint



1



Locations on Surface



Key In



Done

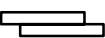


Y Force: -1000




OK

Recovery Point



*File
Save*

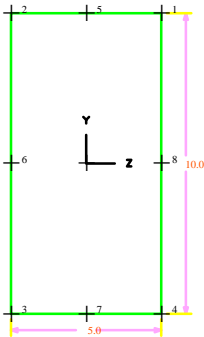
 If you don't have a license for the *Beam Sections* task, **skip to page 20**.

What: Create and store a beam cross section.

How:



Beam Sections



Check I-DEAS Prompt.



base: 5



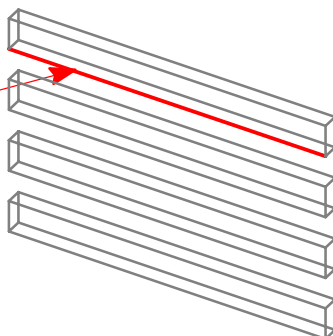
height: 10



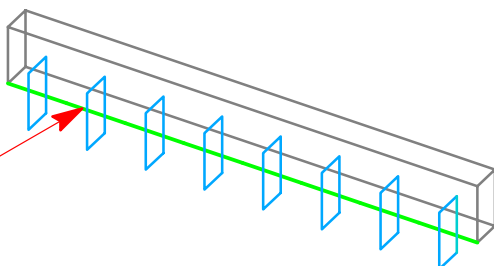
Yes



Hint



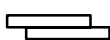
Element Length: 10



Yes

Why: It's convenient to mesh the beam elements on the lower edge, although the edge is not centered on the part. In a real model, the location of the beam center might make a difference. In this case, it doesn't.

Recovery Point

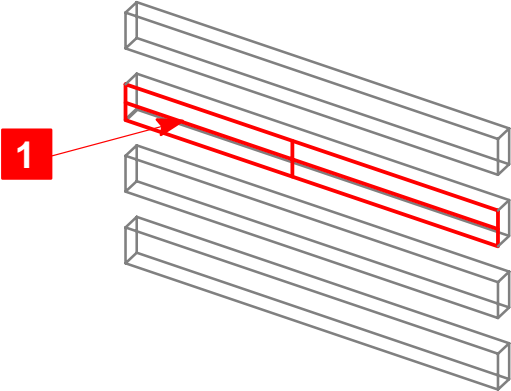


File

Save

What: Mesh thin-shell elements on the front surface of the second volume. Be sure to use the 5mm thick physical property table.

Hint



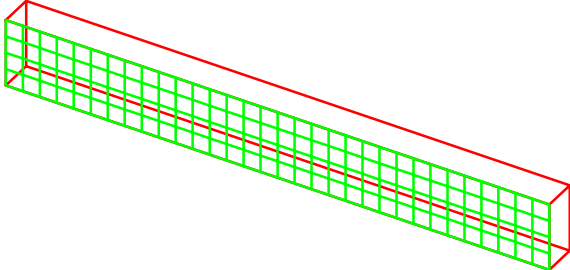
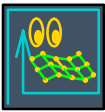
Element Length: 2.5
Physical Property



Physical Property Table form



Shell 5 MM

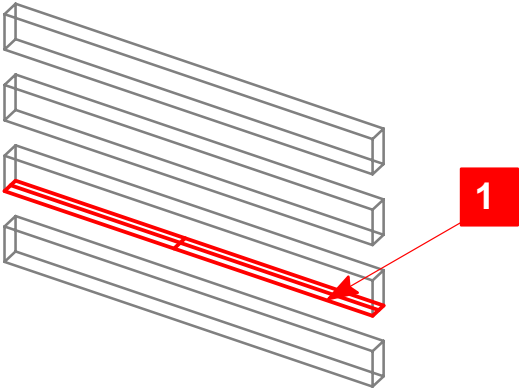


Keep Mesh

Things to notice Because of the vertical direction of the load, deformations of this model will be in the plane of the elements. The membrane behavior of the elements will control the results.

What: Mesh thin-shell elements on the bottom surface of the third volume. Be sure to use the 10mm thick physical property table.

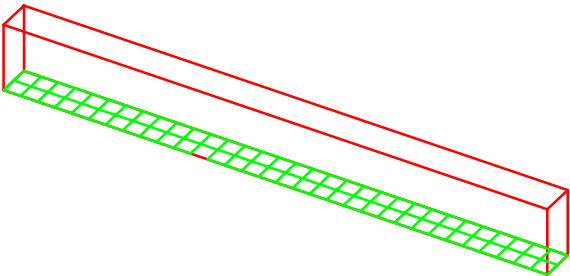
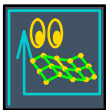
Hint



Element Length: 2.5
Physical Property



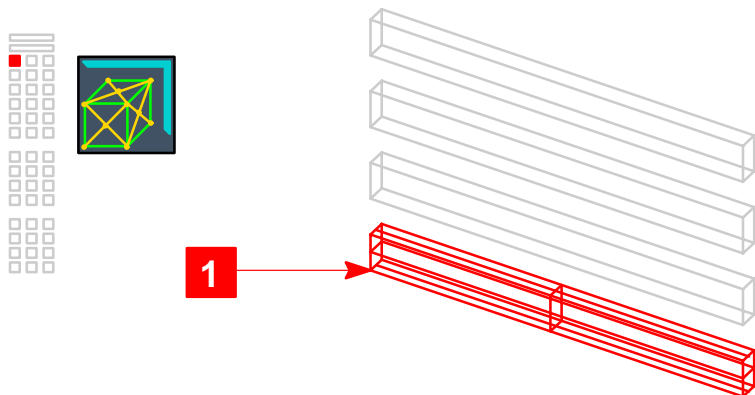
Shell 10 MM



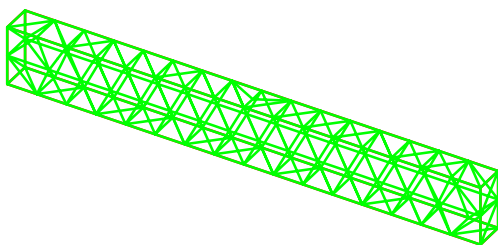
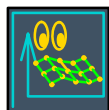
Keep Mesh

Things to notice Because of the vertical direction of the load, deformations of this model will be perpendicular to the plane of the elements. The bending behavior of the elements will control the results.

Hint



Element Length: 5



Keep Mesh

Recovery Point



What: Solve the model using linear statics.

How:



Model Solution



Manage Solution Sets form



Create...

Solution Set form



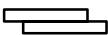
OK

Manage Solution Sets form



Dismiss

Recovery Point

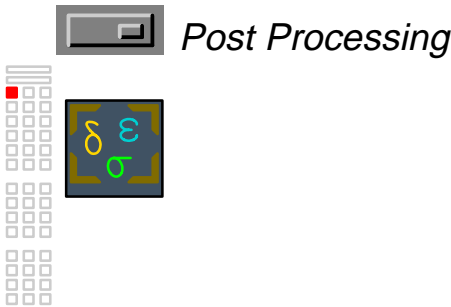


*File
Save*



What: Select just deformation results for the display.

How:



Results Selection form



What: Perform a hand calculation.

The equation for beam bending is:

$$Y = -P(L^3)/48EI$$

- P = 1000
- L = 80
- I = 5*(10^3)/12 = 416.7
- E = 2.068E8

Y = _____

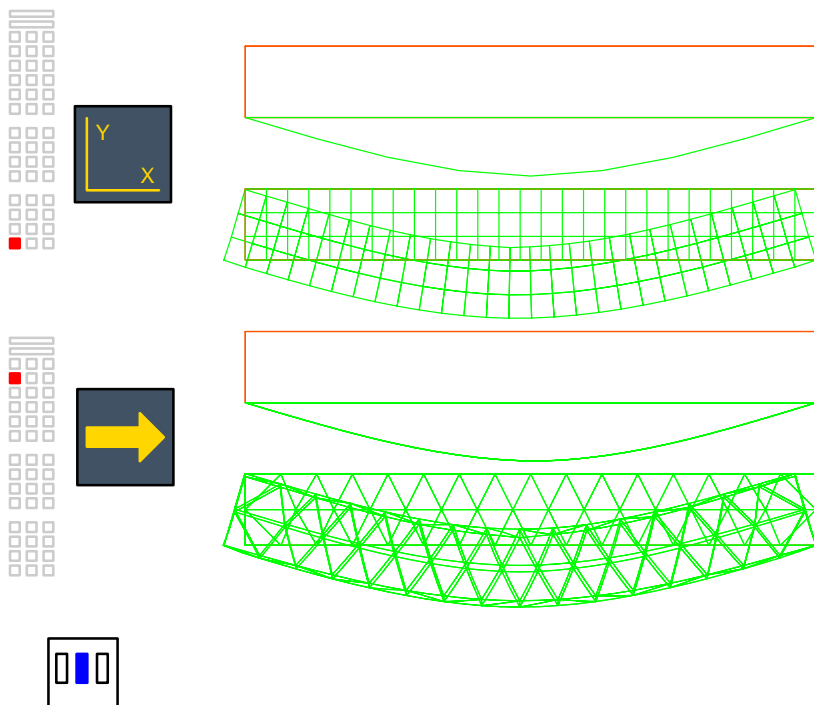
Hint

Most workstations have a calculator (such as xcalc) as one of the desktop tools to use for this calculation.

Why: It's always a good idea to calculate the expected result by hand. By this formula, expect the maximum displacement to be about -0.000124mm.

What: Display the deformations.

How:



Things to notice

All four models should display nearly identical deflection. If they don't, you entered the wrong physical properties or boundary conditions.

How do the displacements compare to the calculation ($Y = -.000124$)?

Remember the formula only includes bending deflection, not shear deflection. The deflection in all four models should be slightly higher than predicted by the formula.

What: Perform a hand calculation to check the stress.

Stress = MC/I

$M = PL/4$
 $L = 80$
 $P = 1000$
 $C = 5$
 $I = 416.7$

Stress = _____

What: Select the stress results to display.



Results Selection form



Stress_2 (select)



Display Results




Component: X



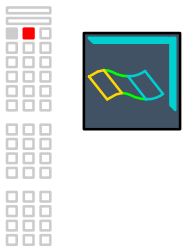
OK

Why: Use the X component of stress so that you can distinguish tension from compression in the X direction.

 If you don't have a license for the *Beam Sections* task, **skip to page 29**.

What: Turn off *Fast Display*, which doesn't support beam stress display.



How:



Display Template form

 *Contour...*

Contour Options form

 *Fast Display*
 *OK*

Display Template form

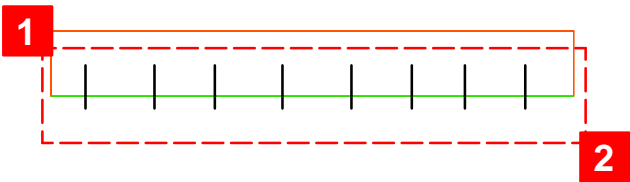
 *OK*

What: Display the beam element stresses in the top model.

How:



area-select the beam elements



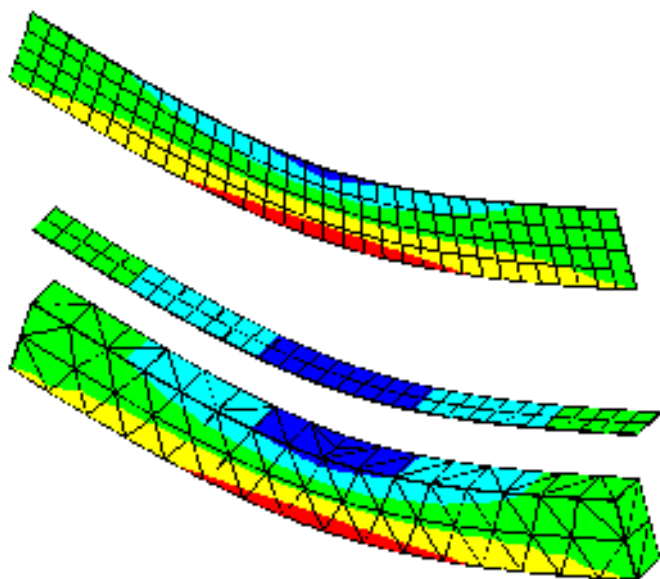
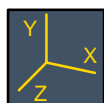
Why: Beam stresses display separately because line elements can't display as a shaded contour.

Things to notice

How does the maximum stress compare with the hand-calculated value? (Stress = 240)

What: Display a shaded contour plot of the shell and solid element stresses.

How:

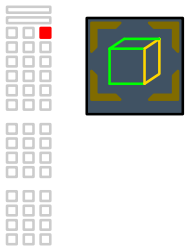


Things to notice

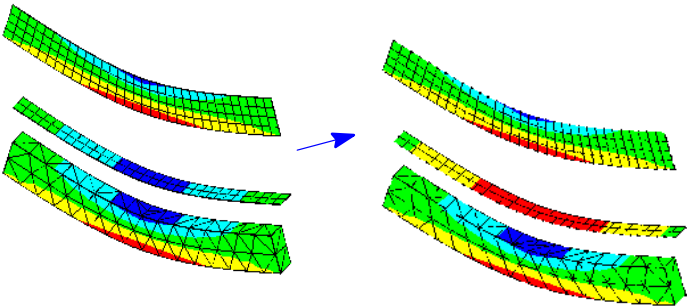
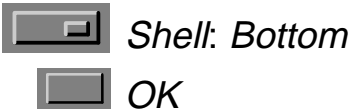
Read the maximum stress from the header. How does the maximum stress in these elements compare with the hand-calculated value? (Stress = 240)

What: Display stresses on the bottom instead of the top of the thin-shell elements.

How:




Calculation Domain form



Things to notice

The stresses in the center model change to red because the bottom of the elements are in tension. Why is there no change for the top shell model?

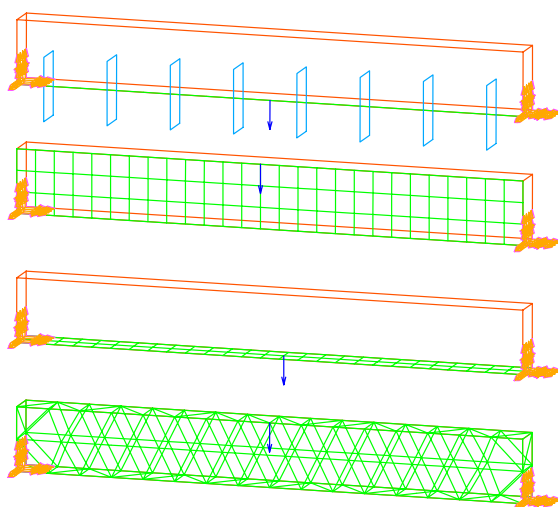
 With shell elements, make sure you know whether you are displaying stresses for the top, middle, or bottom.

Things to notice

This tutorial has shown the three classical methods of finite element modeling:

- beams
- shells
- solids

Each method has advantages and disadvantages.



Beam elements

Use beams when you want to see overall deflections in a long, slender beam or truss-type structure.



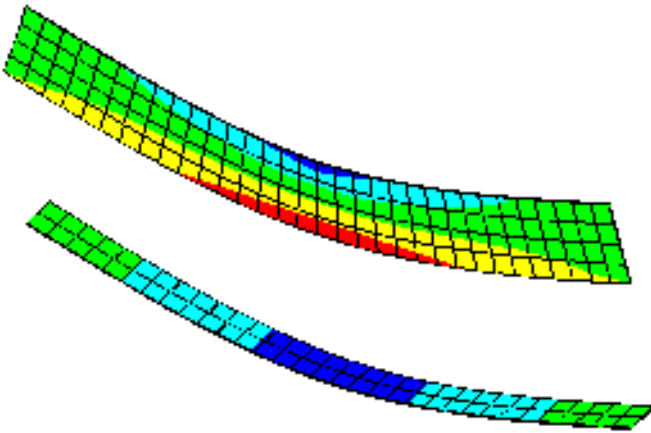
Although the beam elements give the closest results to the equations in this simple case, beam elements do not show local stress concentrations where the beams are attached. The cross section of beam elements does not deform under loading.

You must carefully examine the beam stress results. You may display stresses at different recovery points or display stress contours on the cross section of an individual beam element.

Although beam elements take longer to define since you must define the cross section properties, the solve time is the fastest with this element type.

Thin-shell elements

Use shell elements for thin-walled parts. Use these elements if you have a general 2D model to analyze. (Note: There are also other 2D elements, such as plane stress, plane strain, and axisymmetric elements which should be used in some cases.)

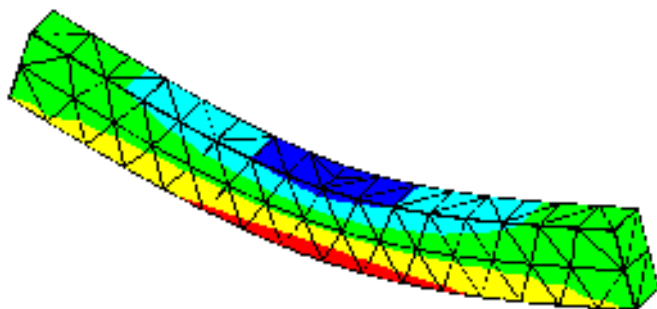


With thin-shell elements, be careful to enter the correct physical property to define the thickness. This element assumes that the stress varies linearly from top to bottom. These elements give a good display of stresses along the plane of the elements, but display stresses on both top and bottom surfaces to get the complete picture.

A thin-shell model requires more elements than beams to get good results, but still solves relatively fast.

Solid elements

Use solid elements for complex, thick parts.



Solid elements are the easiest to model, because you don't need to add information with a physical property table.

Solid elements give a complete picture of the 3D stress through the part, but use more degrees of freedom (DOF), so take longer to solve. Models may become very complex and take a very long time to solve if you have small features to accurately model.

Summary

Thin-shell elements and beam elements are abstractions of the 3D physical model. Thin-shell elements are abstracted to 2D elements by storing the third dimension as a thickness on a physical property table. Beam elements are abstracted to 1D elements by storing the 2D cross-section as separate beam section property.

Each level of abstraction takes more preparation time, but reduces the solution time. Understanding the behavior of each element type helps to make the best modeling decisions.

Tutorial wrap-up

You've completed the Which Element Type Should I Use 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

 Meshing a Model

 Modeling Beam Sections

Simulation: Element Library

 Introduction to the Element Library

What's next?

Another tutorial that discusses modeling issues is "How Many Elements Should I Use."

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