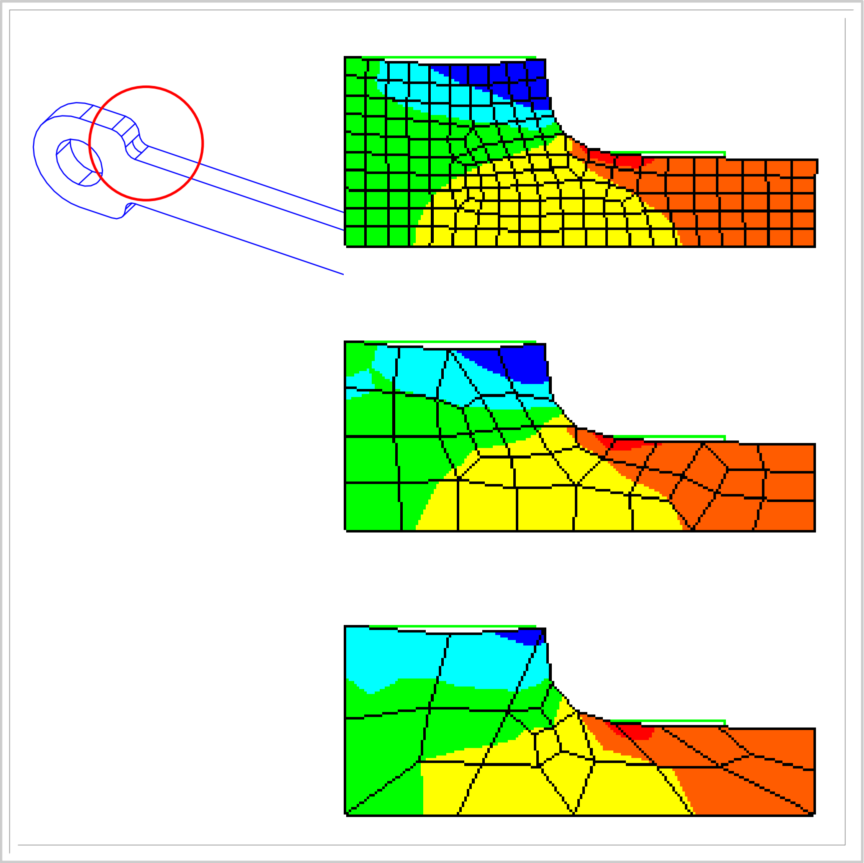


How Many Elements Do I Need?

I-DEAS™ Tutorials: Fundamental Skills

In this tutorial you'll see how using different numbers of elements affects the accuracy of the results.

Learn how to:

- study the effect of element size
- graph stress convergence
- modify part geometry
- investigate stress convergence
- estimate stress error
- understand automatic convergence methods

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
- Which Element Type Should I Use?

This tutorial doesn't teach you how to create a mesh on a model and solve it. These skills are presented in the prerequisite tutorials. You should be comfortable with these skills before you do this tutorial.

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

File
Open

Open Model File form

Model File name: any unique name

OK

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

Simulation

Master Modeler

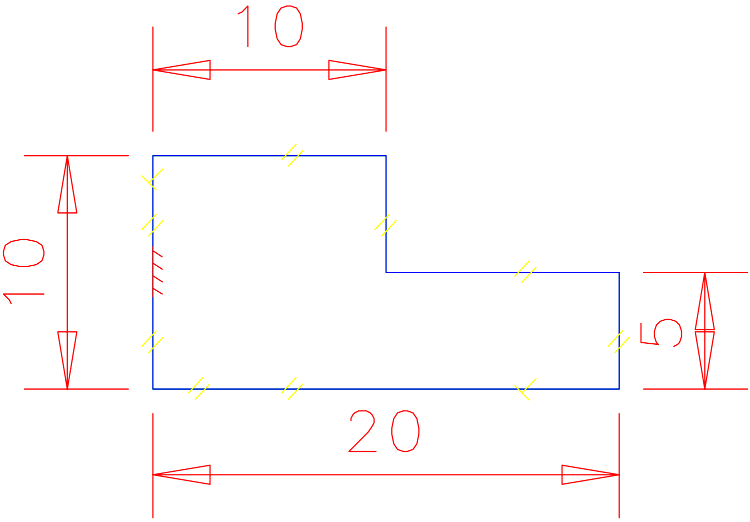
Set your units to mm.

Options
Units

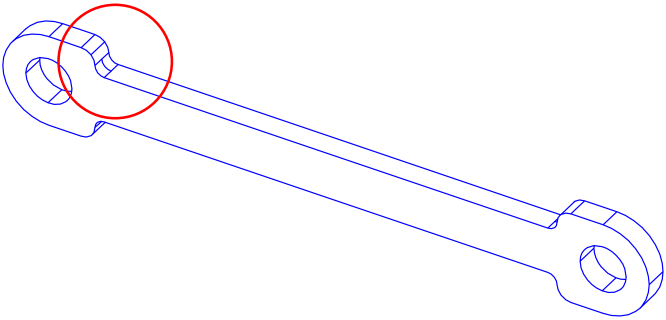
mm (*milli newton*)

What: Sketch and dimension the shape shown.

Hint

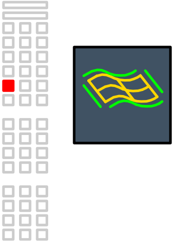


Why: The section you sketch will be used to model the detailed area of the linkage shown below. For the first model, the fillet won’t be included.



What: Use *Surface by Boundary* to create a surface.

Hint



Options

Autochain Wireframe

OK

pick edge

Yes

Why: The *Surface by Boundary* command is a good way to make a surface for meshing a 2D model.

Things to notice

In line mode, you'll see only the edges of the surface. In shaded mode you'll see that there is a surface inside the boundary.

Save your model file.

File
Save

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

No

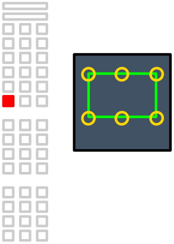
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

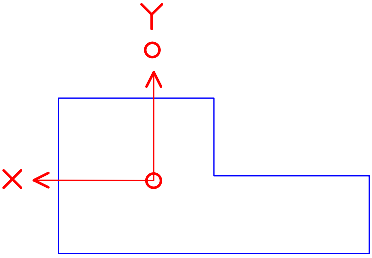
Why:
If you make a mistake at any time between saves and can’t recover, you can reopen your model file to the last save and start over from that point.

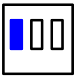
Hint
To reopen your model file to the previous save, press Control-Z.

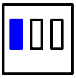
What: Make a pattern of three, spaced 15mm in the vertical direction.

Hint





 pick anywhere on the part

 pick the surface

Things to notice If your Y axis is not vertical, either pick *Align* on the form and align the Y axis or just switch the X and Y field values below.

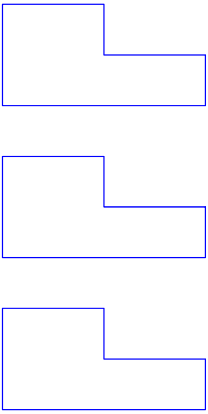
Rectangular Pattern form

Number along X: 1

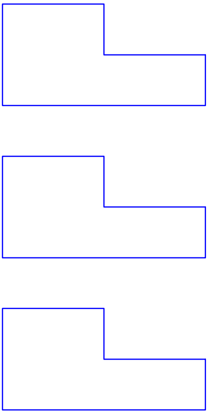
Number along Y: 3

Distance between: 15

OK

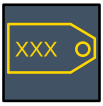


Why: The three surfaces are contained in one part which you will use to perform three case studies in one FE model.



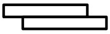
What: Name this part.

Hint



 *Name*: Fillet Model

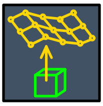
Recovery Point

 *File*
Save

What: Create an FE model associated with the part.

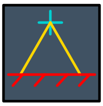
Hint

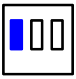
 *Boundary Conditions*

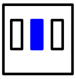


What: Restrain all six degrees of freedom at the left edges.

How:

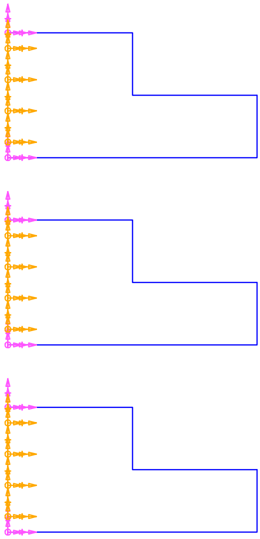


 shift–pick three left edges



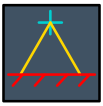
Displacement Restraint on Edge form

 *OK*



What: Restrain the bottom edges on the plane of symmetry.

How:



shift–pick three bottom edges

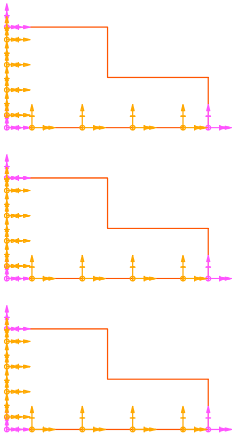
Displacement Restraint on Edge form

X Translation: free

Z Translation: free

Y Rotation: free

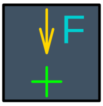
OK

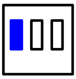


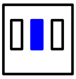
Why: To maintain symmetry, nodes must be restrained on the plane of symmetry so that they can't move perpendicular to the plane.

What: Apply a –1000mN total in-plane force on the edges on the right side.

How:



 shift–pick three right side edges

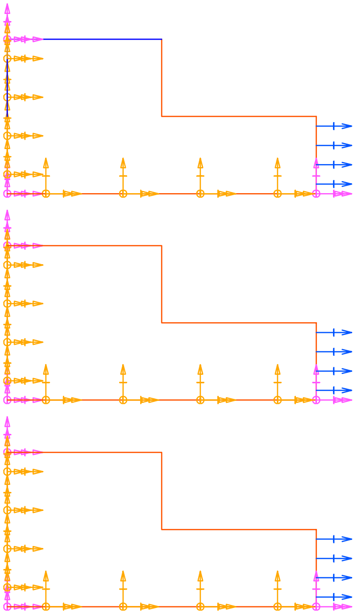


Force on Edge form

Total Force

In Plane Force: –1000

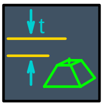
OK



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

Hint

Meshing



Thin Shell

physical property name: Shell Thickness

No

Directory

TK THICKNESS [4V]

1st value for thickness: 5

<Return> (for rest of the prompts)

Done

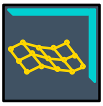
Things to notice For this tutorial, you will use the default material property, which is for isotropic steel.

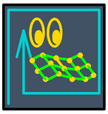
Recovery Point

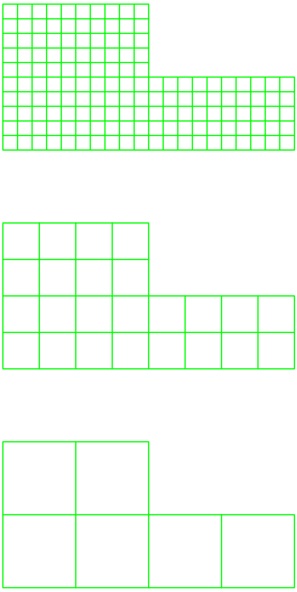
File
Save

What: Mesh the three surfaces with thin-shell elements with sizes of 1mm, 2.5mm, and 5mm.

Hint







What: Create the solution set, using default parameters.

Hint

Model Solution

Create...

Recovery Point

File

Save

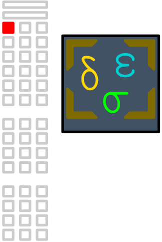
What: Solve the model.

Hint

What: Select the displacement results only.

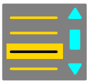
Hint

 *Post Processing*



Results Selection form

 *Display Results*: *Clear*

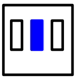
 Displacement_1

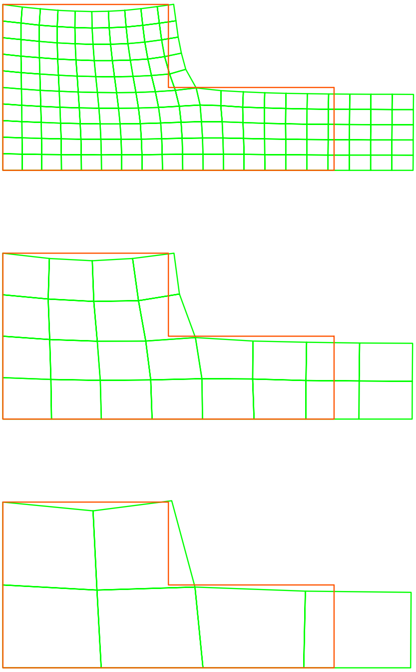
 *Deformation Results*

 *OK*

What: Display the displacements.

Hint





Things to notice

How much does the element size affect the overall displacement in the three different cases?

If your only objective is to compute displacements, you can use a relatively coarse model and still get good results.

Study the effect of element size

What: Display the stresses.

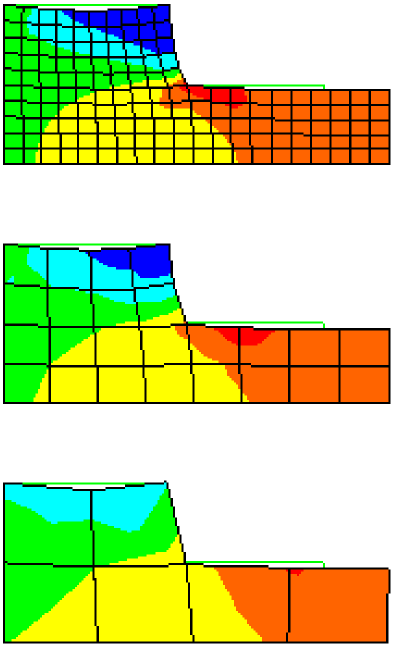
Hint

Results Selection form

Stress_2

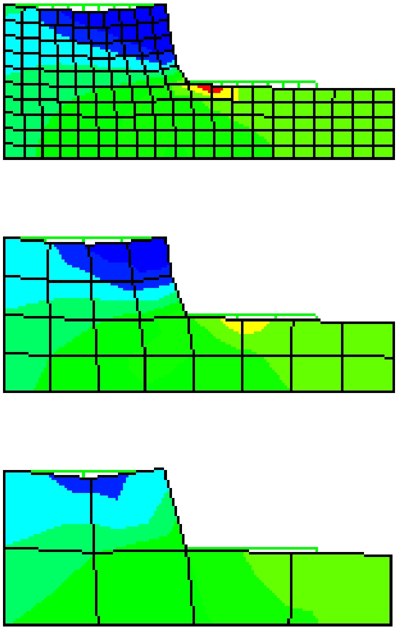
Display Results

OK



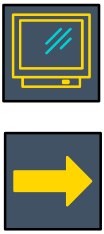
Things to notice

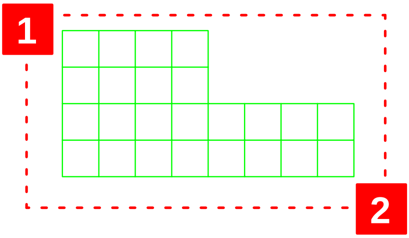
How is stress affected by element size?



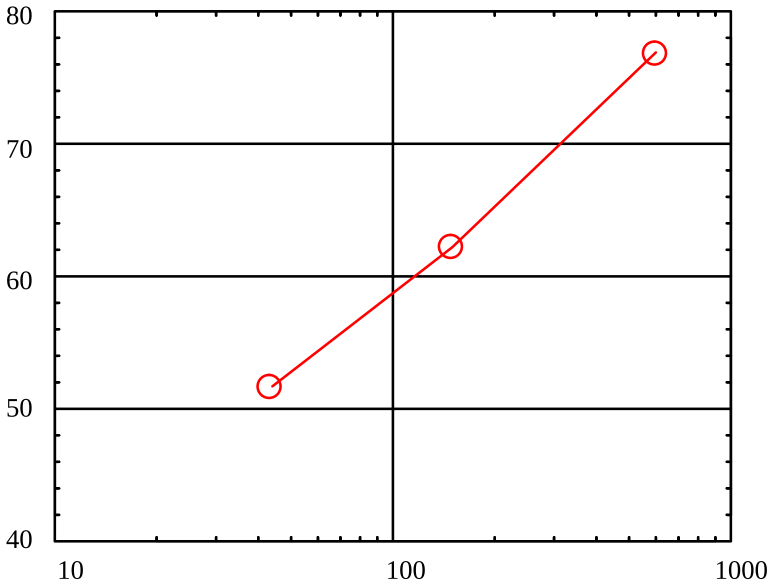
Hint Find the maximum stress in each by dragging a box around one model at a time.

Hint





In a typical graph, the maximum stress in each model is plotted against the number of elements in each model.

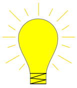


Things to notice

As the number of elements increases (and the elements get smaller), is the stress value converging on a stable value? Why not?

Remember

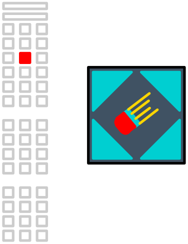
The stress concentration in a square corner is infinity. This is called a singularity. If you continue to refine the mesh smaller and smaller, the stress will continue to increase.



This is an example of a geometric singularity. Another type of singularity is caused by applying a restraint or a force at a point. If you find high stresses at the point of load or restraint, you'll need to more accurately model the boundary conditions as distributed loads and restraints.

What: Delete all results so that you can modify the part model.

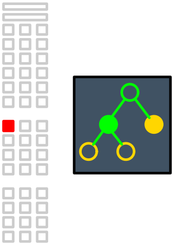
Hint

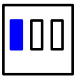


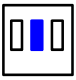
What: Select the first feature of the part.

How:

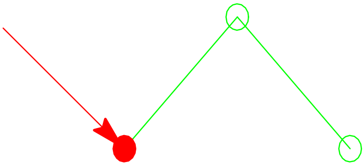
 *Master Modeler*



 pick the part



1

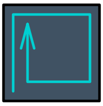
1 

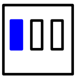
 *Dismiss*

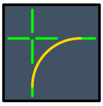
Modify part geometry

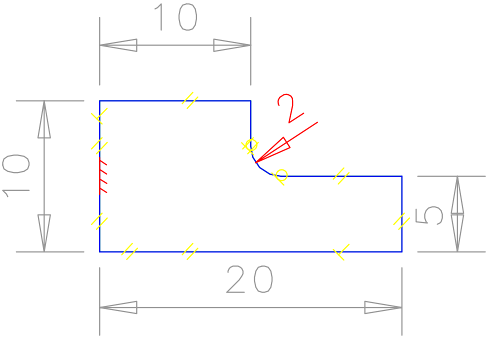
What: Add a fillet with a 2mm radius to the wireframe of the feature, to remove the geometric singularity.

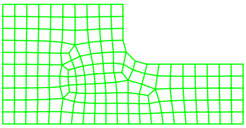
How:

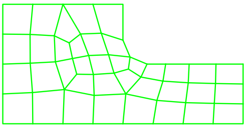


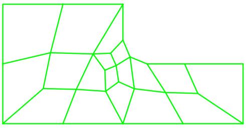
 *Wireframe*











Things to notice

The software automatically updates the mesh after modifying the part.

What: Solve the new model.

Hint

Model Solution

You do not need to create a new solution set. Even though the model has changed, nothing defined in the solution set has changed.

What: Display the stresses.

Hint

Post Processing

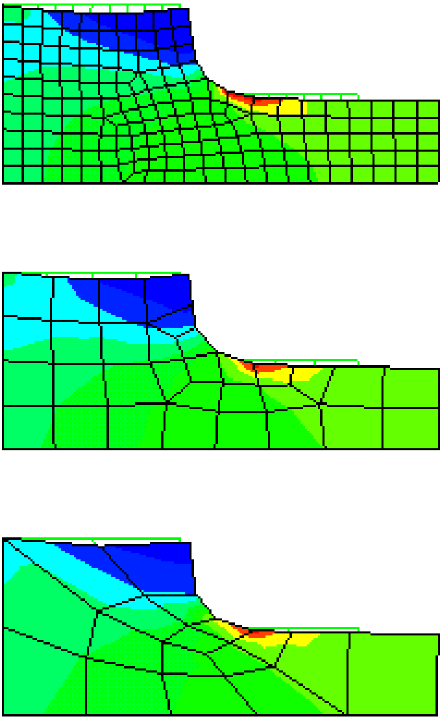
Results Selection form

Stress_2

Display Results

OK

Result

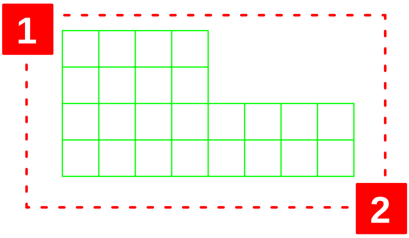


Things to notice

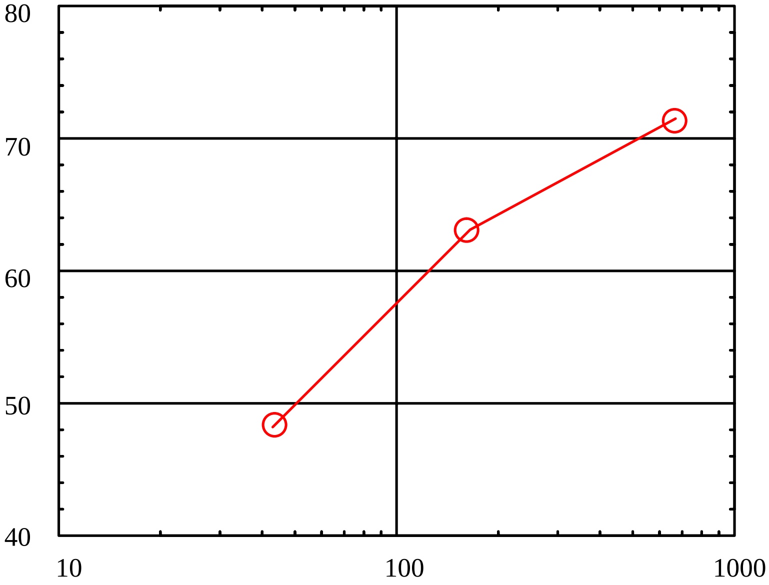
There is less apparent difference in the stress display between the three different cases.

What is the highest stress in each case?

Hint



The graph now will tend to show convergence as elements get smaller. (Your results will differ, depending on meshing parameters.)



Things to notice

Is the stress value approaching a stable value as the number of elements in the model increases? Has the stress converged on a value?

This is the real way to answer the question of how many elements you need—to refine the model and see if the answers have converged.

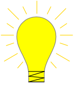
With experience, you'll begin to get a feel for how many elements are required for the types of problems that you solve, and the accuracy that you require.

Is there a way to estimate how close your results are to the correct answer by looking at one solution? The technically correct answer is no, but there are some simple methods you can use to assess the quality of the results.

What: Compare the smoothness of the contours in each of your three models and those shown here. (The model shown here used even smaller elements, using a stepped shaded display.)

Things to notice

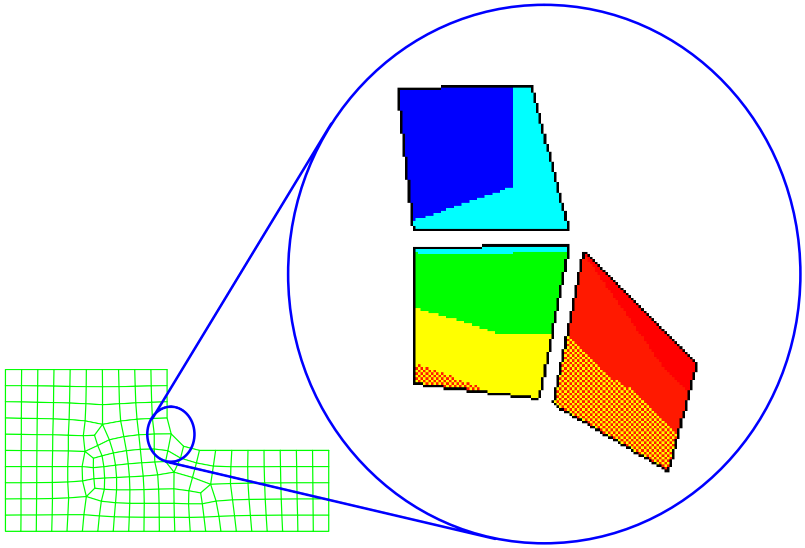
In which of your models can you see deviations in the stress contours along element edges?



The smoothness of the stress contours is an indication of the quality of the results. Stress contours should not have any unexplained jagged bends or follow the lines of elements.

Another technique is to plot the stresses without averaging between elements to get an “order of magnitude” feel for how close your model is to the true answer.

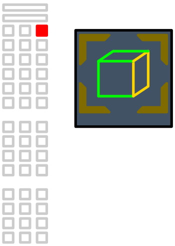
Stress is calculated for each element in the model. Where elements connect at common nodes, stresses are averaged. As you converge on an answer, the difference in stress between adjacent elements converges to zero. This value gives you a feel for how far off your results might be.



An analogy is, if you have two clocks, each showing a different time, you don’t know the correct time, but you know that the true time is off from one or both of the clocks by at least half the difference between the two.

What: Turn off averaging across elements, and generate the stress display again.

How:

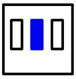


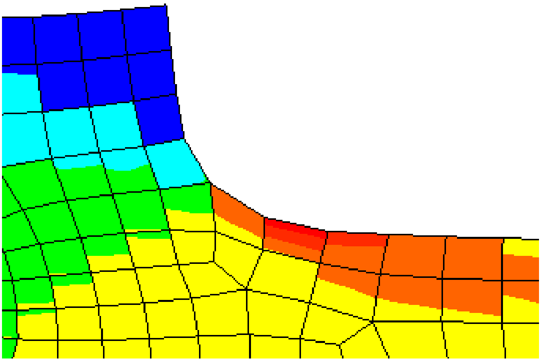
Calculation Domain form

Do Not Average Across

Elements

OK





Things to notice

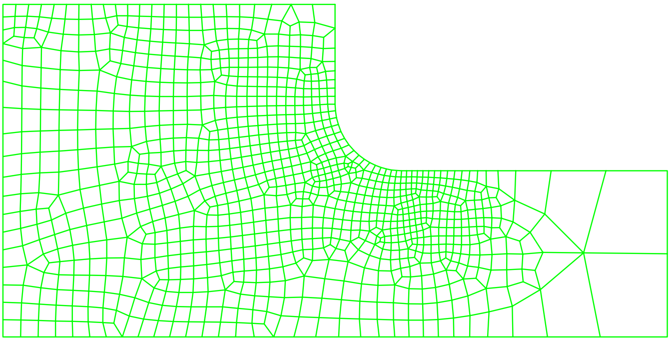
The contours have discontinuities across element edges.

Understand automatic convergence methods

Although you may use the manual methods previously shown to converge on accurate answers, there are two methods in the software to automatically control the iterative process. These are the Linear Adaptive Statics method and the P element method.

Linear Adaptive Statics

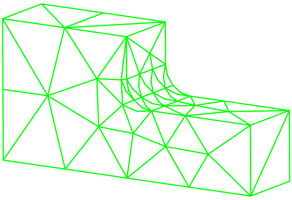
The Linear Adaptive Statics method uses an automatic process to iteratively change element size to converge on accurate results. Elements will be refined where needed, and may actually get larger where they don’t need to be as small as you started, based on strain energy. The model below was automatically refined using this method.

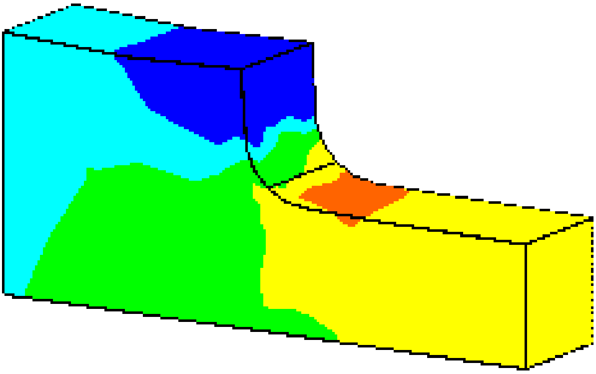


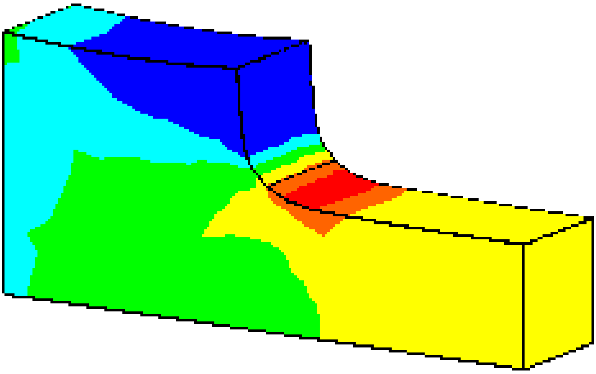
Understand automatic convergence methods

P Element Method

Another automatic method is the P element method, which increases the order of the equations in each element rather than adding more elements to converge on stable results. This method works with tetrahedral solid elements.



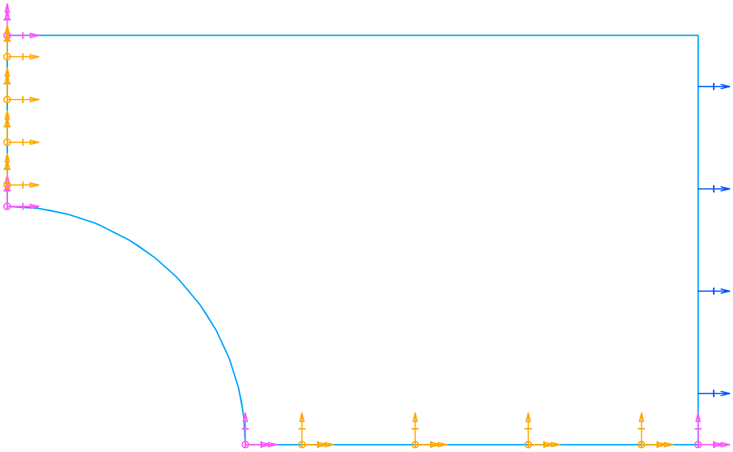


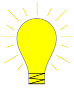


These two results are from the same model with the same color bar values. The first is from the initial model, the second is from the third iteration of increasing the P element order.

On your own...

Model a classical text–book problem such as a plate with a hole that you can check with known results. Run several cases to determine how many elements you need, to converge on an accurate answer.





Use symmetry to model only one quarter of the problem by restraining DOF so nodes can't move perpendicular to the symmetry plane. On the left edge, restrain X translation, Y rotation, and Z rotation to be constant. On the bottom edge, restrain Y translation, X rotation, and Z rotation constant. Leave the other directions free.

Remember

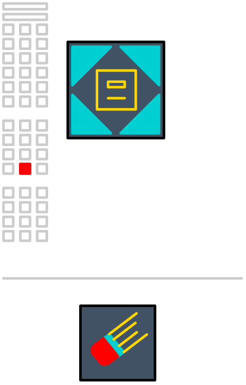
One final warning:

Don’t let convergence give you a false sense of security! Just because you have a model that converges on a value doesn’t guarantee that the value is correct. If you have used the wrong boundary conditions, material properties, or physical properties, you will converge on the wrong answer.

You have completed the “How Many Elements Do I Need?” 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
  - Creating a Mesh
- Post–Processing Results
  - Icon Overview for Post Processing
  - Displaying Results

Simulation: Model Solution/Optimization User's Guide
- Using the Solvers
  - Icon Overview for Model Solution
  - Getting Started with the Solvers
  - Defining the Analysis
  - Solving the Model
  - Using Linear Statics Analysis
  - Using Linear Statics Adaptive Analysis
  - Using Linear Statics – P Method

What's next?

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

To exit this tutorial, select:

File
Exit

Warning!
Do not use the menu in the *I-DEAS Icons* window to exit. Use the menu in the Acrobat Reader window.

I-DEAS Master Series™ Online Tutorials

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