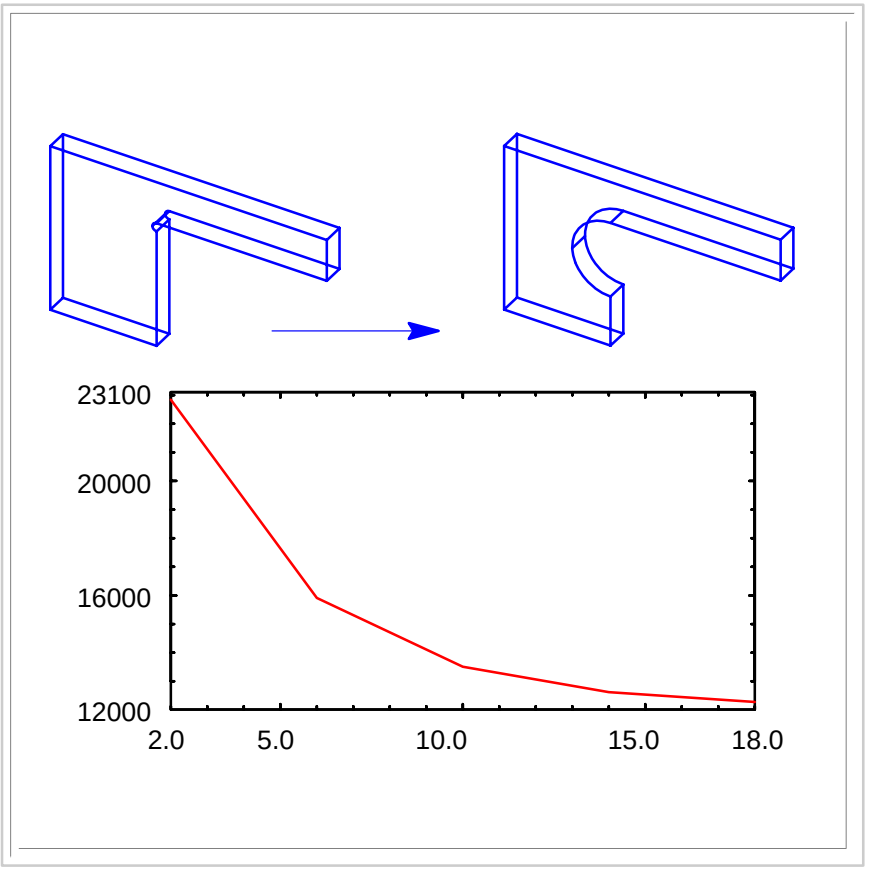


Parameter Studies

I-DEAS™ Tutorials: Fundamental Skills

A parameter study is a quick way to perform a series of solutions using different values as a design parameter. In this tutorial, you'll study the effect of changing the stress relief radius on a hinge snap.

Learn how to:

- create a design
- create a design parameter
- create output monitors
- solve the parameter study
- display results sets
- graph monitor output

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

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

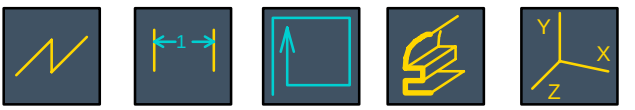
Set your units to mm.

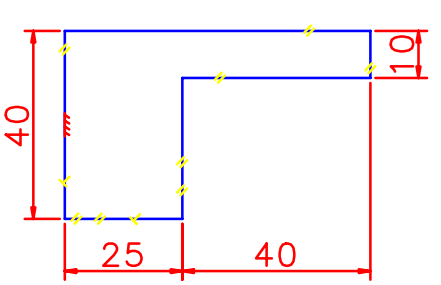
Options
Units

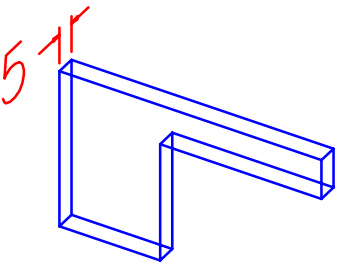
mm (*milli newton*)

What: Create this part to the dimensions shown.

Hint



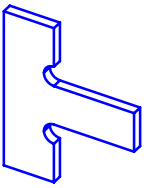




What: Name the part “Hinge Snap.”

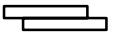
Hint

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*

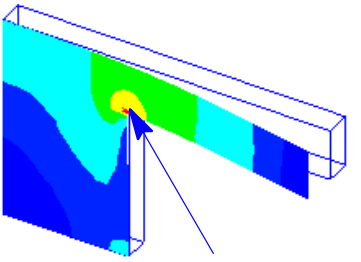
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

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.)

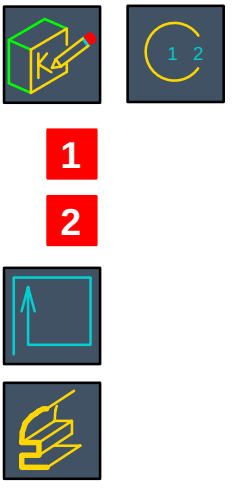
You don't need to perform this analysis. This result is shown only to explain why you're doing the step below.

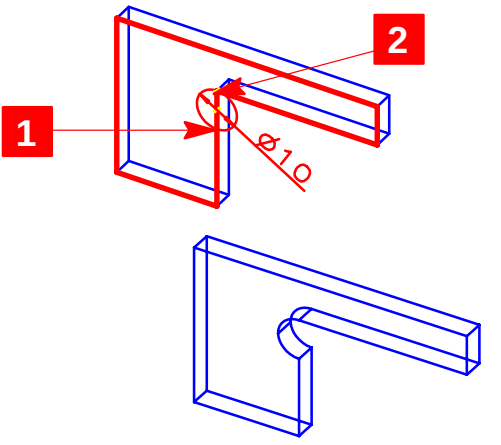


What: Cut a stress-relief in the corner.

Hint

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

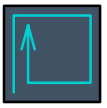




Why: For parameter studies or optimization redesigns, the program will modify dimensions of the part. It is important that the part is constructed properly so it maintains design intent when it is modified.

What: Verify that your part will update properly.

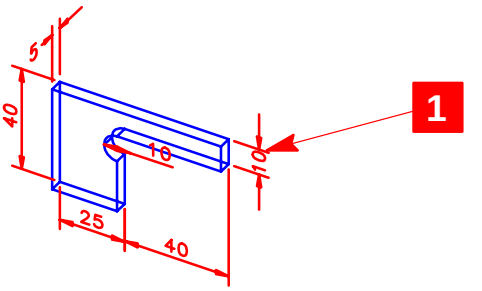
How:



 pick part

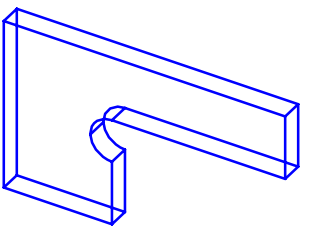
 *Show Dimensions*

1 pick dimension

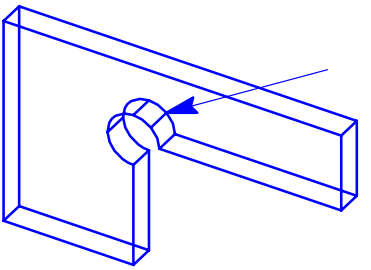


 15

 *OK*



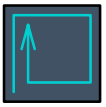
If your part looks more like the one below than the one on the previous page, do not continue. Open your model file to the last save and redo the cutout making sure the stress relief remains tangent in the corner.



Hint
Control-Z

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

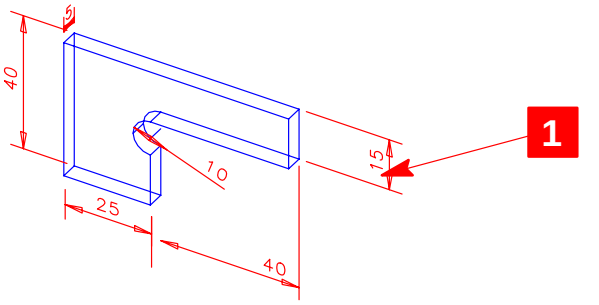
How:



 pick part

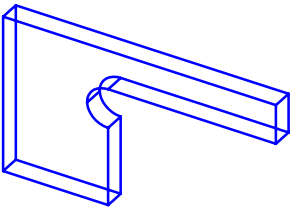
 *Show Dimensions*

1 pick dimension



10

OK



What: Create an FE model associated with the part.

Hint

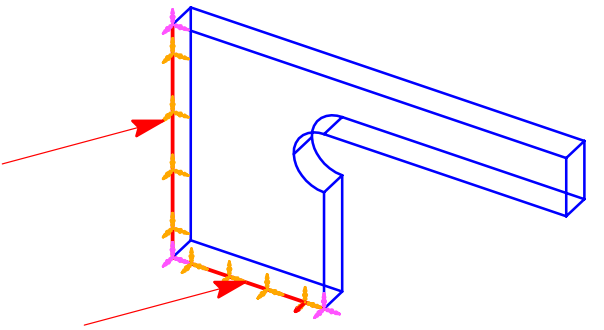
Boundary Conditions

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.

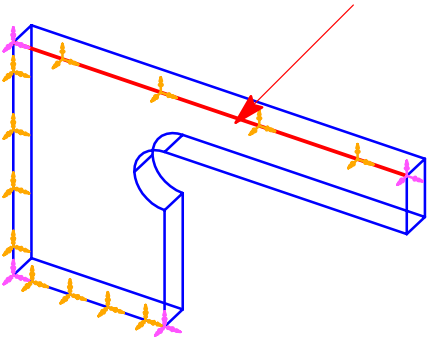
Hint



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

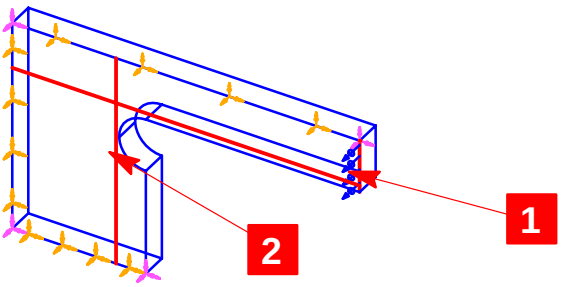
1 pick edge

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

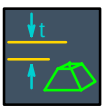

Shear Modulus


Null Property


OK (all forms)

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

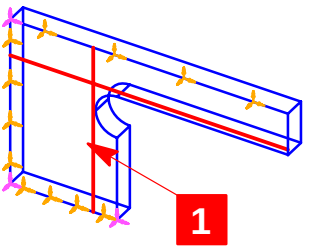
Hint




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

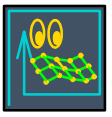
OK

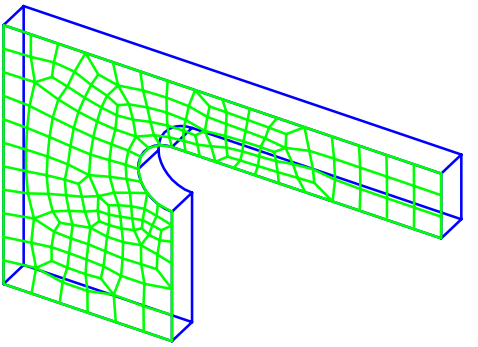
Do not close the Define Mesh form.

What: Generate the mesh on the front surface.

Hint

Define Mesh form





Keep Mesh

Recovery Point

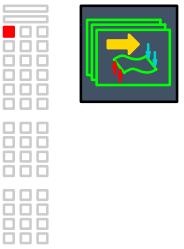
File
Save

What: Create a design to perform a parameter study on the effects of changing the stress relief size.

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



Manage Designs form

Create...

Design form

Name: Radius Study

Description: vary stress relief size

Parameter Study

OK

Dismiss

What: Create a solution set to describe the analysis to be solved at each step of the parameter study.

How:

Manage Solution Sets form

Create...

Solution Set form

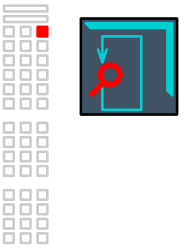
OK

Dismiss

Parameter studies can include linear statics, normal mode dynamics, linear buckling, and heat transfer. You may create more than one solution set to perform different types of solutions for each step of the design variable.

What: Define the number of steps for the parameter study.

How:



Step Control form

Total Number of Steps: 5

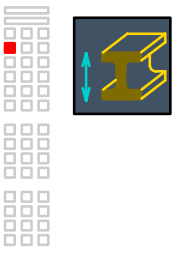
OK

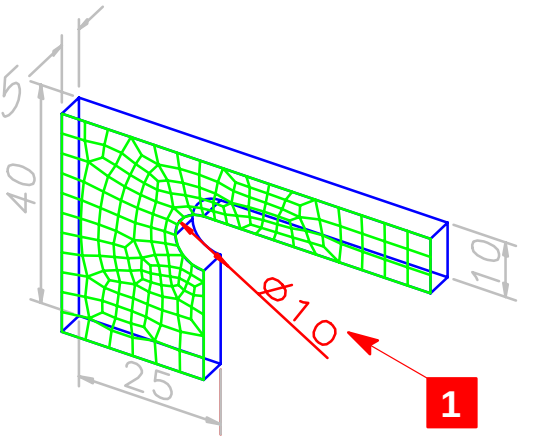
Things to notice

Starting from the present value, the parameter being studied will be changed two steps in each direction.

What: Create a design parameter.

How:





Manage Design Parameters form

Create...

Design Parameter form

Design Parameter Type: *Geometry*

Name: Stress Relief Diameter

Select Dimension

Show Dimensions

1

Step Size: 4

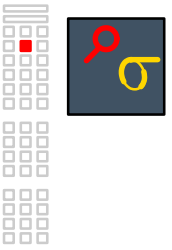
OK

Dismiss

Why: With two steps in either direction (set on the previous page), this step size will result in values of 2, 6, 10, 14, and 18 for the stress-relief diameter.

What: Create a stress monitor to track the maximum stress in all the elements for each parameter step.

How:



Manage Stress Monitors form

Create...

Stress Monitor form

Select Geometry

pick front surface

(*Done*)

OK

Dismiss

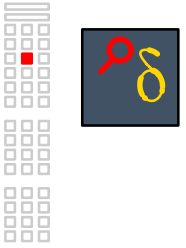
Things to notice

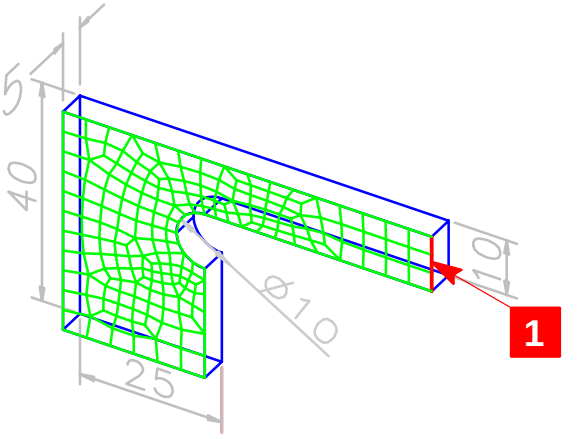
You can create multiple stress monitors to graph the stress in different locations in the model as the design parameter changes.

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

What: Create a displacement monitor to track the displacements for each parameter step.

How:





Manage Displacement Monitors form

Create...

Monitor Displacement form

Displacement Direction: *Z*

Select Geometry

1

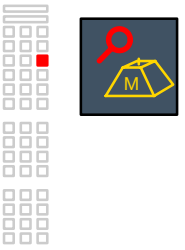
(*Done*)

OK

Dismiss

What: Create a mass monitor.

How:



Manage Mass Monitors form

Create...

Mass Monitor form

Select Geometry

pick front surface

(*Done*)

OK

Dismiss

Things to notice

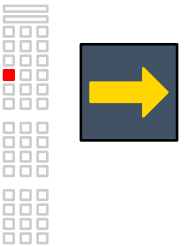
Mass is useful to monitor, since it represents the amount of material used in the part. Reducing material reduces both cost and size.

Recovery Point

File
Save

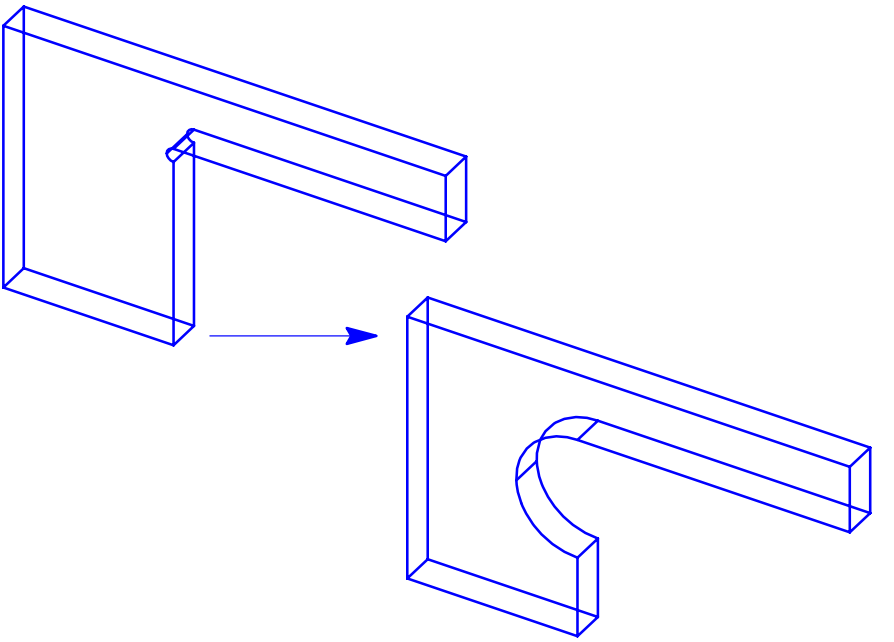
What: Solve the parameter design study.

How:



Things to notice

During the solve, you can watch the part shape change during each step.

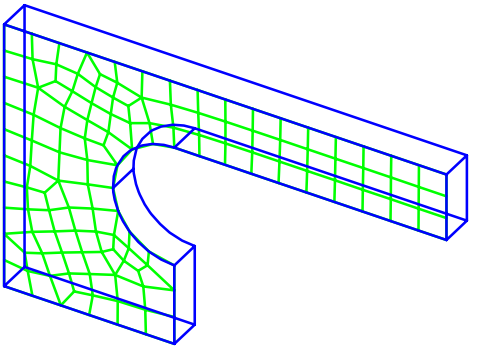


Solve the parameter study 2 of 2

What: After the solve, report any errors and warnings.

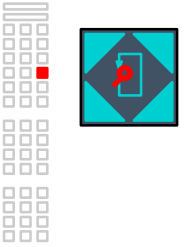
How:

Check *I-DEAS List*.
Notice: The warnings are informational only.



What: Examine some of the stored steps. See how the model and the mesh change with each value of the parameter.

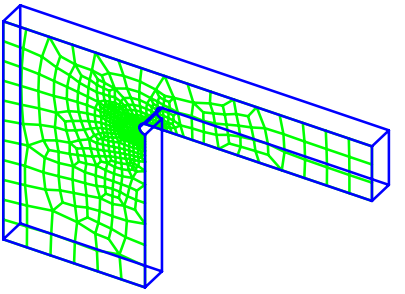
How:



Manage Steps form


Step1

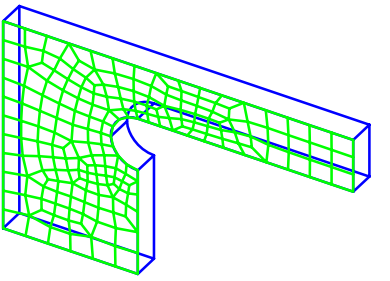

Current




Step3 (*Original*)


Current


Dismiss

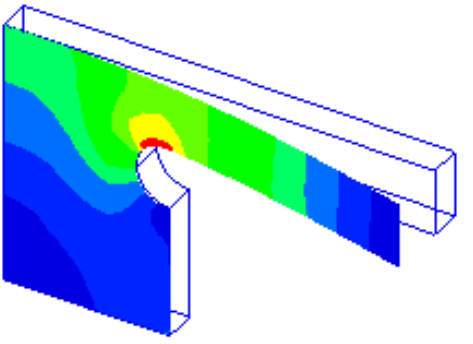


Things to notice

You can display the stress and displacement results for any step containing stored results by making the step current and then switching to the *Post Processing* task.

Hint


Post Processing



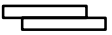
If you want to manipulate the part model, leave the original step as the current step on the display before you switch out of the *Optimization* task. The other step configurations are stored only in *Optimization*, not in the bins like other parts.

What: Switch back to the *Optimization* task before continuing the tutorial.

Hint

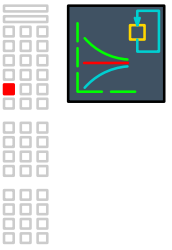

Optimization

Recovery Point


File
Save

What: Display a graph of the stress as a function of the stress-relief diameter.

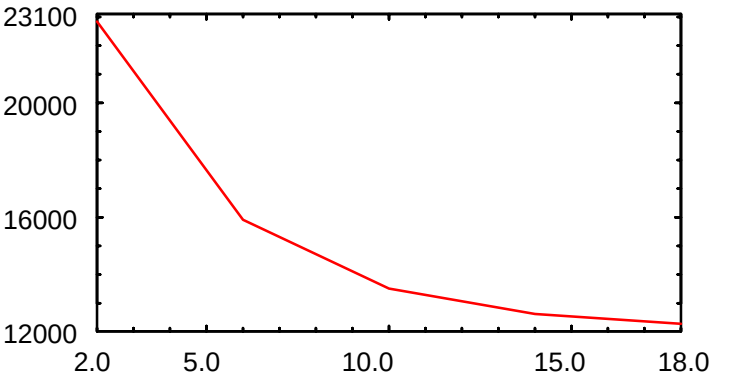
How:



Parameter Study Histories form

Stress Monitor 1

Parameter Study Stress History



Stress Relief Diameter
1 – Stress Monitor 1

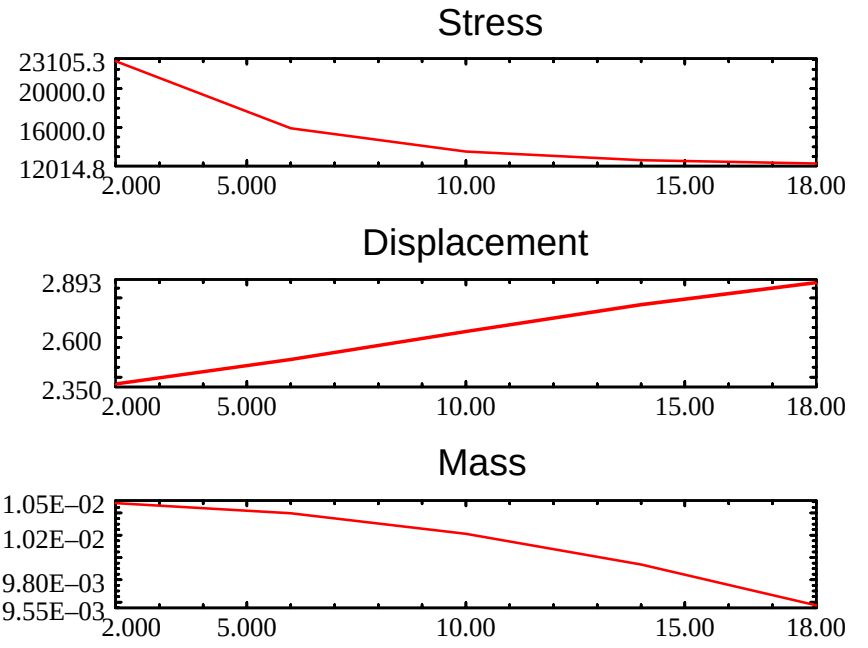
Things to notice

The stress-relief diameter has a large effect on the stress. If the diameter is above 4mm, the stress is below the allowable value of 20,000.

What: Graph all three monitors together.

Stress Monitor 1
Displacement Monitor 1 (Control-pick)
Mass Monitor 1 (Control-pick)

Dismiss



Things to notice

The displacement is lower than the target of 3mm for all values of the diameter. Other parameters will need to be modified to raise the displacement to the desired value. In the tutorial “Optimization Redesign,” you’ll perform a redesign analysis to vary the design parameters to increase this displacement.

Hint

To display the part again, click on any display-type icon, such as *Line* or *Hidden*.

Tutorial wrap-up

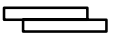
You have completed the Parameter Studies tutorial.


Don’t delete the FE model or part.

What: Make sure you save your model file. This part is also used in the tutorial “Optimization Redesign.”

See also...

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

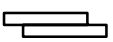

Help, Manuals, Table of Contents

Simulation: Model Solution/Optimization User’s Guide
Using Optimization
Overview and Concepts...
Definitions and Concepts of Parameter Study
Understanding Differences between
Redesign and Parameter Study

What’s next?

Other Advanced Projects tutorials introduce different element types and solution methods. The Optimization Redesign tutorial continues with the part from 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