

Introduction to Simulation

I-DEAS™ Tutorials: Fundamental Skills

Simulation involves three major steps: Pre-processing (modeling, applying boundary conditions, meshing); solving the model; and post-processing (displaying the results).

Learn how to:

- create a finite element model
- apply boundary conditions
- mesh the FE model
- solve the FE model
- display the results

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

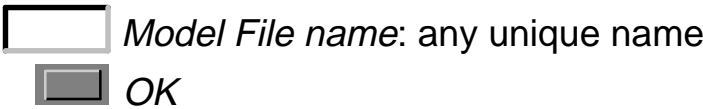
—and—

Creating Parts

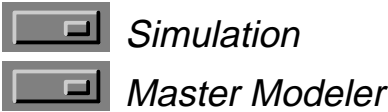
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



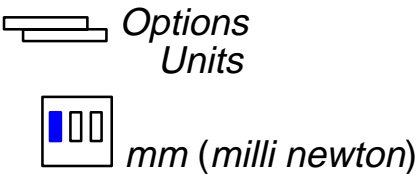
Open Model File form



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

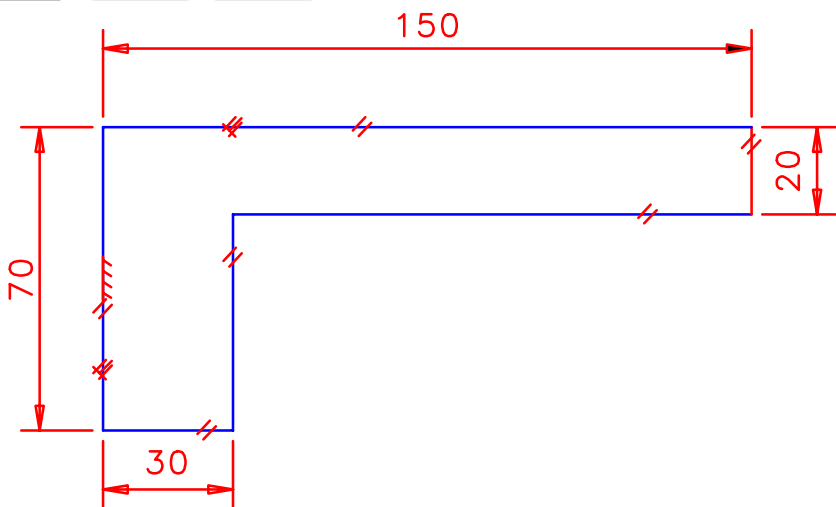
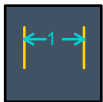


Set your units to mm.



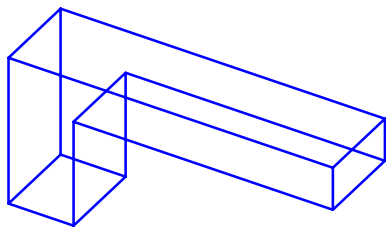
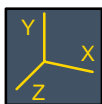
What: Sketch this closed shape to the dimensions shown.

Hint



What: Extrude the wireframe a distance of 40mm.

Hint



What: Name the part.

Hint



Name form

Name: Bracket

Save the 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

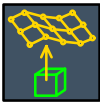
To reopen your model file to the previous save, press Control-Z.

What: Create a finite element (FE) model to associate with the part. An FE model is always associated directly to a part.

How: Change the task to *Boundary Conditions*.



Boundary Conditions



FE Model Create form

FE Model Name: Static Analysis

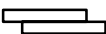


Geometry Based Analysis Only



OK

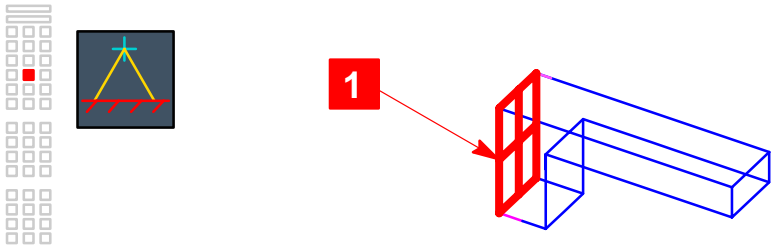
Recovery Point



*File
Save*

What: Fully restrain the rear vertical surface.

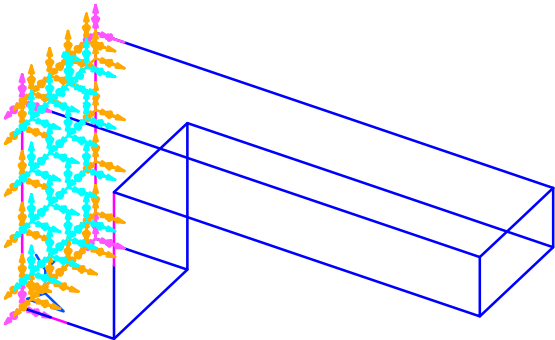
How:



1 pick surface

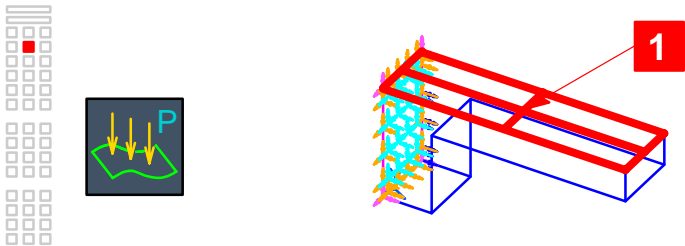


Displacement Restraint on Surface form



What: Create a pressure on the top surface.

How:

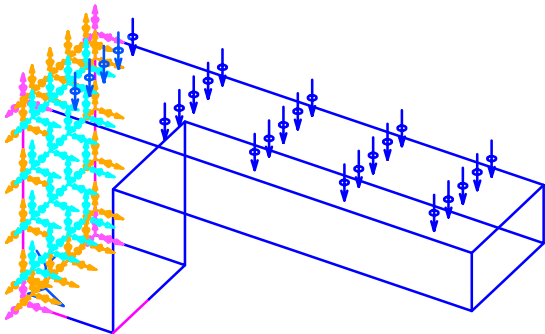


1 pick top surface



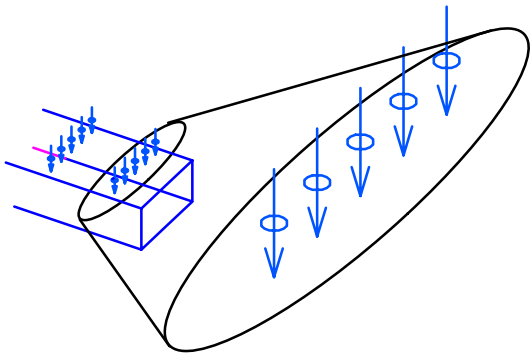
Pressure on Surface form

Pressure: 500



Things to notice

Circles around the arrows mean the pressure is applied to the part geometry.



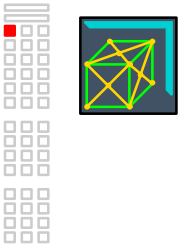
Recovery Point

 *File*
Save

I-DEAS Simulation includes manual and automatic meshing tools. Next, a mesh is automatically generated on the entire part.

What: Define the overall element mesh length for the volume.

How: Change the task to *Meshing*.

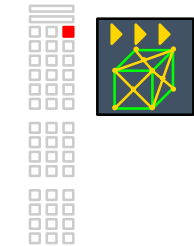



Define Mesh form



What: Generate the solid mesh.

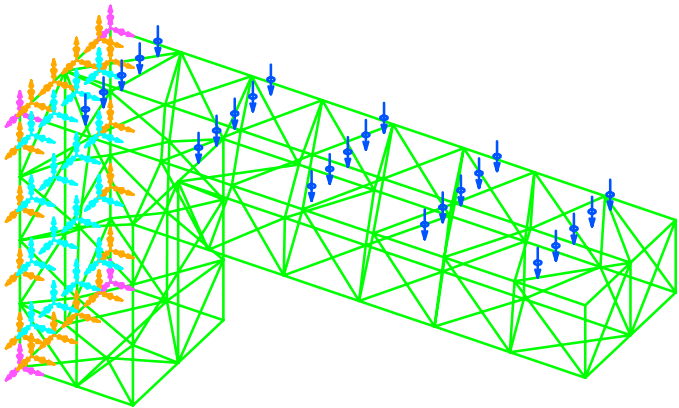
How:



 pick anywhere on part

 Done

 Yes



Things to notice

The elements are parabolic tetrahedral elements. These are more accurate than linear tetrahedral elements for structural analysis.

Recovery Point



What: Create a solution set (accepting all defaults) and solve the model using linear statics for deflection and stress.

How:

 *Model Solution*



Manage Solution Sets form

 *Create...*

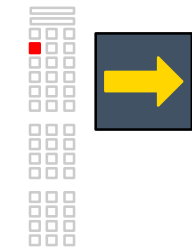
Solution Set form

 *OK*

 *Dismiss*

What: Solve the model.

How:



Check *I-DEAS List*.

When the solve is finished, the *I-DEAS List* region displays any warnings or errors.

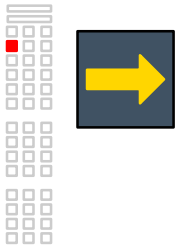
When the solver finishes, change the task to *Post Processing*.



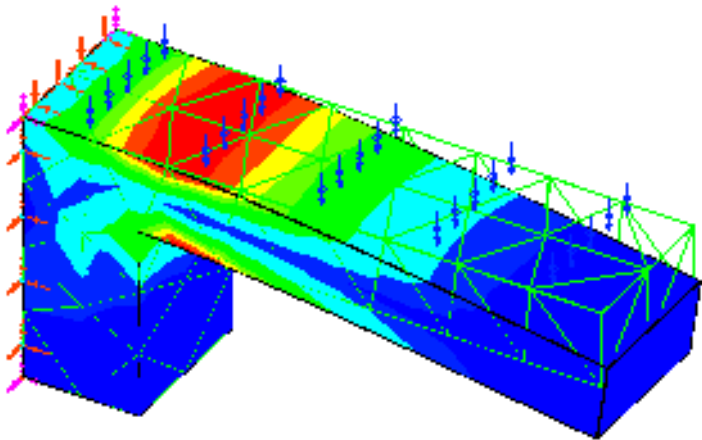
Post Processing

What: Display the default set of displacement and stress results on the deformed geometry.

How:



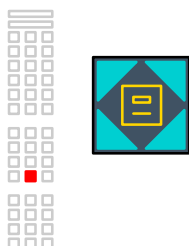
(the default displays all elements)



You have completed the Introduction to Simulation tutorial.

Delete the part. It is not used in any other tutorial. To delete the part, first delete the finite element model.

How:



Manage form



Bracket... (double-click)



Static Analysis (select)



Bracket (select)



Dismiss

See also...

For additional information on many of the concepts covered in this tutorial, see the following in the I-DEAS *Help* facility:

 *Help, Manuals, Table of Contents*

Simulation: Finite Element Modeling User's Guide

Simulation Overview

Using Simulation Tools

Overview

Creating an FE Model on a Part

Creating an FE Model from an Assembly

Managing Models in Simulation

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

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