

## 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



Model File name: any unique name

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

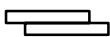


Simulation



Master Modeler

Set your units to mm.



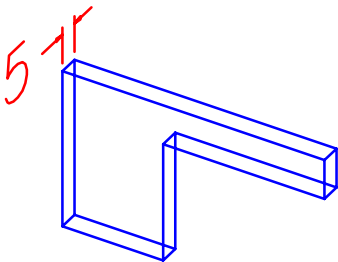
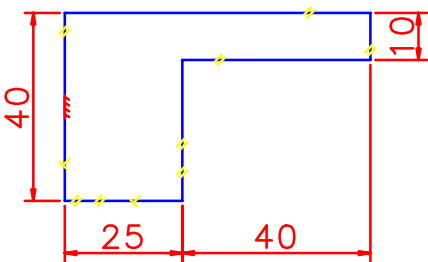
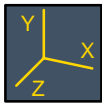
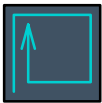
Options  
Units



mm (milli newton)

**What:** Create this part to the dimensions shown.

Hint

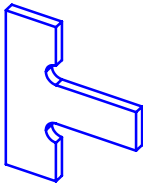


**What:** Name the part “Hinge Snap.”

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.**



**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:**

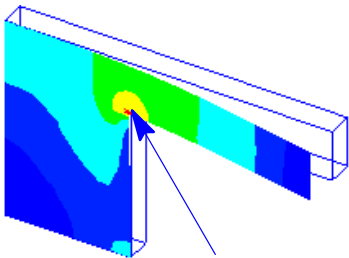
If you make a mistake at any time between saves and can’t recover, you can reopen your model file to the last save and start over from that point.

Hint

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

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.

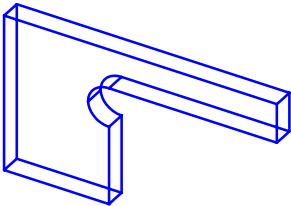
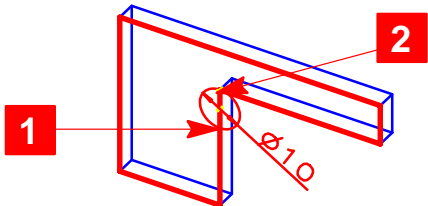
**Hint**

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



1

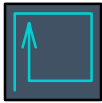
2



**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:**



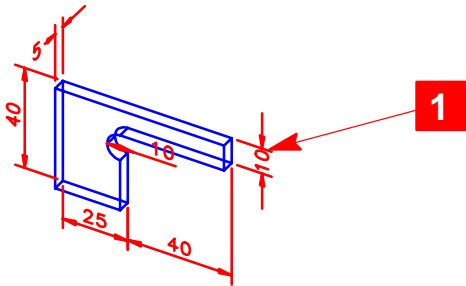
pick part



*Show Dimensions*



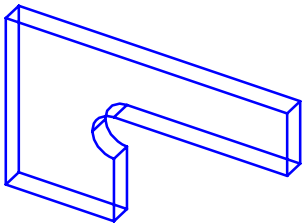
pick dimension



15

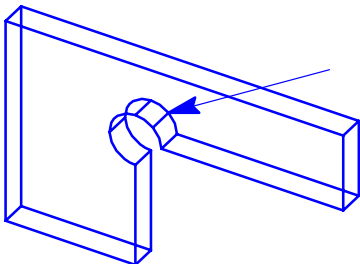


OK





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:

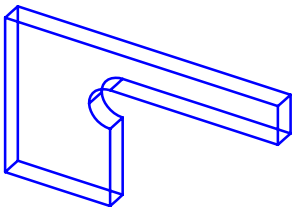
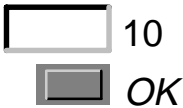
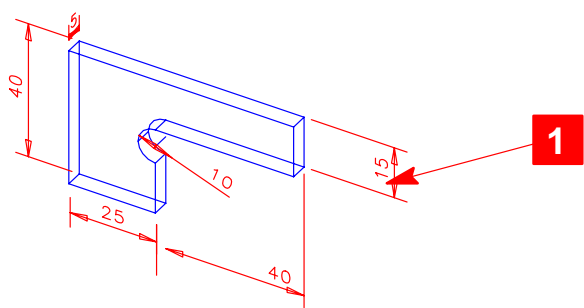


pick part



Show Dimensions

**1** pick dimension



**What:** Create an FE model associated with the part.

**Hint**

 *Boundary Conditions*



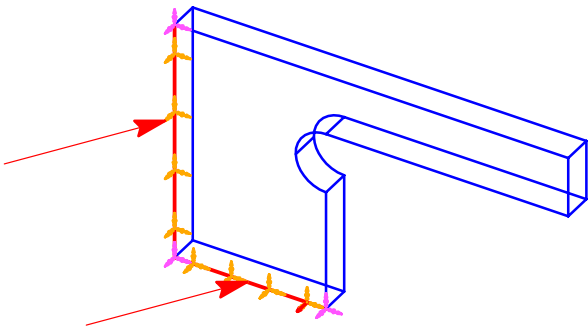
 *Geometry Based Analysis Only*

**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.

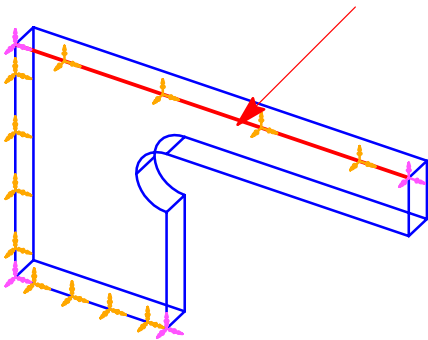
**Hint**



**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

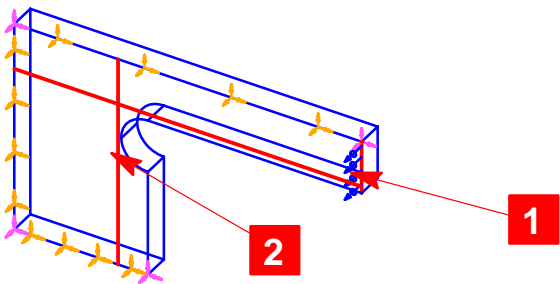


*Total Force*



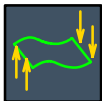
*Out of Plane Force: -10000*

**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**



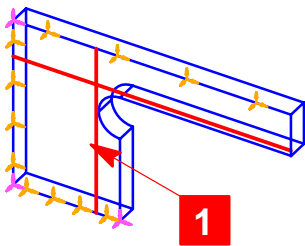
*1st value for thickness: 5*

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

**Hint**



1



Define Mesh form

*Element Length: 4*

☒ *Percent Deviation*

*Percent Deviation: 5*

*Material*

☒ *Other*

Materials form

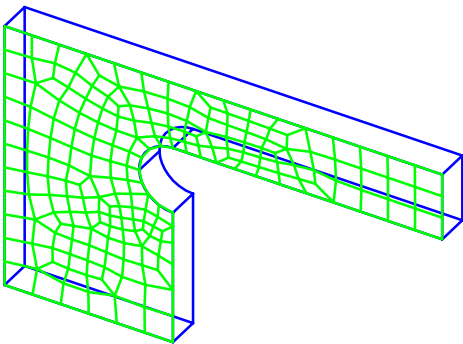
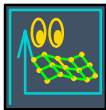


Do not close the Define Mesh form.

**What:** Generate the mesh on the front surface.

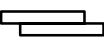
**Hint**

Define Mesh form



*Keep Mesh*

**Recovery Point**

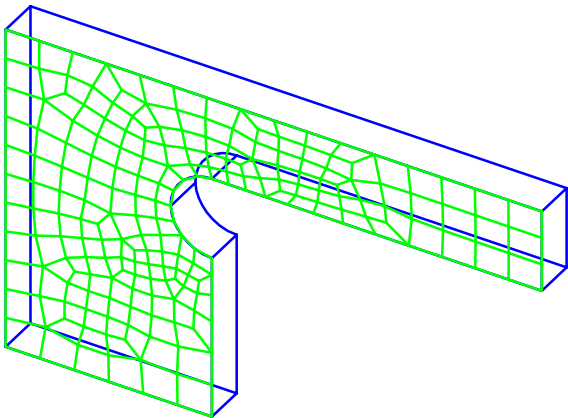
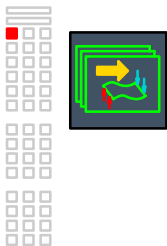


*File  
Save*

**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:**



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.

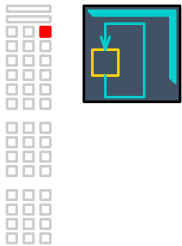
Things to notice

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*

**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.

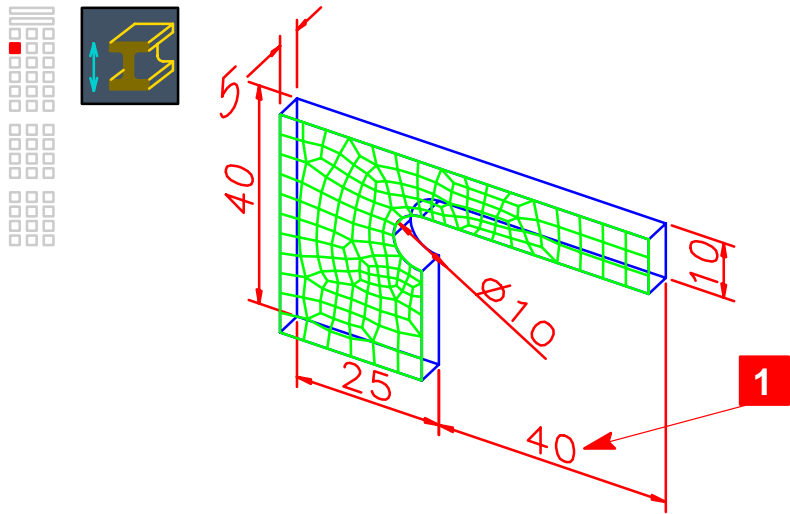
Recovery Point

*File*

*Save*

**What:** Create a design parameter for the tab length.

**How:**



Manage Design Parameters form



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.

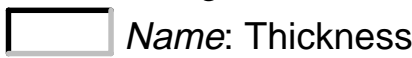
**What:** Create a second design parameter for the thickness defined in the physical property table.

**How:**

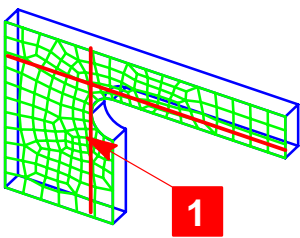
Manage Design Parameters form



Design Parameter form



**1** select front surface

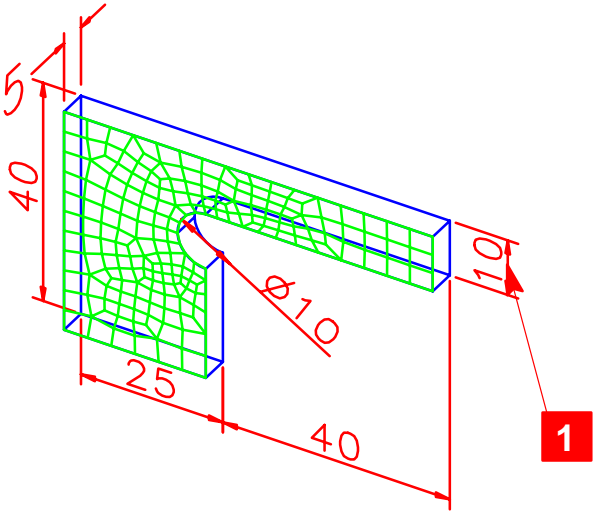


**Why:** The thickness of the thin-shell elements is defined by the physical property table, not by the extrude distance of the part. If you had used solid elements instead of thin-shell elements, you would select the geometry dimension.

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

**How:**

Manage Design Parameters form



Design Parameter form



1



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

- Length
- Thickness
- Width

Manage Design Parameters form



Things to notice

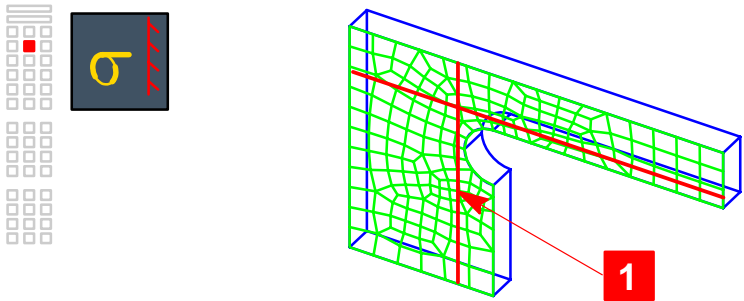
Each design parameter will be varied by the redesign analysis within the given bounds to optimize the design. Other limits on stress and deflection will be defined next.

Recovery Point



**What:** Create a stress limit.

**How:**



Manage Stress Limits form

 *Create...*

Stress Limit form

*Maximum Stress: 20000*

 *Select Geometry*

**1**




 *OK*

 *Dismiss*

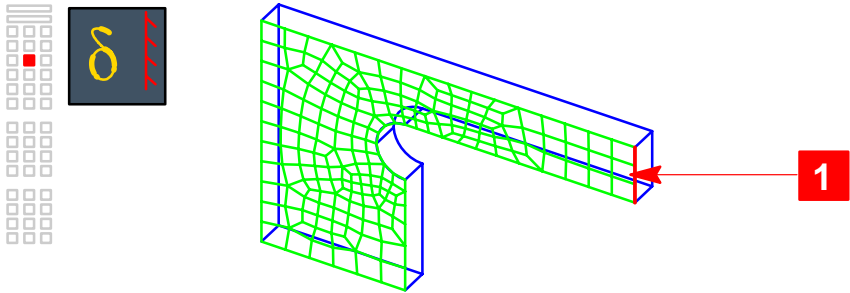
Things to notice

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.

**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

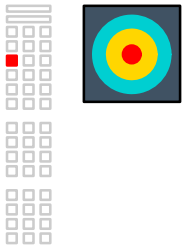


 OK or Dismiss (all forms)

**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.

**What:** Create a design goal to minimize the mass.

**How:**



Design Goal form



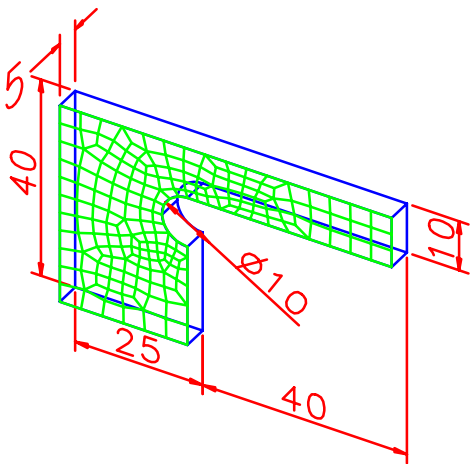
Type: *Minimize Total Mass*



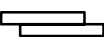
OK

Things to notice

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



Recovery Point

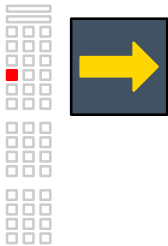


File  
Save



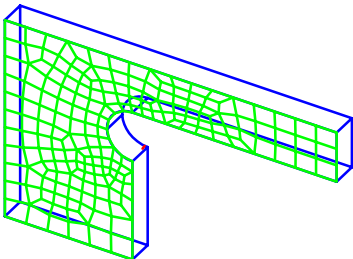
**What:** Solve the optimization redesign analysis.

**How:**



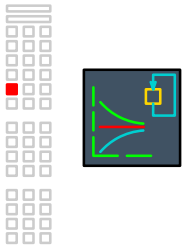
**Things to notice**

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.

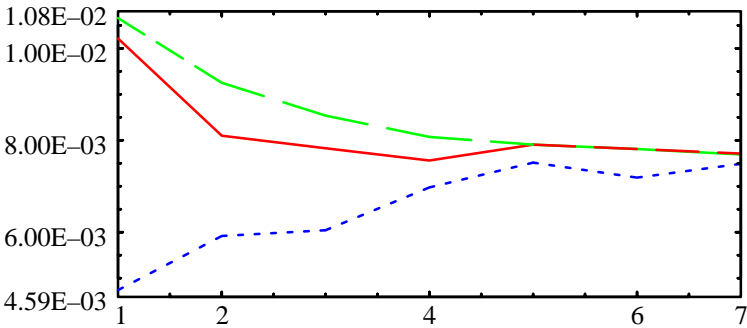
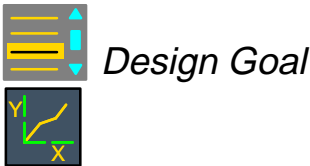


**What:** Display a graph of the design goal objective function.

**How:**



Redesign Histories form



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**



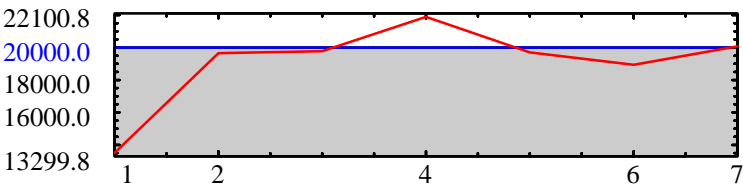
Stress Limit 1



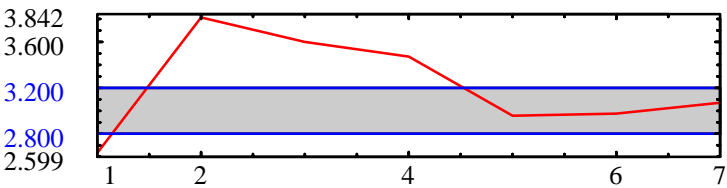
Displacement Limit 1 (Control-pick)



Redesign Stress History



Redesign Displacement History



**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.

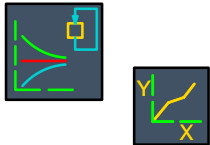


Dismiss



**What:** Examine the history of each design parameter.

**Hint**

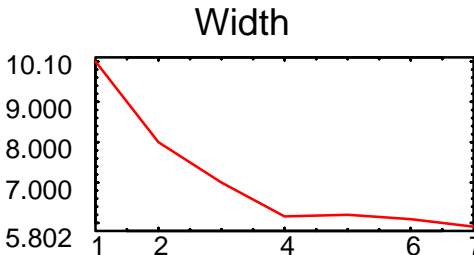
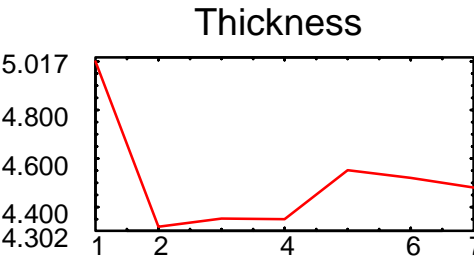
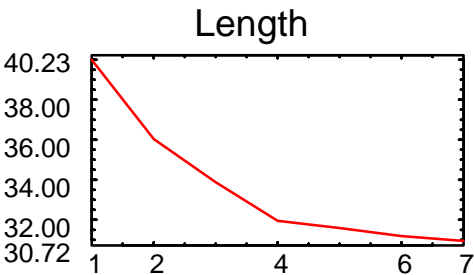



**Things to notice**

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

Which values were limited by upper or lower bounds?

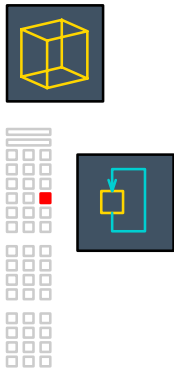
Which parameters converged steadily, and which oscillated before converging on an optimum value?




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

**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)



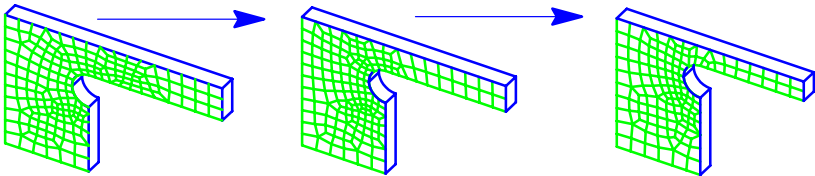
Predicted



Current



Dismiss

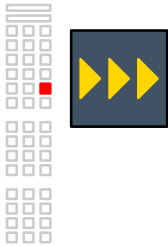


Things to notice




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

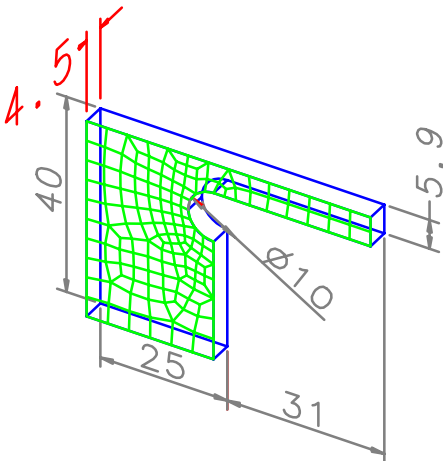
-  *New part*
-  *Part: Optimized Hinge*
-  *OK*

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

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