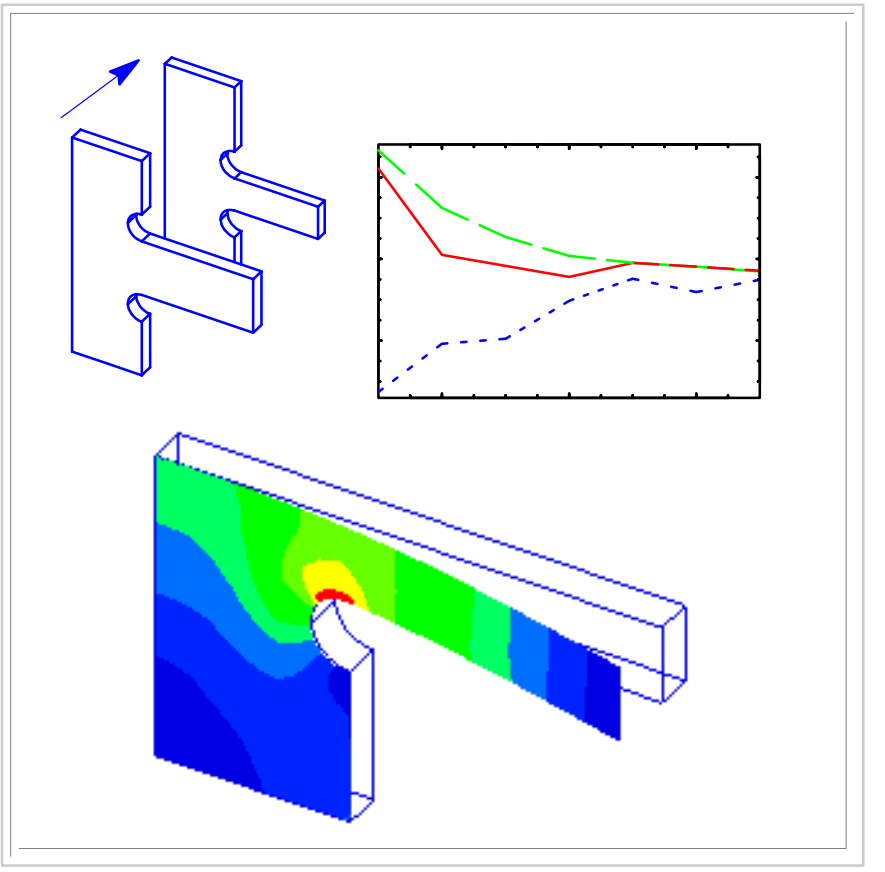


Optimization Redesign

I-DEAS™ Tutorials: Fundamental Skills

Learn how to:

- create a design
- create design parameters
- create limits and design goal
- solve the optimization redesign
- graph optimization history
- display optimization iterations
- update the part

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

  Quick Tips to Using I-DEAS
  –and–
  Creating Parts
- Introduction to Simulation
- Managing Parts in Model Files
- Free Meshing
- Boundary Condition Sets
- Boundary Condition Surface Loads
- Boundary Condition Symmetry
- Displaying Results

Recommended tutorials:

- Parameter Studies

If you're using the model file saved in the tutorial "Optimization Redesign," **skip to page 14.**

Otherwise, open a new model file and give it a unique name. Then create the part shown on the following Setup pages.

Hint

File
Open

Model File name: any unique name

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

Simulation
Master Modeler

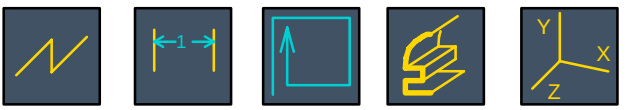
Set your units to mm.

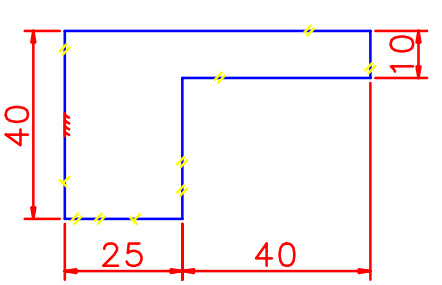
Options
Units

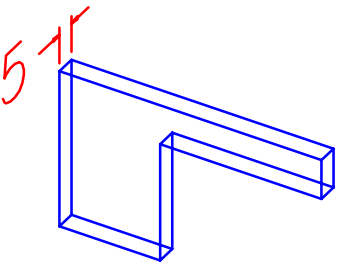
mm (*milli newton*)

What: Create this part to the dimensions shown.

Hint



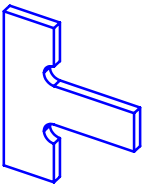




What: Name the part “Hinge Snap.”

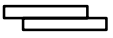
Hint

Why: This part represents one half of the plastic tab that acts as a spring in a self-closing hinge. You will use symmetry to reduce the model size.



The goal is to design the tab to give the desired stiffness. The deflection should be 3mm with a force of 10,000 mN. The stress must be less than 20,000 mN/mm**2. A design goal is to minimize the amount of material to reduce cost, weight, and part size.

Save your model file.



File
Save

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

No

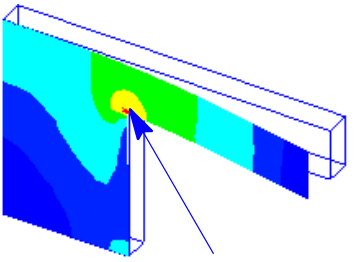
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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.

A first analysis shows high stresses in the square corner. (This square corner is actually an infinite singularity.)

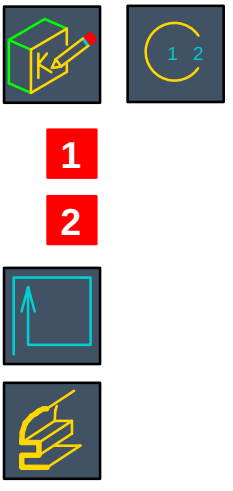
You don't need to perform this analysis. This result is shown only to explain why you're doing the step below.

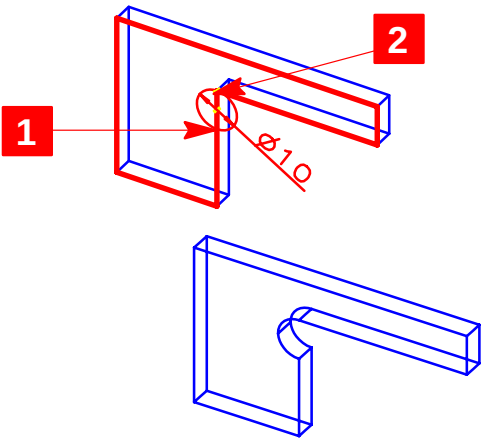


What: Cut a stress-relief in the corner.

Hint

Sketch a circle on the face, and make a cutout. Make sure that the circle edge is tangent to the corner.

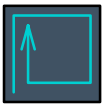




Why: For parameter studies or optimization redesigns, the program will modify dimensions of the part. It is important that the part is constructed properly so it maintains design intent when it is modified.

What: Verify that your part will update properly.

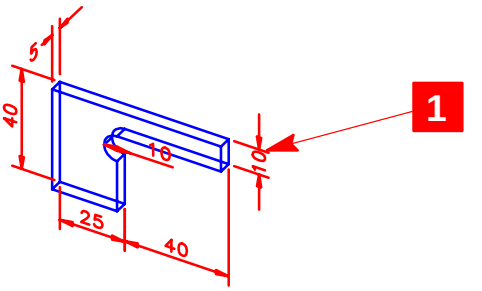
How:



 pick part

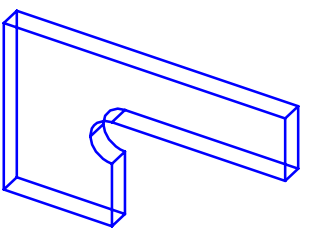
 *Show Dimensions*

1 pick dimension

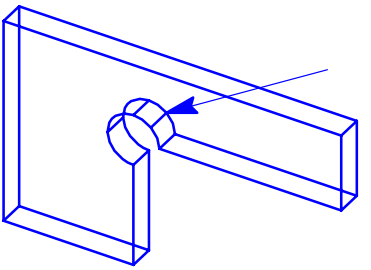


 15

 *OK*



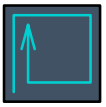
If your part looks more like the one below than the one on the previous page, do not continue. Open your model file to the last save and redo the cutout making sure the stress relief remains tangent in the corner.



Hint
Control-Z

What: If your part is constructed properly, modify the dimension back to 10 before continuing.

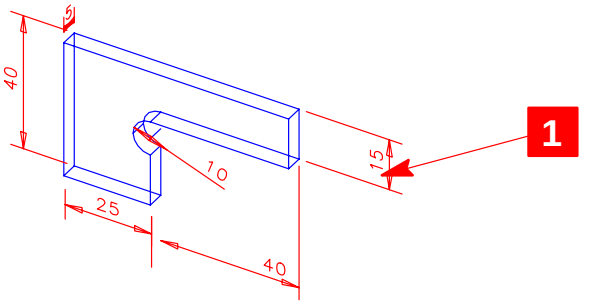
How:



 pick part

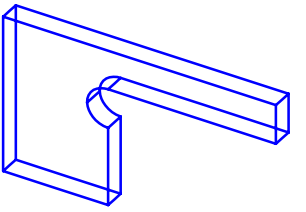
 *Show Dimensions*

1 pick dimension



10

OK



What: Create an FE model associated with the part.

Hint

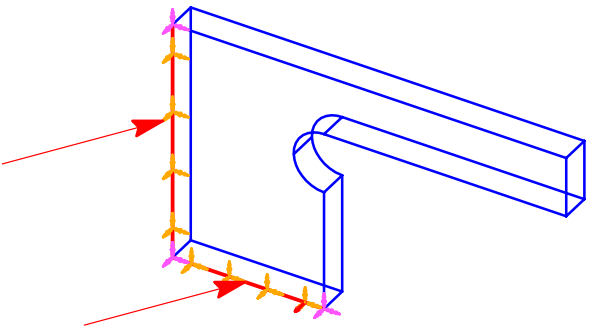
Boundary Conditions

Geometry Based Analysis Only

Warning! Make sure you turn on geometry based analysis, or you won't be able to use geometry parameters for optimization.

What: Fully restrain the two edges shown.

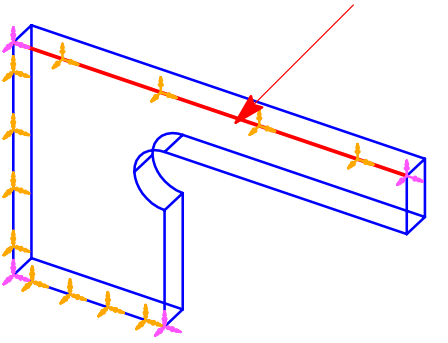
Hint



Why: Since you'll use thin-shell elements to model the part, only the front edge needs to be restrained.

What: Restrain the edge shown to impose symmetric boundary conditions.

Hint




X Translation: *Free*


Z Translation: *Free*


Y Rotation: *Free*


OK

What: Apply a force on the edge shown.

Hint

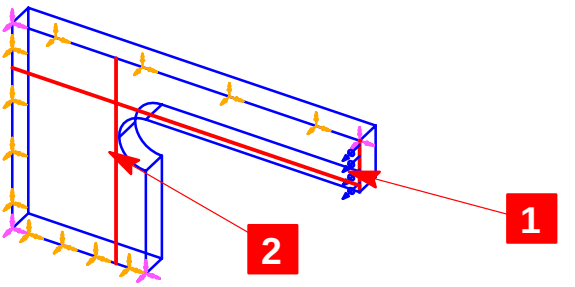
1 pick edge

2 pick front surface

Total Force

Out of Plane Force: –10000

Why: Applying a total force instead of an intensity means that the force will remain constant if the tab width changes. The direction of the force is important if you set a displacement limit as part of an optimization redesign.



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

Hint

What: Create a material for the analysis.

How:


Meshing


Quick Create


Hinge Material


Modulus of Elasticity: 2E6


Modify Value


Mass Density: 1.5E–6


Modify Value

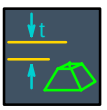

Shear Modulus


Null Property


OK (all forms)

What: Create a physical property defining the thin-shell element thickness.

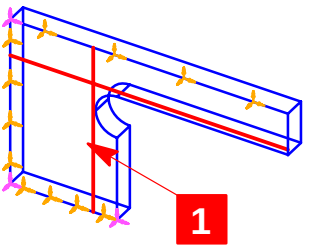
Hint




1st value for thickness: 5

What: Define the mesh for the front surface using thin-shell elements.

Hint



Define Mesh form

Element Length: 4

Free Options...

Percent Deviation

Percent Deviation: 5

OK

Material

Other

Material Selection

Materials form

Hinge Material

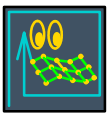
OK

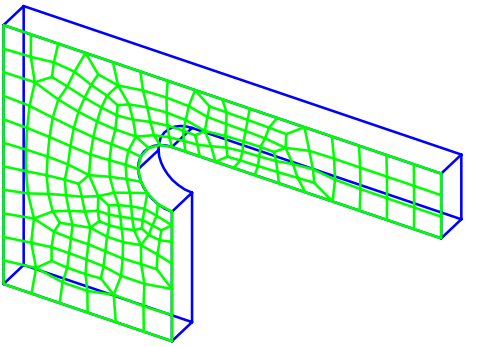
Do not close the Define Mesh form.

What: Generate the mesh on the front surface.

Hint

Define Mesh form





Keep Mesh

Recovery Point

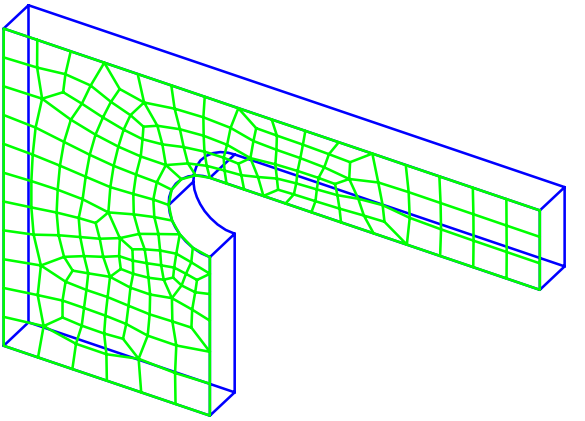
File
Save

What: Create a design to perform an optimization redesign.

Why: A design is the container for a parameter study or optimization redesign. It may contain multiple solution sets, as well as design parameters and limits.

How:

Optimization



Continued on next page...

Manage Designs form

Create

Design form

Name: Redesign Shape

Description: vary length, width, thickness

Math Programming Redesign

OK

Dismiss

Why: This analysis will determine an optimum combination of length, width, and thickness to increase the displacement, while keeping the stress below the desired limit.

Things to notice

The math programming option means that design parameters will be adjusted based on calculated sensitivity values. The fully stressing option can't be used in this case, because we need to place limits on displacement.

What: Create a solution set.

How:

Manage Solutions Sets form

Create

Solution Set form

Output Selection...

Output Selection form

Strain Energies

Store/List

No Output

OK or *Dismiss* (all forms)

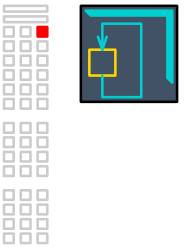
To save file space, you can request stored output for only the first and last iterations. You will still be able to graph the monitored results for each iteration.

Things to notice

Redesign solution sets can include linear statics and normal mode dynamics. If more than one solution set is created, each will be solved for every iteration of the optimization.

What: Define the maximum number of iterations.

How:



Iteration Control form

Maximum Number of Iterations: 8

Redesign Factor (percentage): 20

OK

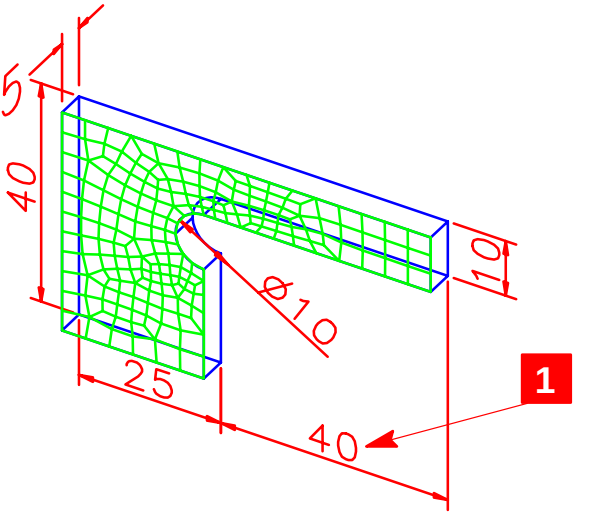
Why: Each defined design parameter will be adjusted in each iteration up to 20% of its current value. If a converged solution is not found in 8 iterations, the program will stop.

Recovery Point

File
Save

What: Create a design parameter for the tab length.

How:



Manage Design Parameters form

Create

Design Parameter form

Design Parameter Type: *Geometry*

Name: Length

Select Dimension

Show Dimensions

1 pick dimension

Upper Limit: 50

Lower Limit: 30

OK

Do not dismiss the Manage Design Parameters form yet.

Create design parameters 2 of 4

What: Create a second design parameter for the thickness defined in the physical property table.

How:

Manage Design Parameters form

Create

Design Parameter form

Design Parameter Type: *Physical*

Name: Thickness

Physical Property: *Thickness*

Upper Limit: 8

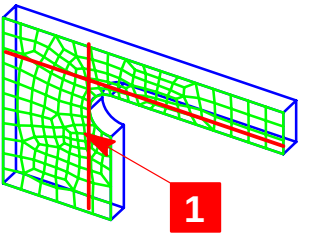
Initial Value: 5

Lower Limit: 3

Select Geometry

1 select front surface

OK



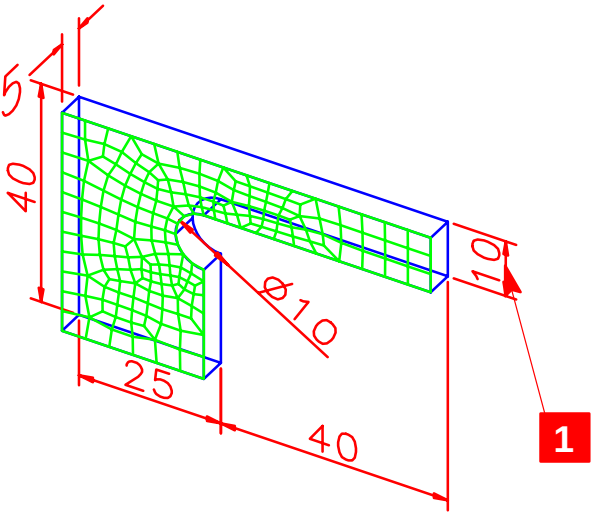
Why: The thickness of the thin-shell elements is defined by the physical property table, not by the extrude distance of the part. If you had used solid elements instead of thin-shell elements, you would select the geometry dimension.

What: Create a third design parameter for the half-width of the tab.

How:

Manage Design Parameters form

Create



Design Parameter form

Design Parameter Type: *Geometry*

Name: Width

Select Dimension

1

Upper Limit: 15

Lower Limit: 5

OK

Create design parameters 4 of 4

What: Verify that there are now three design parameters listed on the form:

- Length
- Thickness
- Width

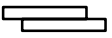
Manage Design Parameters form

 *Dismiss*

Things to notice

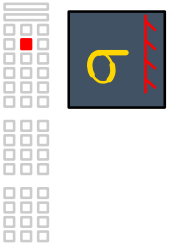
Each design parameter will be varied by the redesign analysis within the given bounds to optimize the design. Other limits on stress and deflection will be defined next.

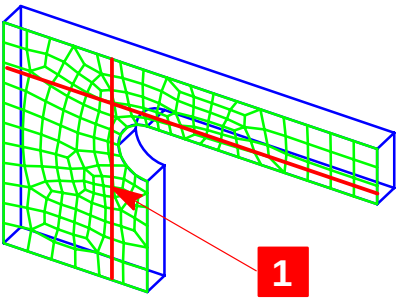
Recovery Point

 *File*
Save

What: Create a stress limit.

How:





Manage Stress Limits form

Create...

Stress Limit form

Maximum Stress: 20000

Select Geometry

1

OK

Dismiss

Things to notice

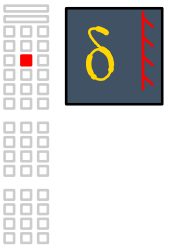
You can set different stress limits in different locations.

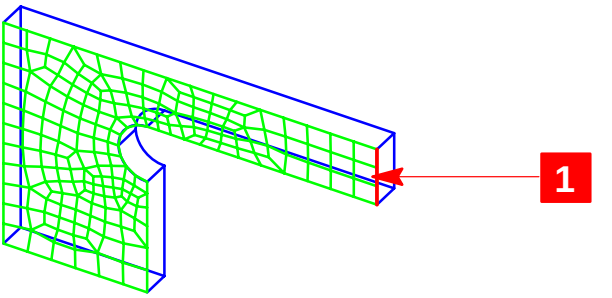
In a parameter study, you can monitor stress at each iteration. In an optimization redesign, you can create limits on stress.

Create limits and design goal 2 of 3

What: Limit the Z component of displacement between two given values.

How:





Manage Displacement Limits form

Create...

Displacement Limit form

Full Form...

Limit Bounds: *In_Band*
Maximum Displacement: 3.2
Minimum Displacement: 2.8
Displacement Direction: *Z*
Select Geometry
1 pick edge

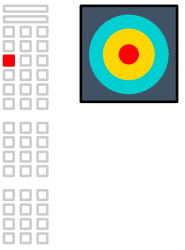
OK or *Dismiss* (all forms)

Why: Often, you want to limit displacement below a maximum value. In this case, we want the displacement to be limited between two values to result in the desired stiffness.

Create limits and design goal 3 of 3

What: Create a design goal to minimize the mass.

How:



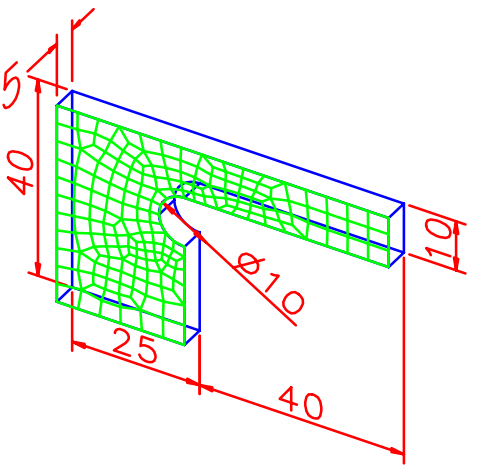
Design Goal form

Type: *Minimize Total Mass*

OK

Things to notice

Minimizing mass means that less material will be used, reducing cost and size.



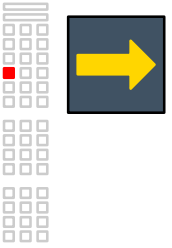
Recovery Point

File
Save

Solve the optimization redesign 1 of 1

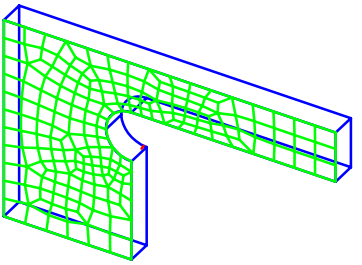
What: Solve the optimization redesign analysis.

How:



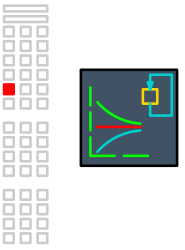
Things to notice

With the given initial values of the design parameters, the displacement is outside the allowable band, so the optimization solver is starting with an “infeasible” solution.

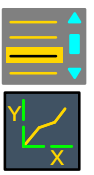


What: Display a graph of the design goal objective function.

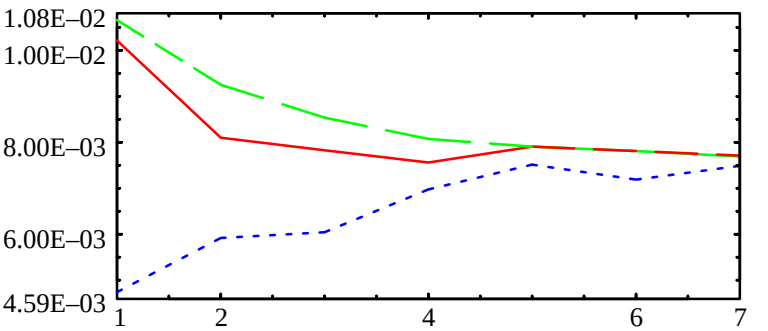
How:



Redesign Histories form



Design Goal



Things to notice

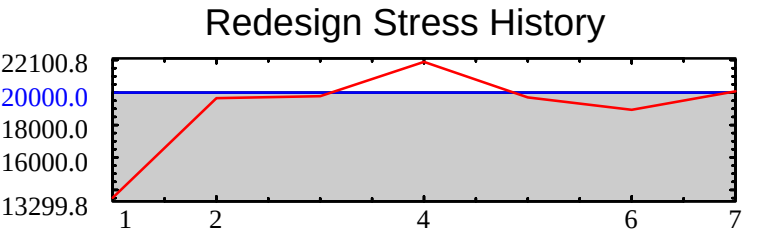
The middle value is the actual mass. The other two values are calculated upper and lower bounds. These values converge as an optimized design is reached.

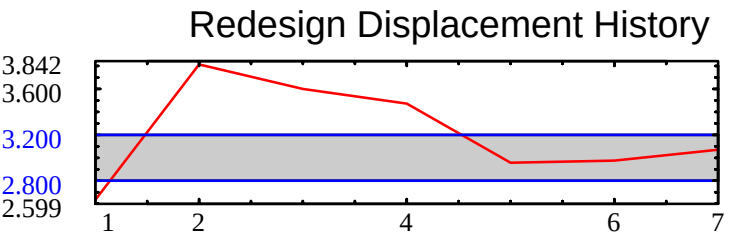
What: Graph the stress and displacement.

Hint


Stress Limit 1


Displacement Limit 1 (Control-pick)





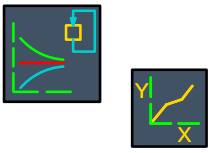
Things to notice

The ranges of the limit values are highlighted on the graphs here (not in I-DEAS) to show how the stress and displacement iteratively converged to within these values.


Dismiss

What: Examine the history of each design parameter.

Hint

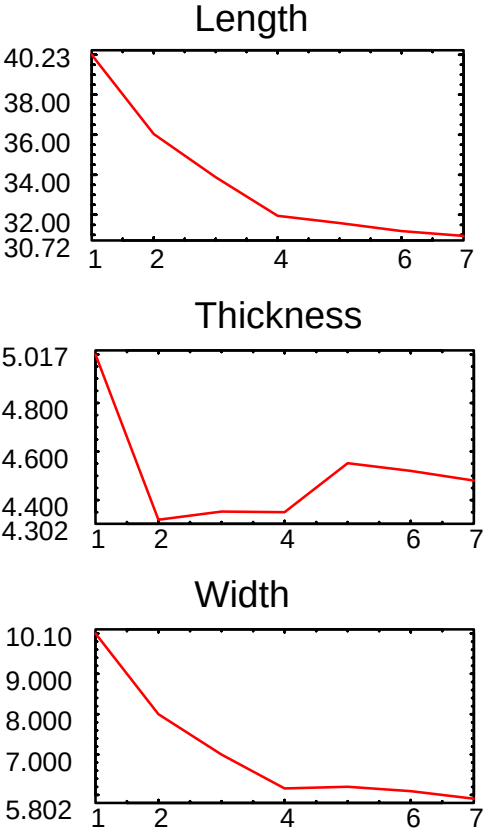


Things to notice

What are the final values of the parameters compared with the starting values?

Which values were limited by upper or lower bounds?

Which parameters converged steadily, and which oscillated before converging on an optimum value?

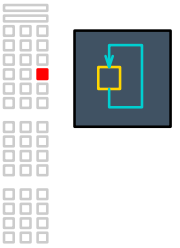


To list specific values in the *I-DEAS List* window, use the *Info* icon on the form.

Display optimization iterations 1 of 1

What: Display the geometry of the part generated during the design optimization iterations.

How:



Manage Iterations form

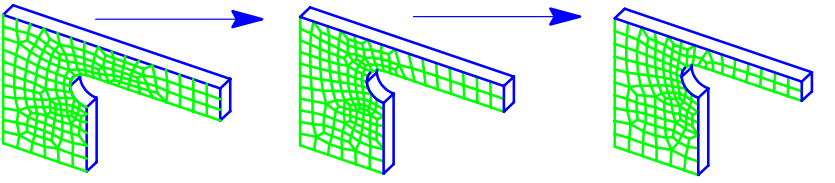
Optimization_Solv1

Current
(continue to view other iterations)

Predicted

Current

Dismiss

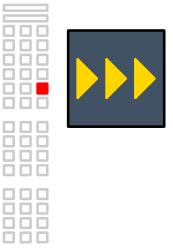


Things to notice

The part doesn't show the thickness change, because the change was made to the finite element physical property, not to the part geometry. If solid elements were used instead of shell elements, this design parameter would change the geometry. Shell elements were used in this example to shorten solution time.

What: Create a new part with the dimensions from the redesign.

Hint



Update Part/Model form

New part

Part: Optimized Hinge

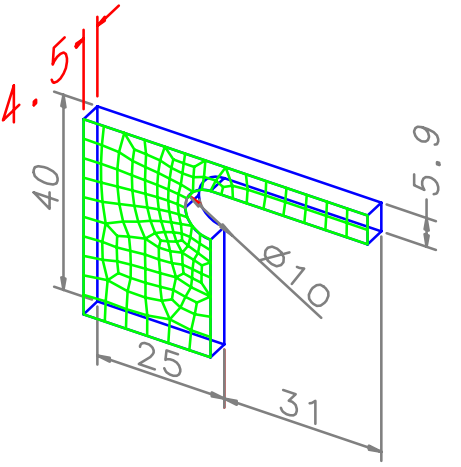
OK

Things to notice

A copy of the FE model is created along with a new version of the part. Using this FE model, you could continue to perform other parameter studies or redesigns starting at this point.

Remember

Since the thickness parameter was an FE-based design optimization parameter, this change is not reflected in the new part unless you manually modify this value.



Tutorial wrap-up

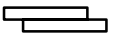
You have completed the Optimization Redesign tutorial.


Don’t delete the FE model or part.

What:
Save your model file if you haven’t done the tutorial “Parameter Studies” yet. This part is used in that tutorial.

See also...

For additional information on the concepts covered in this tutorial, see the following:

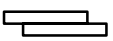

Help, Manuals, Table of Contents

Simulation: Model Solution/Optimization User’s Guide
- Using Optimization
  - Icon Overview for Optimization
  - Overview and Concepts...
  - Getting Started with the Software
  - Defining the Analysis
  - Defining the Optimization
  - Solving the Model
  - Examining Optimization Results
  - Applying Design Changes to the Model

What’s next?

Other Advanced Projects tutorials introduce different element types and solution methods. The Parameter Studies tutorial contains information related to this tutorial.

To exit this tutorial, select:


File
Exit

Warning!
Do not use the menu in the *I-DEAS Icons* window to exit. Use the File, Exit 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