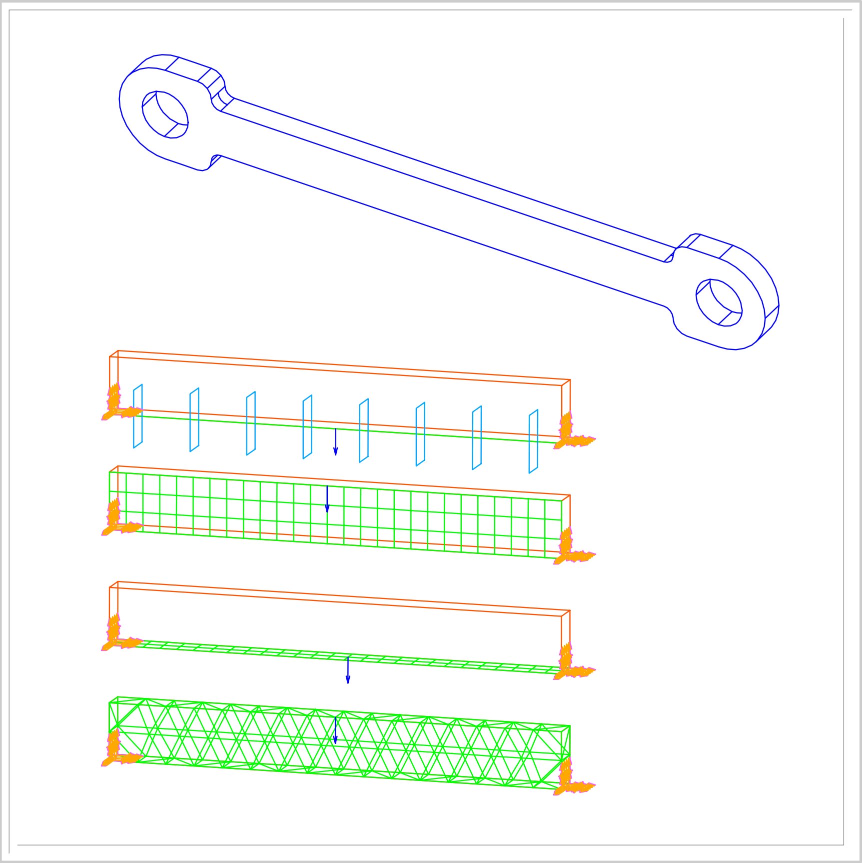


Which Element Type Should I Use?

I-DEAS™ Tutorials: Simulation Projects

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–

  Introducing the I-DEAS Interface
  Quick Tips to Using I-DEAS
  –and–
  Creating Parts
- Extruding and Revolving Features
- 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.

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

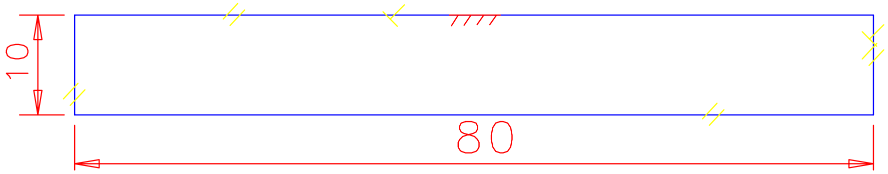
Set your units to millimeters.

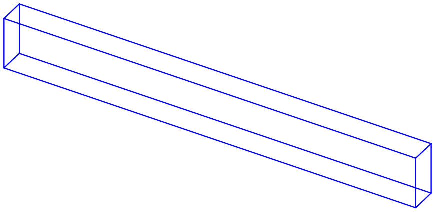
Options
Units

mm (milli newton)

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.

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.

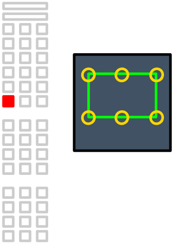
Hint
To reopen your model file to the previous save, press Control-Z.

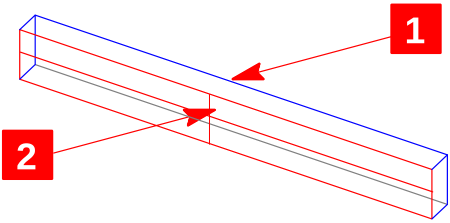
Setup

3 of 4

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

How:

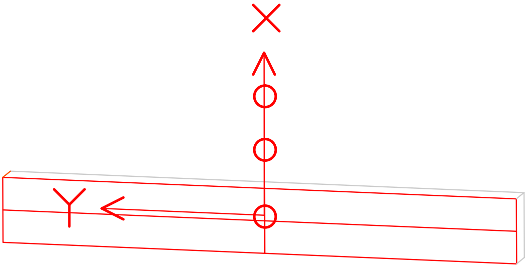




1

▯▮▯

2 pick the front face
(*Accept* if necessary)



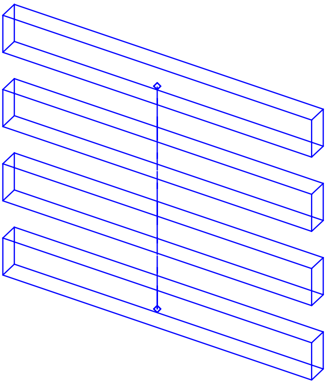
Rectangular Pattern form

☐ *Number along X*: 4
☐ *Distance between*: 20
☐ *Number along Y*: 1
☐ *OK*

Setup

4 of 4

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 the part.

Hint

Meshing

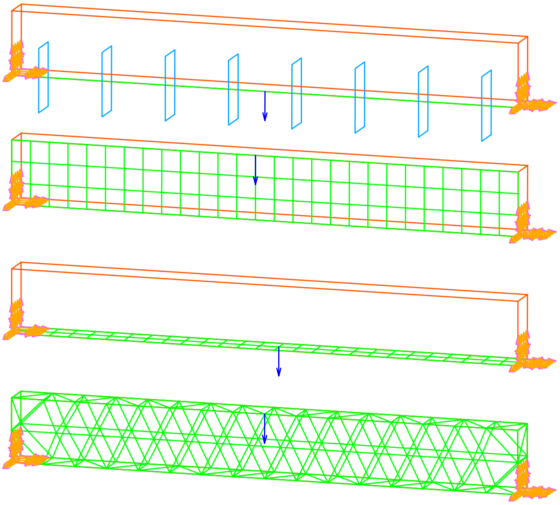
Recovery Point

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.



Create physical property tables 1 of 3

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

How:

Physical Property Tables form

Select Element Type form

◆ *Element: 2D*

Element Family: Thin Shell

OK

Thin Shell Physical Property Table form

Name: Shell 5 mm

Thicknesses: 5

There are four values of thickness (4V)—one value at each corner. For a uniform thickness, enter only one value.

OK

Don’t close the *Physical Property Tables* form.

Create physical property tables 2 of 3

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

Hint

Select Element Type form

Element: 2D

Element Family: Thin Shell

OK

Thin Shell Physical Property Table form

Name: Shell 10 mm

Thicknesses: 10

OK (all forms)

Create physical property tables 3 of 3

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 have no value.

OK

Recovery Point

File
Save

Define restraints 1 of 3

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

How:

Boundary Conditions

Filter...

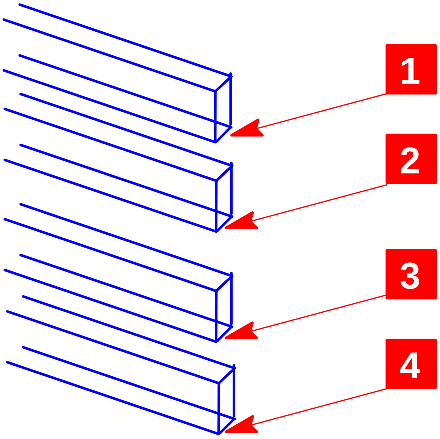
Selection Filter form

Edge...

Pick Only

---

shift-pick the 4 edges



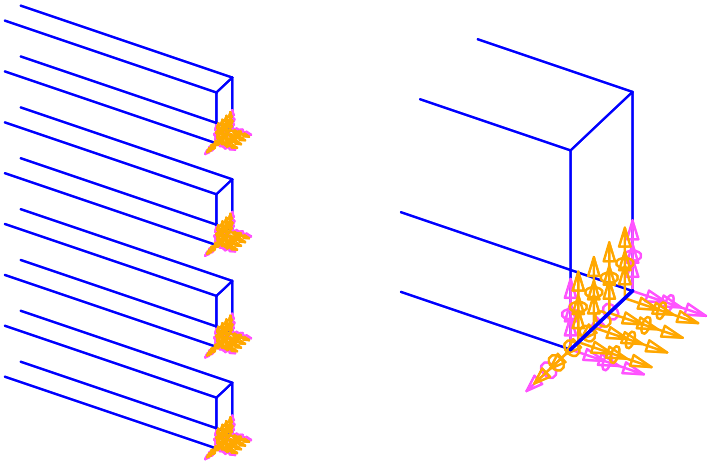
Done

Define restraints 2 of 3

Displacement Restraint on Edge form

- *Z Rotation*: *Free*
- *OK*

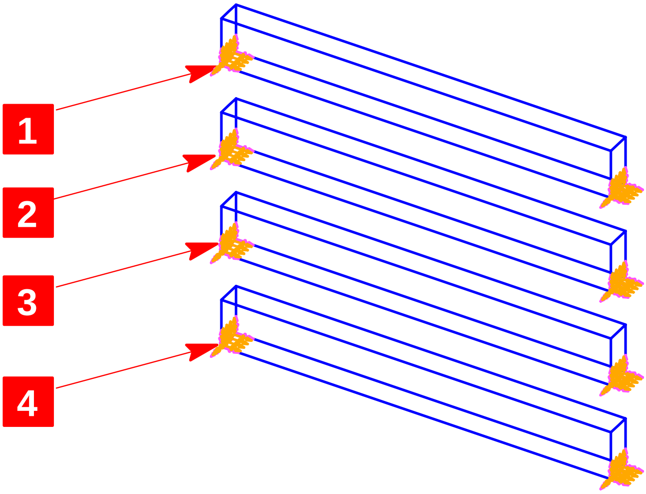
Result



Define restraints 3 of 3

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

Hint



Done

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

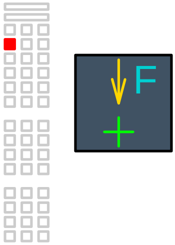
File
Save

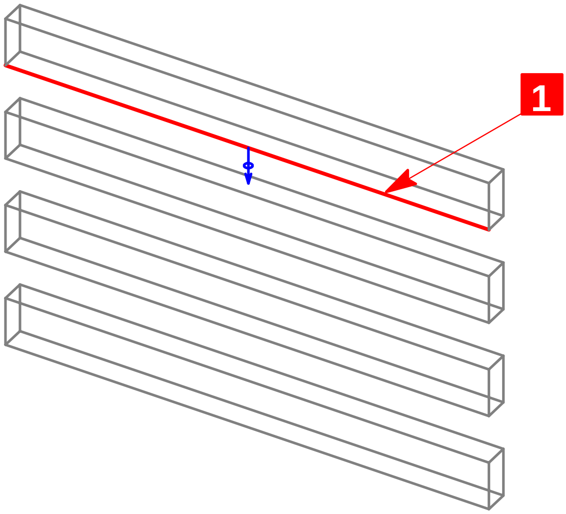
Define forces

1 of 5

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

How:





1

▯▯▮ *Locations on Edge*

▯▯▮ *Key In*

▯▮▯

▮▯▯ *Done*

Force on Vertex/Location on Geometry form

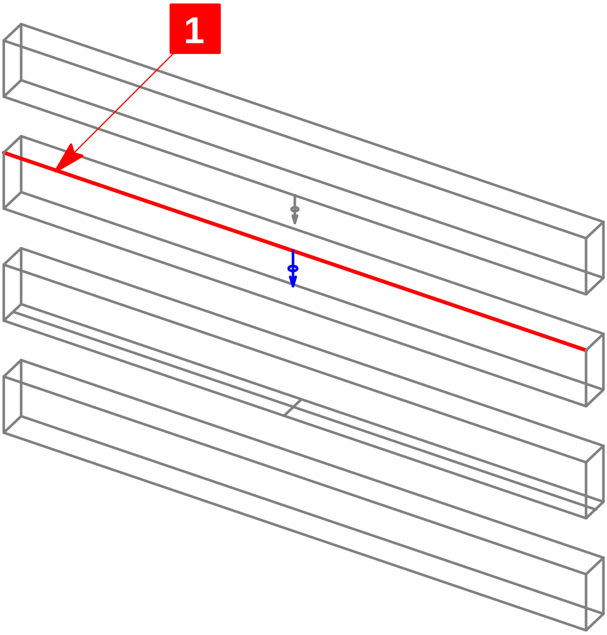
Y Force: –1000

OK

Define forces 2 of 5

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*

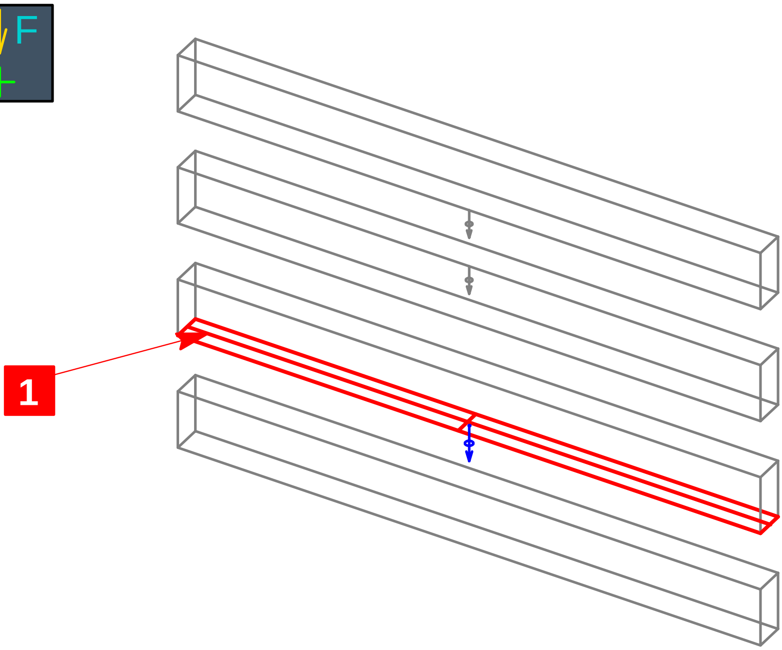
▭ *Y Force*: –1000

▭ *OK*

Define forces 3 of 5

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*

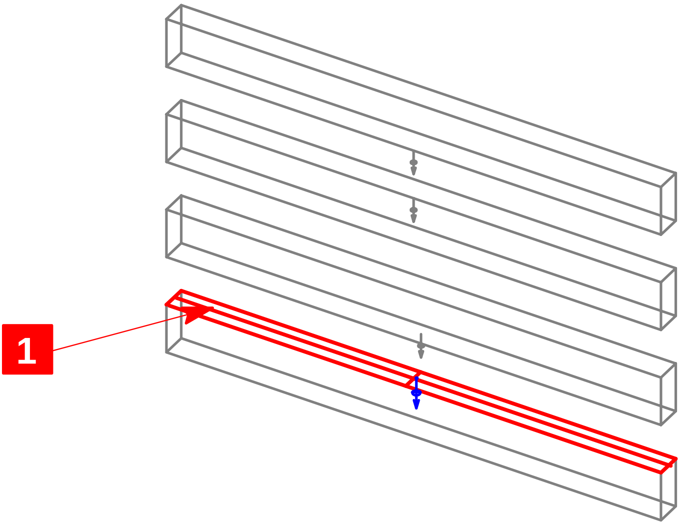
☐ *Y Force*: –1000

☐ *OK*

Define forces

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

Define forces

5 of 5

What: Create a boundary condition set that includes the restraint set and the load set.

Hint

Recovery Point

File
Save

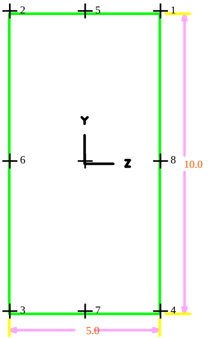
Define a beam element mesh 1 of 2

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

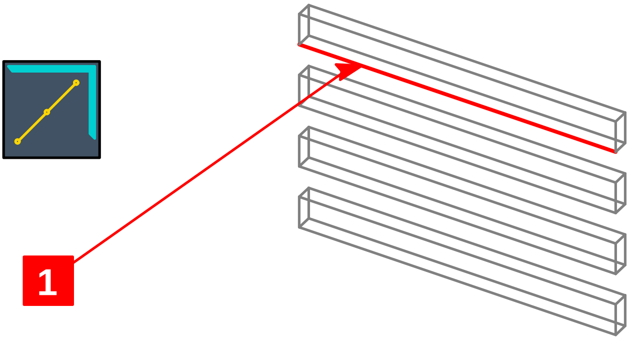
Yes

Define a beam element mesh 2 of 2

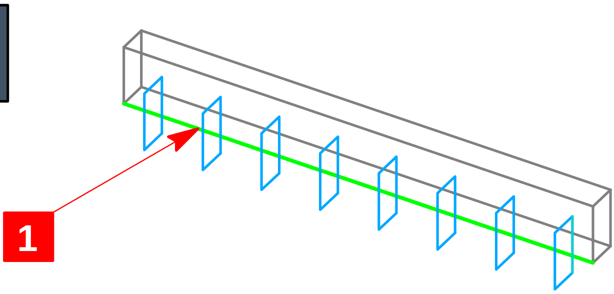
What: Mesh the edge shown on the top volume with beam elements.

Hint

Meshing



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.

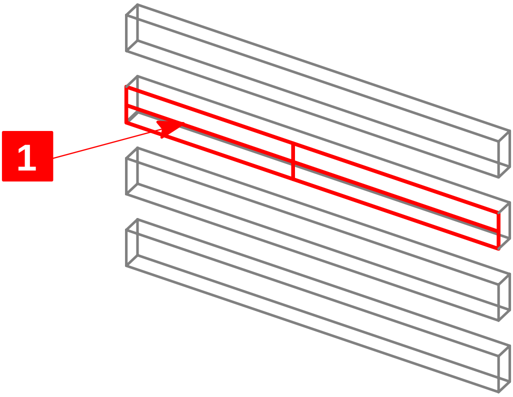
Recovery Point

File
Save

Define thin-shell meshes 1 of 2

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

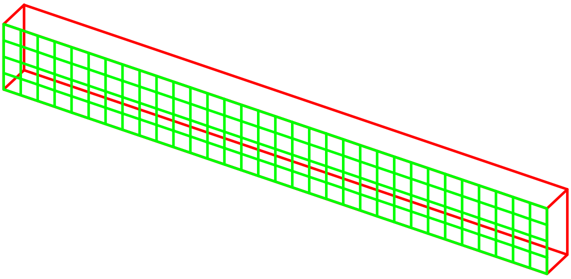
Hint



Element Length: 2.5
Physical Property

Physical Property Table form

Shell 5 MM



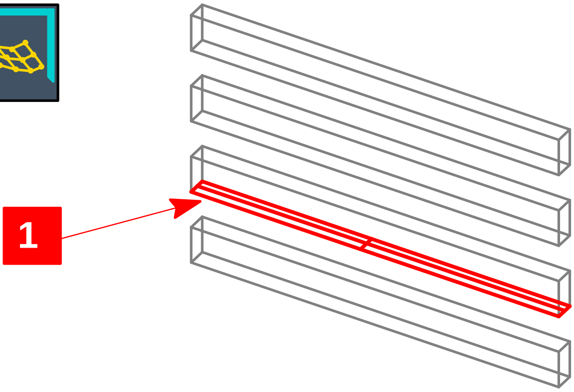
Keep Mesh

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.

Define thin-shell meshes 2 of 2

What: Mesh thin-shell elements on the bottom surface of the third volume. Be sure to use the10mm thick physical property table.

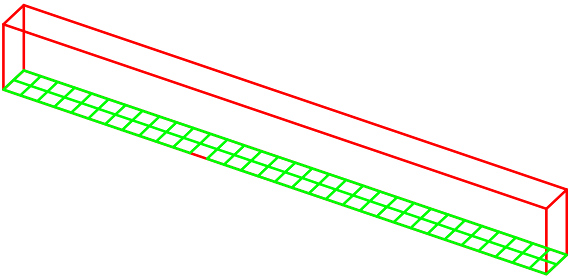
Hint



☐ *Element Length*: 2.5
Physical Property

?

Shell 10 MM



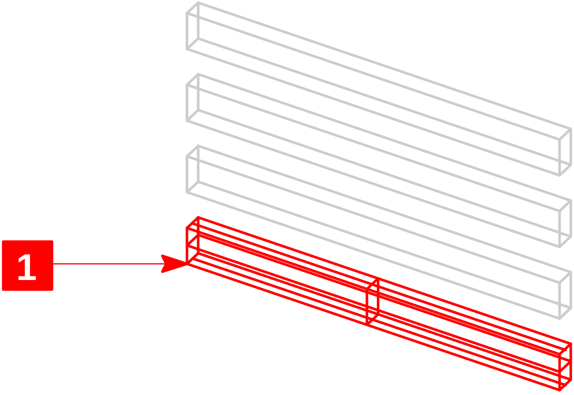
☐ *Keep Mesh*

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.

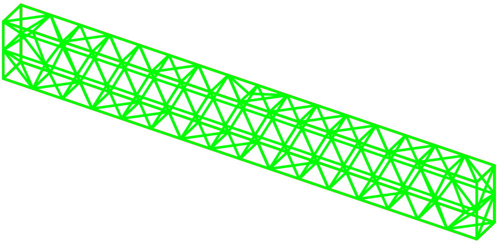
Define a solid mesh 1 of 1

What: Mesh the bottom volume with solid elements.

Hint



Element Length: 4



Keep Mesh

Recovery Point

File
Save

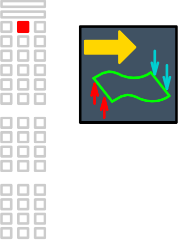
Solve the model

1 of 1

What: Solve the model using linear statics.

How:

Model Solution



Manage Solution Sets form

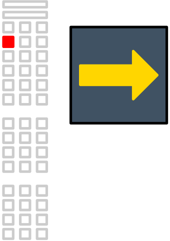
Create...

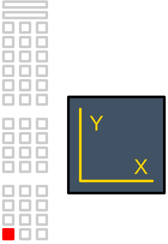
Solution Set form

OK

Manage Solution Sets form

Dismiss





Recovery Point

File
Save

Compare displacements 1 of 2

What: Select just deformation results for the display.

How:

Post Processing

Results Selection form

Display Results: *Clear*

OK

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 = ______________

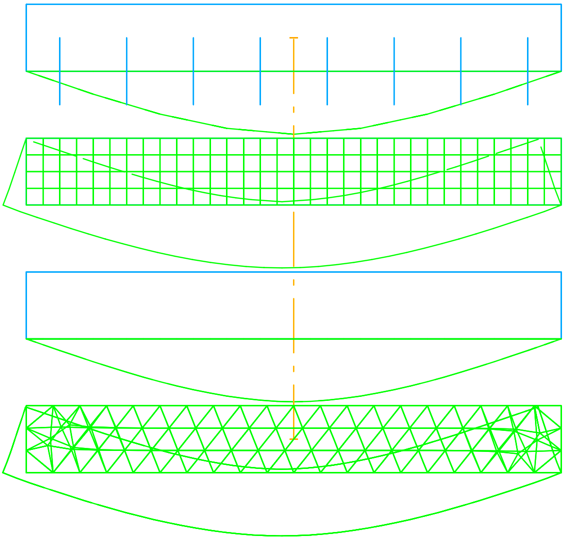
Hint
Most workstations have a calculator (such as xcalc) as one of the desktop tools to use for this calculation.

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.

Compare displacements 2 of 2

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: Reopen the file to the previous save.

Hint

Control–Z

Compare stresses 1 of 6

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_3 (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.

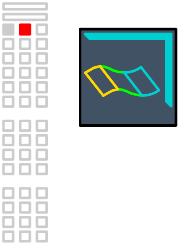
Compare stresses

2 of 6

If you don't have a license for the *Beam Sections* task, **skip to page 30**.

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*

Recovery Point

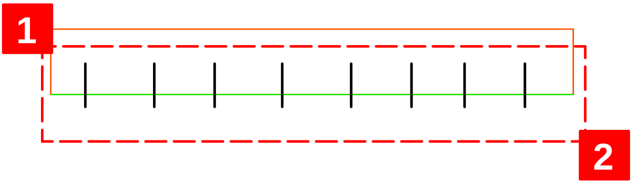
File
Save

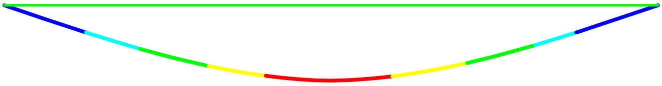
Compare stresses 3 of 6

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: Reopen the file to the previous save.

Hint

Control–Z

Compare stresses 4 of 6

What: Turn on element borders and turn off the undeformed model display.

How:

Display Template form

▭ *Deformed Model*...

Deformed Options form

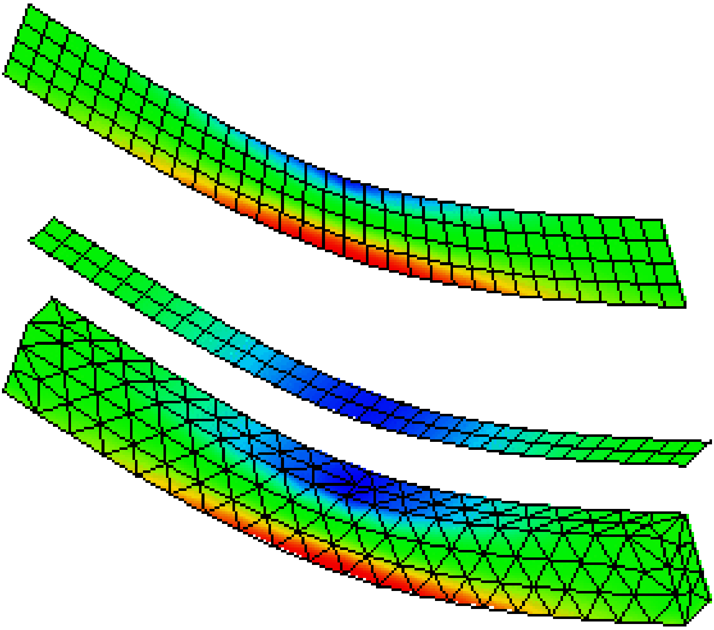
☑ *Element Borders* (on)

☐ *Undeformed Model* (off)

Compare stresses 5 of 6

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)

Compare stresses 6 of 6

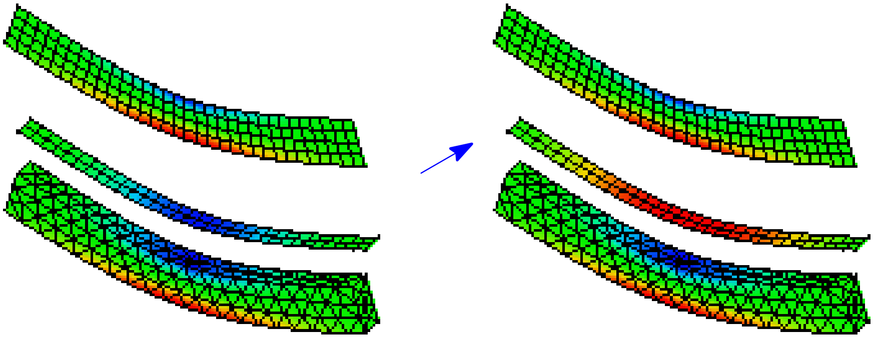
What: Display stresses on the bottom instead of the top of the thin-shell elements.

How:

Calculation Domain form

Shell: *Bottom*

OK



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?

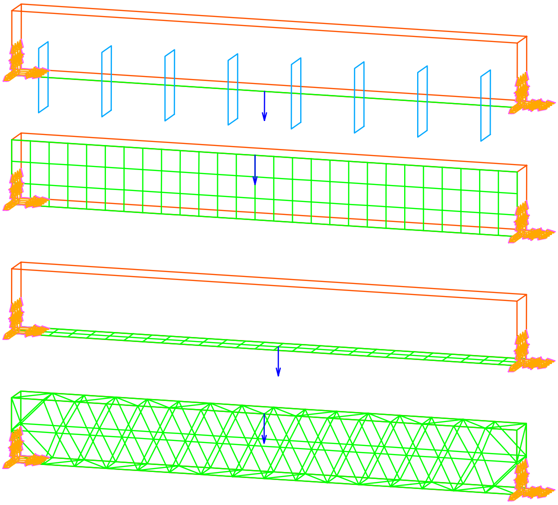
With shell elements, make sure you know whether you are displaying stresses for the top, middle, or bottom.

Things to notice

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

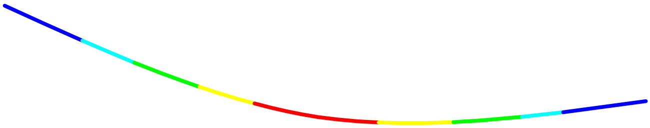
- beams
- shells
- solids

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.



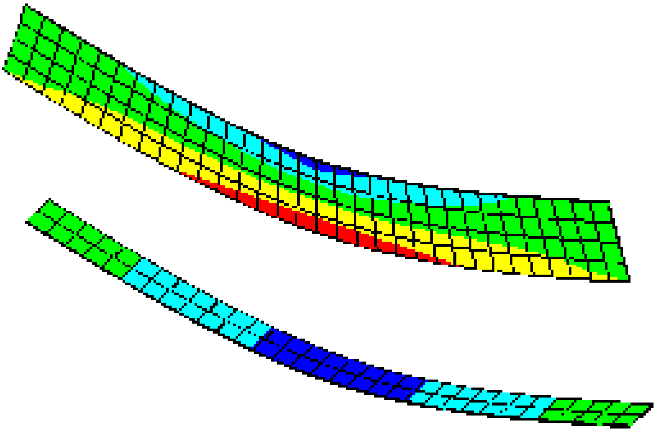
Although the beam elements give the closest results to the equations in this simple case, beam elements do not show local stress concentrations where the beams are attached. The cross section of beam elements does not deform under loading.

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.)

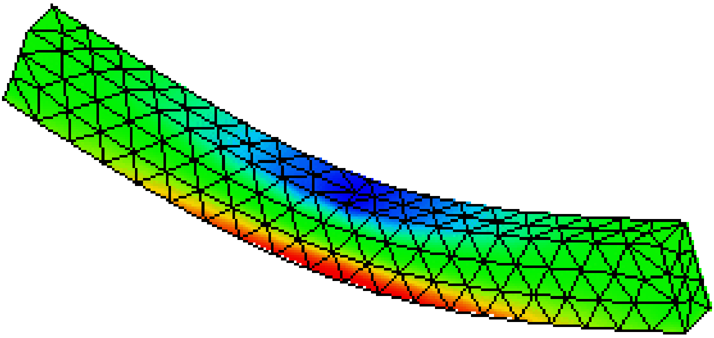


With thin-shell elements, be careful to enter the correct physical property to define the thickness. This element assumes that the stress varies linearly from top to bottom. These elements give a good display of stresses along the plane of the elements, but display stresses on both top and bottom surfaces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

What’s next?

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