

Simulation Troubleshooting

I-DEAS™ Tutorials: Fundamental Skills

This tutorial will help you diagnose some of the common problems you may encounter when using Simulation.

Learn how to:

- manage FE models
- diagnose solver errors
- modify FE models
- solve display problems
- improve selection skills
- run verification checks
- delete files

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:

- Master Modeler Troubleshooting

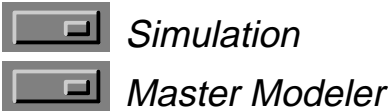
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



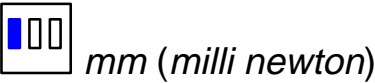
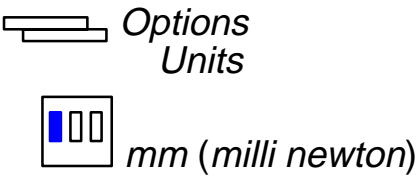
Open Model File form

Model File name: any unique name

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

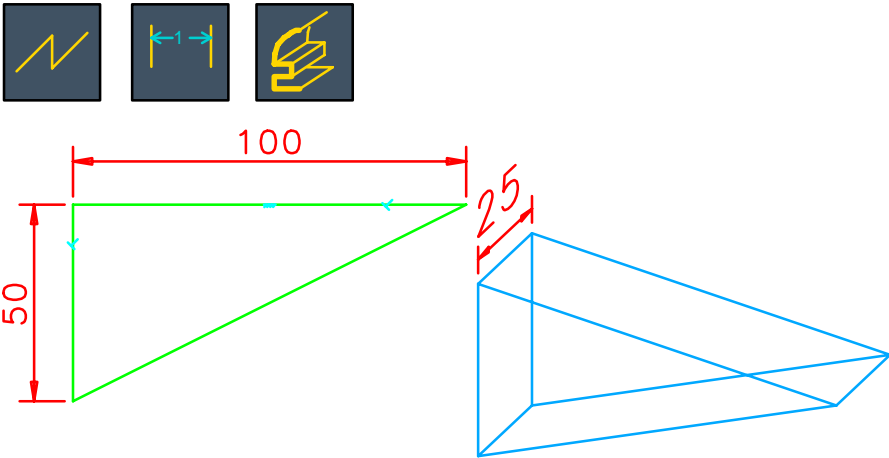



Set your units to mm.



What: Create a simple part as shown.

Hint



 For the purposes of this tutorial, do not name the part yet.

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.

What: Create an FE model, entering a name in the part field.

Hint



Boundary Conditions



FE Model Create form

Part or Assembly: Bracket1

FE Model Name: (click in field)

I-DEAS Warning form



OK

Things to notice

By entering a new part name, you're creating a new part; you are not working with the part you see on the workbench.

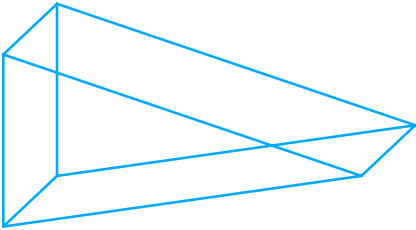
FE Model Create form

FE Model Name: Fem1

Things to notice

The *Geometry Based Analysis Only* toggle is grayed out, because the newly created part has no geometry (edges and faces).

OK



What: Try to apply boundary conditions.

Hint

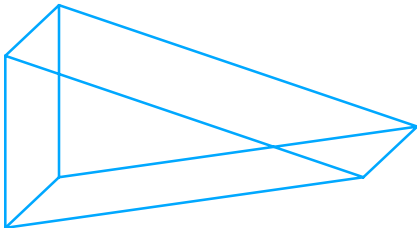


Things to notice

No faces or edges are pickable because the visible part is not associated with the current FE model on the workbench.

What: Name the part that is visible on the workbench.

Hint



pick part



Name: Bracket2



OK

What: Look at the Manage form.

Hint



Manage form



Bracket1... (double-click)

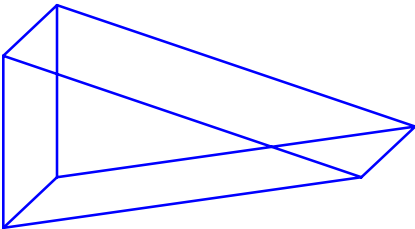
Things to notice

There are two parts listed. Bracket1 has an FE model (named Fem1) associated with it.

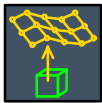


Dismiss

What: Create an FE model associated with Bracket2, the visible part.



Hint



FE Model Create form



pick Bracket2 part



Check I-DEAS List.

Notice the name of the selected part in the list window.

FE Model Name: Fem1

☐

Geometry Based Analysis Only

OK

What: Verify that the other (null) part was put away when you created the FE model.



All



Check I-DEAS List.

Notice that there is only one part on the workbench.



Deselect All

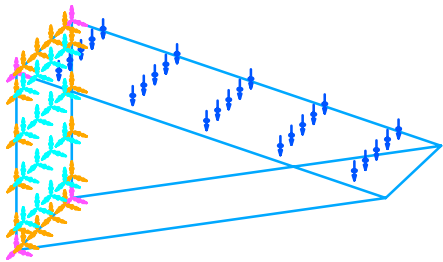
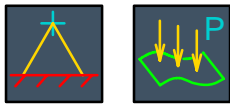


If you can't select part geometry to mesh or apply boundary conditions, make sure your FE model is associated with the part you are trying to select.

In the next few steps, you'll create results to be used in the remainder of the tutorial.

What: Restrain the left face and apply a pressure load to the top face. Accept all defaults.

Hint



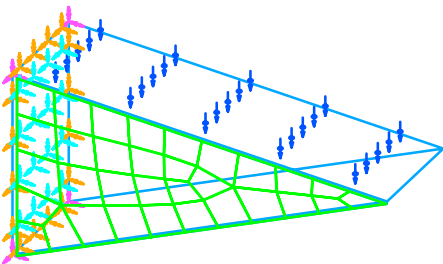
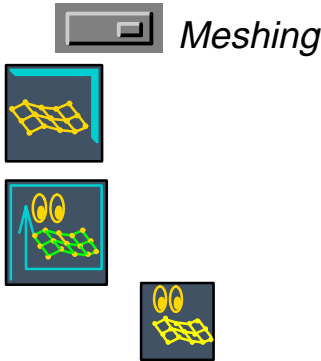
What: Create a boundary condition set.

Hint



What: Mesh the front face.

Hint



Things to notice

There is an intentional modeling error in the steps above. Do you know what it is? (Answer is on page 13.)

Recovery Point



What: Solve the model for linear statics.

Hint



What: Check for errors.

Hint



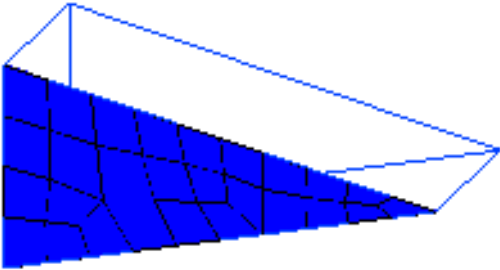
 **Check I-DEAS List.**

What: Display the stress and displacement results.

Hint



Post Processing



Things to notice

Although there were no errors, and valid result sets have been calculated and stored, all displacements and stresses are zero. Why? (Answer is on next page.)



Pressure applied to a face does not apply loads to elements meshed on adjacent faces, even if sharing a common edge. This is not true for restraints.



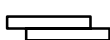
Other errors are reported as singularity errors. These can be caused by:

- insufficient restraints. (Structure not supported in all six DOF.)
- improper physical properties. (Entering a zero value for thickness for one or two of the four values.)
- lack of connections between elements. (Gaps with duplicate nodes.)

For more information on these types of errors, read the articles listed at the end of this tutorial.

What: Even though these results are useless, save the model file.

Recovery Point



File

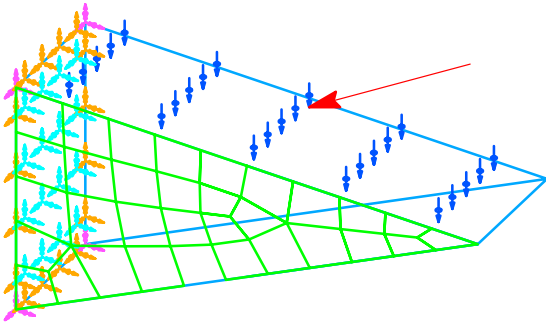
Save

What: Try to delete the face pressure.

Hint



Boundary Conditions



Things to notice

You can't delete the load while it's referenced by stored results. To modify the boundary conditions, you may either:

- delete the results
- create new load and boundary condition sets, or
- create a new FE model



Continue

What: Make a copy of the FE model associated with Bracket2.

Hint



Bracket2... (double-click)



Fem1 (under Bracket2)



Name: Fem2



OK



Fem2



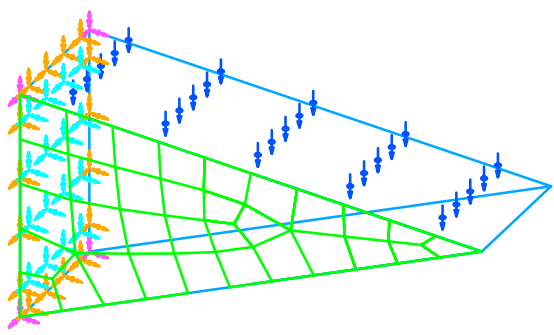
Get



Dismiss



By making a copy rather than creating a new FE model, you save the steps of creating boundary conditions and re-meshing. The copy also contains any solved results.



What: Verify that FEM2 is the active FE model on the workbench.

Hint



FEM1

Things to notice

When FEM1 is selected, the Put Away button is grayed out, meaning the FE model has already been put away.



FEM2

Things to notice

When FEM2 is selected, the Get button is grayed out, meaning the FE model is already on the workbench.



Dismiss

What: Delete the stored results.

Hint

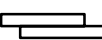


Post Processing



All

Recovery Point



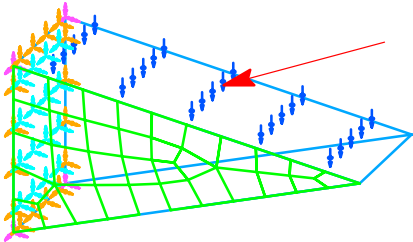
*File
Save*

What: Delete the face pressure, and create an edge load.

Hint



Boundary Conditions



1 pick edge



2 pick front face



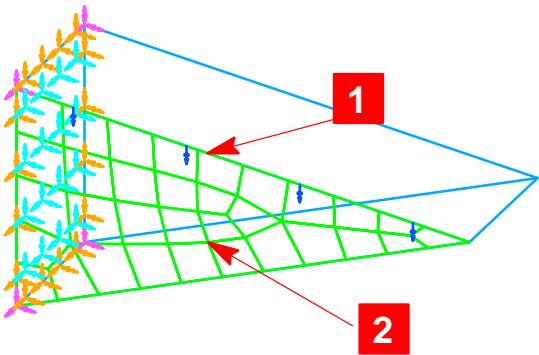
Total force



In Plane Force: 1000



OK



What: Verify that a boundary condition set still exists, containing the restraint and load.

Hint



What: Solve the model.

Hint



Model Solution

Remember

A solution set already exists, since the model was copied.

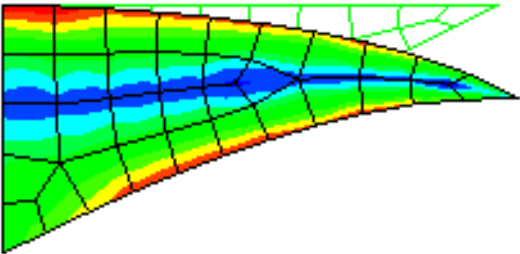


What: Display the results.

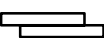
Hint



Post Processing



Recovery Point



*File
Save*

What: Try to modify physical properties.

Hint



Meshing



Directory



Thin Shell



List Owing Model Entities



OK



Yes

Things to notice

The physical property table can't be modified. It's used in models that contain stored results, which lock the models.



Continue

What: Try to modify the load.

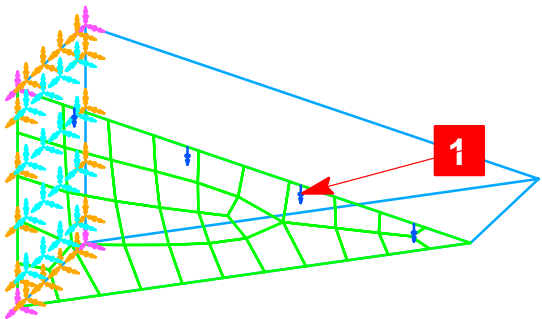
Hint



Boundary Conditions



1 pick load



Things to notice

The load can't be modified because it's used by a locked boundary condition set.

What: Try to modify the boundary condition set.



Restraint Set



OK

Things to notice

The boundary condition set can't be modified while there are stored results.



OK



Cancel

What: Delete the results.

Hint



Post Processing



All



Remember

While results are stored, you can't change elements or solution sets. Doing this would make the results invalid.



There is a hierarchy of results that prevents you from making changes:

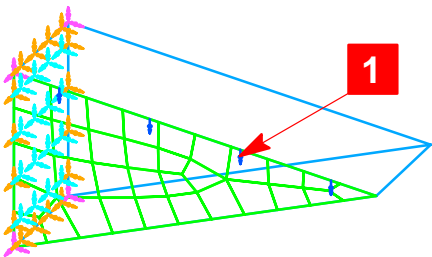
- Result sets
 - Solution sets
 - Boundary condition sets
 - Load sets
 - Restraint sets
- Elements
 - Physical property tables
 - Material property tables

What: Try to modify the load.

Hint



Boundary Conditions



1 pick load

Force on Edge form



OK

Things to notice

You're allowed to modify the load now that the results have been deleted.

What: Try to modify physical properties.



Meshing



Directory



Thin Shell



Yes

Things to notice

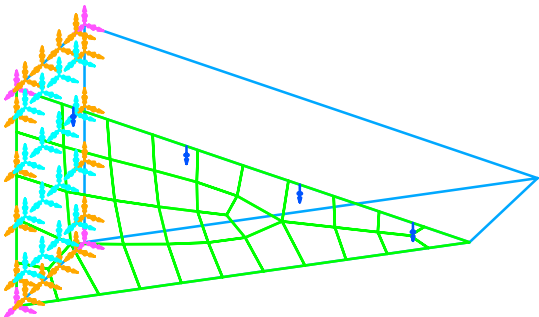
You're still not allowed to modify the physical properties because they're used in Fem1, which still contains results.

What: Try to modify the part dimensions.

Hint



Master Modeler



pick part

Things to notice

You aren't allowed to modify the part. It's locked because Fem1 referring to it contains stored results.

What: Delete Fem1.



Fem1 (under Bracket2)



Things to notice

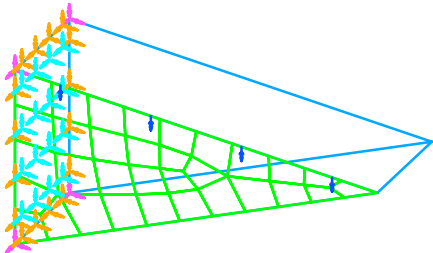
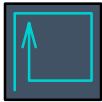
Fem1 has been deleted.



Dismiss

What: Modify the part length dimension.

Hint



pick part



Dimension Values



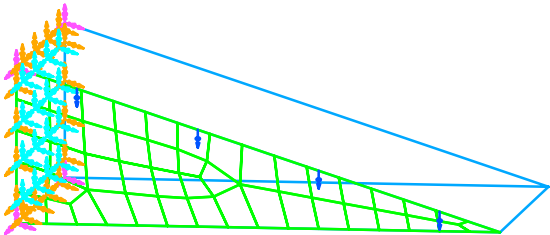
100



150



OK



Things to notice

Both the part and mesh are updated.

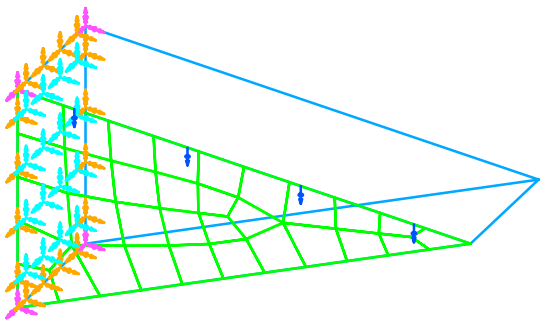


To speed up multiple part changes, put away the FE model before modifying the part. This avoids the time of re-meshing with each change. The mesh will be updated when you get the FE model onto the workbench.

What: Open your model file to the last save.

Hint

Control-Z



What: Slightly move an edge of the Graphics Window.


Things to notice

Move the mouse cursor over the screen, and notice that entities are not selectable.

What: Re-display the graphics.

Hint



 Whenever you have difficulty picking graphical entities, execute a re-display to see if that fixes the problem.

What: Turn off the display of nodes.

Hint

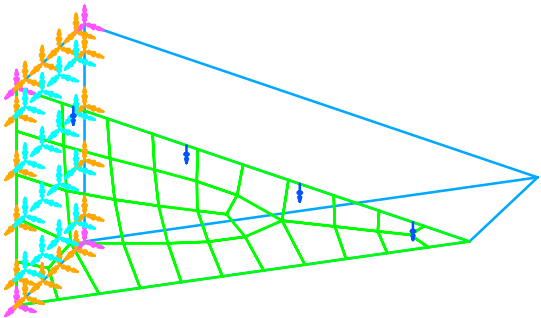


- ☐ *FE Models...*
- ☐ *Node (toggle off)*
- ☐ *OK (all forms)*

What: Try to list the coordinates of a node.


Hint

- ☐ *Boundary Conditions*



Things to notice

Nodes aren't pickable because they aren't displayed.

 To be able to pick an entity, it must be toggled on in the display filter.

What: Display the stress results.

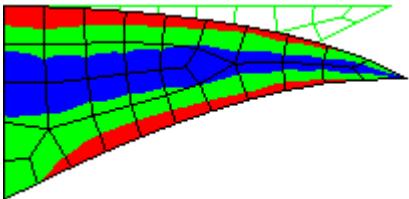
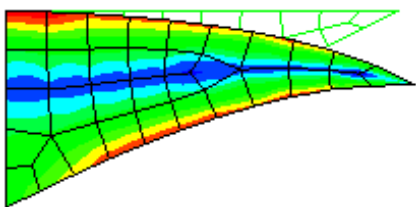
Hint



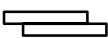
Post Processing



Your display will look like either:



If your display shows only three colors (as on the right), change this preference setting:



*Options
Preferences*



Display



Double buffering



Why:

On some displays (X3D), color planes are used to give smoother animation using double buffering. Turning off this toggle gives better color displays with a trade-off in quality of animation.

How to use “Related to” options

What: Display the stresses of all elements related to the left edge.

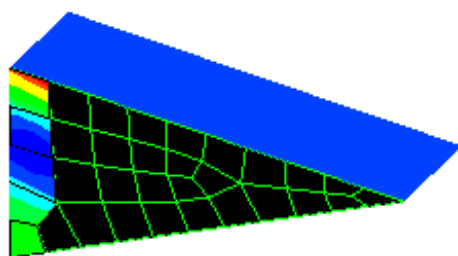
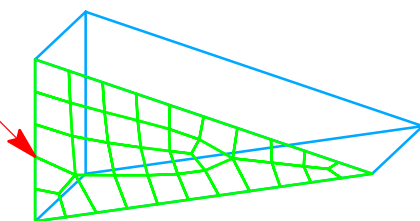
How:



Related To

1

1



You can use this function to pick all elements or nodes on a surface to apply boundary conditions.

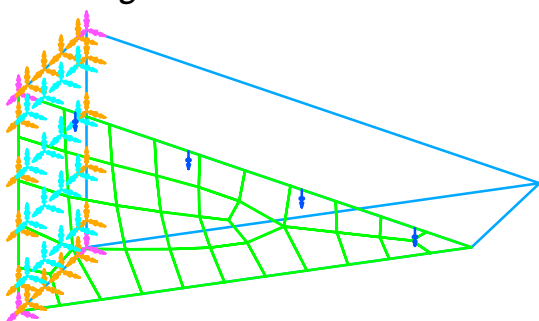
How to use the selection filter

What: Set the selection filter to pick only elements.

Hint



Meshing



Things to notice

As you move the cursor, all entities are selectable.



Filter...



Element



Pick Only

Things to notice

Now, as you move the cursor, only elements are selectable.



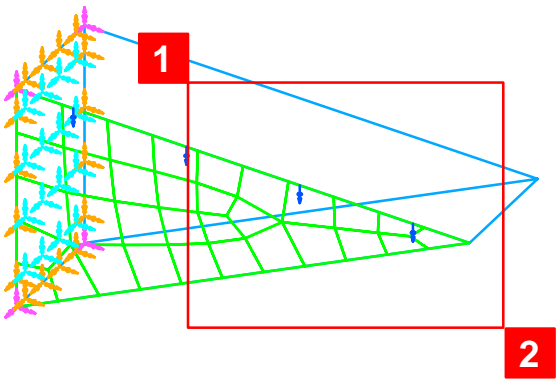
You can use the filter to limit what you want the software to select.

What: Preselect a group of elements in a rectangle.

How:

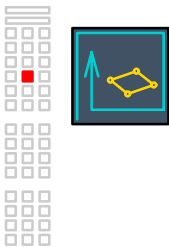
1

2 drag a box



What: Modify the color of the selected elements.

How:



Color



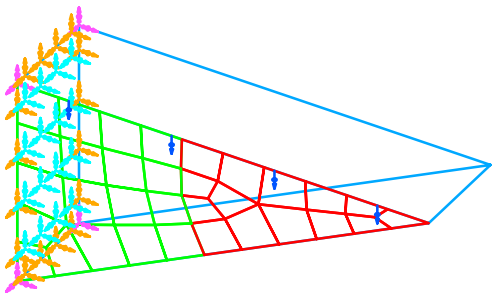
Directory



Red



Yes



What: Reset the filter to pick all entity types.

How:



Filter...

Non-Pickable



Select All



OK



Pre-selecting sometimes works differently than post-selecting, because each icon sets the filter to pick the types of entities that it uses.

How to pick elements by an attribute

What: Display all the green elements.

How:



Deselect All



Filter...



Element



Attributes...



Color



Green

Things to notice There are many other attributes you can use to select elements, such as element type, material, and other values.



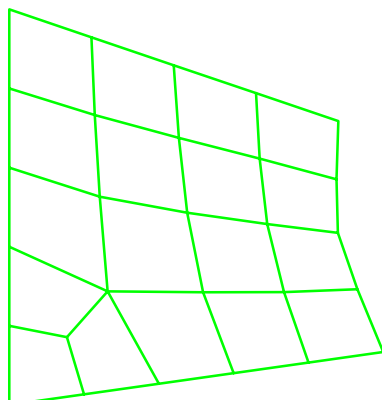
OK



Pick Only



All



What: Select all part edges that are less than 30mm.

How:



Deselect All



Filter...



Select All (Non-Pickable)



Edge... (scroll down)



Attributes...



Max length: 30



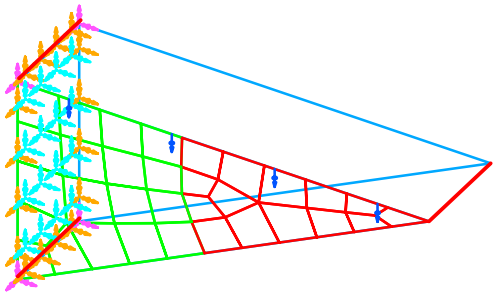
OK



Pick Only



All



Things to notice Only the short edges are selected.



If you have problems meshing imported geometry, use this technique to search for very small edges.

What: Create a solution set to run verification checks and estimate solution time and file sizes.

How:



Model Solution



Create...



Options...



Method: Verification Checks and Estimates



OK or Dismiss (all forms)

What: Keep the hypermatrix file and start the verification solve.



Hypermatrix File Management: Keep



Solve

Why: You normally will want to keep the hypermatrix file only for a restart analysis, such as to change loads but not the model. Keeping the hypermatrix file for a verification run is only being done here to illustrate how this file is managed.

What: Report the status information.



Things to notice

The time estimate is very small. The estimated size for the hypermatrix file is 5 megabytes.

What: Examine your file directory, looking for all the files that begin with the name of your model file.

Things to notice

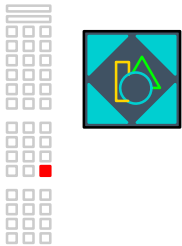
The two files with the extensions .mf1 and .mf2 are your model file. The file with the extension .mfh is the hypermatrix file. This file contains stiffness information, used as a scratch file by the solver.




To solve a large model with limited disk space, you can split this file into two equal size files on different disks.


What: Delete any model files from earlier workshops that you don't need.


How:




Items form

 (select a model file name)

 *Actions*



Delete

 *OK*

Things to notice

The file is deleted from the data management system and from the operating system.

Why: Deleting files this way instead of with operating system commands will help prevent errors with the data management system in the software.

What: Delete the hypermatrix file from the operating system.

Hint

Unix:
rm filename.mfh

NT:
(drag the file to Recycle Bin)

Warning!

This procedure should only be used for files not tracked by the data management system. Most files (such as model files and library parts) should be deleted using *Manage Items*, as shown on the previous page.

You have completed the Simulation Troubleshooting tutorial.

You can delete or put away any FE models and parts. They are not used in any other tutorials.

Hint



See also...

For additional information, see the following:

 *Help, Manuals, Table of Contents*

Simulation: Finite Element Modeling User's Guide

- Simulation Techniques and Examples

- Post Processing (picture file tips)

- Using Simulation Tools

- Overview of Finite Element Models

- Creating an FE Model on a Part

- Managing Models in Simulation

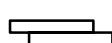
- Selecting Simulation Entities

- Displaying and Deleting Simulation Entities

Simulation: Model Solution/Optimization User's Guide

- Solving the Model

Additional information and tips can be found on the SDRC Web:

 *Help, Web, SDRC*

- Customer Support

- Frequently Asked Questions

- Tech Info

- User Groups

See also...

For information on topics not covered in the tutorials, see the following:

 *Help, Manuals, Table of Contents*

Simulation: Finite Element Modeling User's Guide

Simulation Techniques and Examples

Using the Simulation Advisor

Modeling Laminates

Simulation: Model Solution/Optimization User's Guide

Using the Solvers

Using Potential Flow Analysis

How the Solvers are Formulated

Simulation: Element Library

Simulation: External Solvers User's Guide

Simulation: Model Response Users' Guide

Simulation: Modeling for Moldflow User's Guide

Simulation: Thermal Analysis User's Guide

TMG Thermal Analysis,

ESC I-DEAS Electronic System Cooling

Simulation: Model Solution Open Solution User's Guide

What's next?

After completing the Fundamental Skills tutorials, use the Advanced Projects tutorials to get an introduction to other element types and solution methods.

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