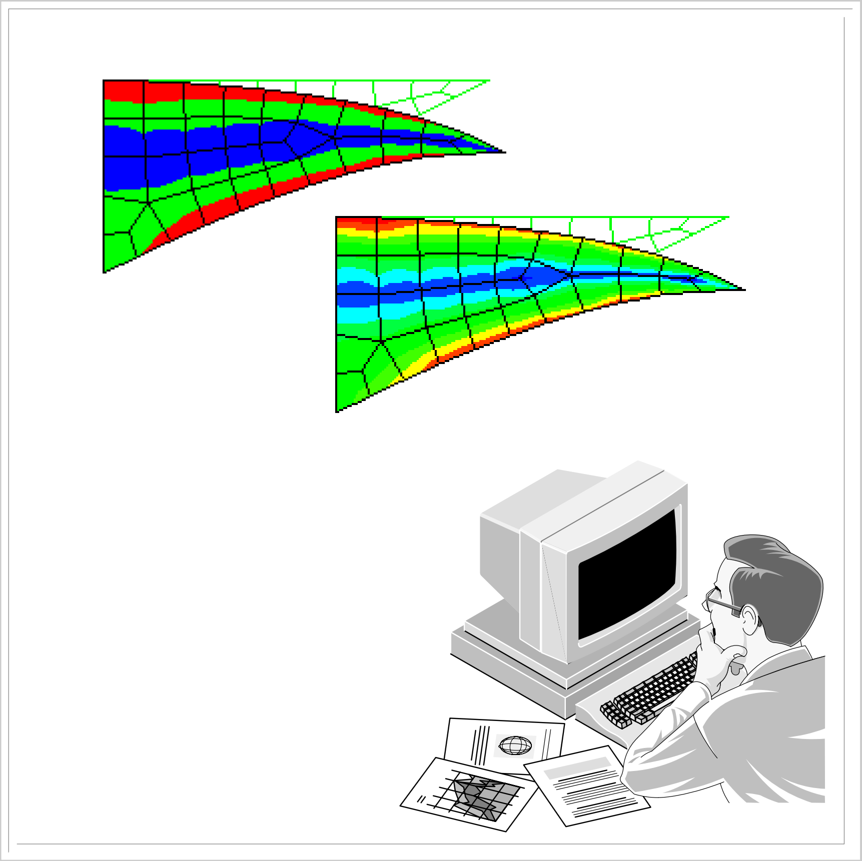


Simulation Troubleshooting

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.

File
Open

Open Model File form

Model File name: any unique name

OK

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

Simulation

Master Modeler

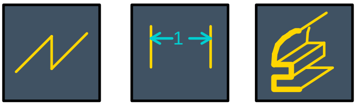
Set your units to mm.

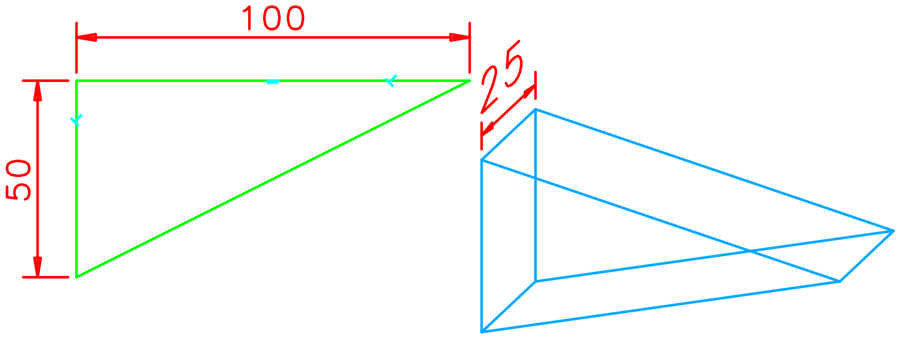
Options
Units

mm (*milli newton*)

What: Create a simple part as shown.

Hint





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

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.

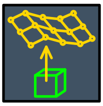
Why:
If you make a mistake at any time between saves and can’t recover, you can reopen your model file to the last save and start over from that point.

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

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.

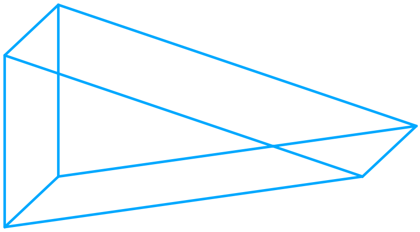
FE Model Create form

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.

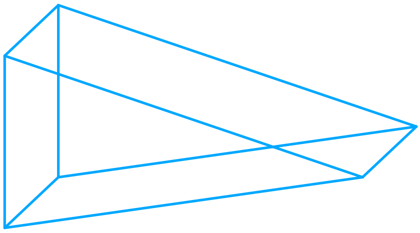
Hint

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

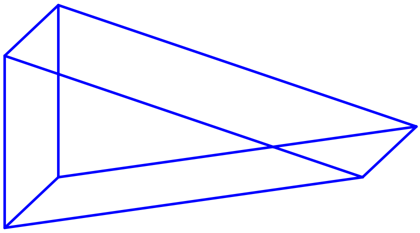
Bracket1... (double-click)

Things to notice

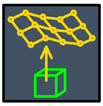
There are two parts listed. Bracket1 has an FE model (named Fem1) associated with it.

Dismiss

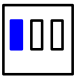
What: Create an FE model associated with Bracket2, the visible part.



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.

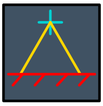
Deselect All

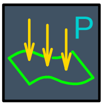
If you can’t select part geometry to mesh or apply boundary conditions, make sure your FE model is associated with the part you are trying to select.

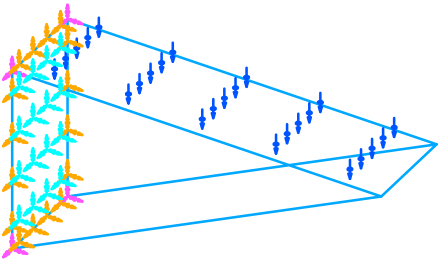
In the next few steps, you’ll create results to be used in the remainder of the tutorial.

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

Hint

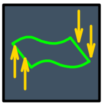






What: Create a boundary condition set.

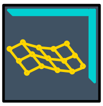
Hint

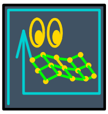


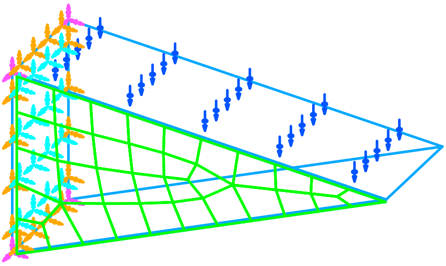
What: Mesh the front face.

Hint

 *Meshing*



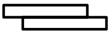




Things to notice

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

Recovery Point

 *File*
Save

What: Solve the model for linear statics.

Hint

 *Model Solution*

 *Create*...

 *Type of Solution*: *Linear Statics*

What: Check for errors.

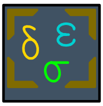
Hint

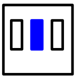
 **Check *I-DEAS List*.**

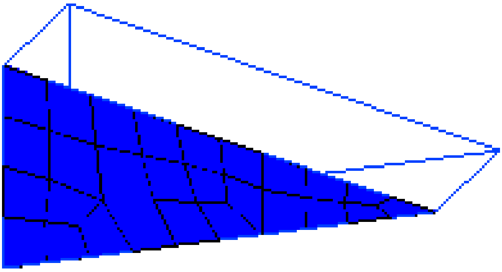
What: Display the stress and displacement results.

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.

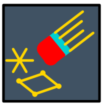
Recovery Point

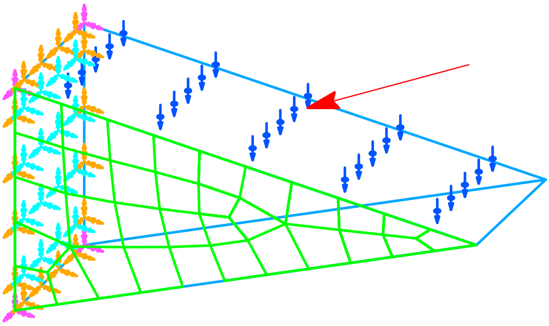
File
Save

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

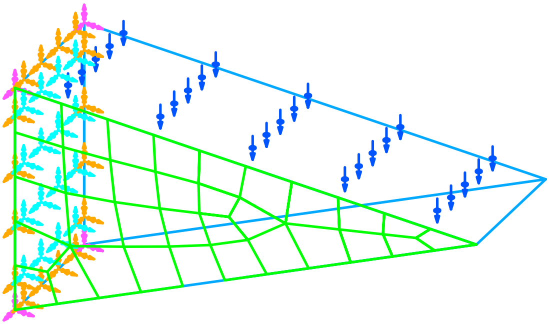
OK

Fem2

Get

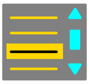
Dismiss

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.



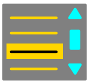
What: Verify that FEM2 is the active FE model on the workbench.

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

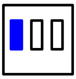
When FEM2 is selected, the Get button is grayed out, meaning the FE model is already on the workbench.

 *Dismiss*

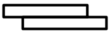
What: Delete the stored results.

Hint

 *Post Processing*

 *All*

Recovery Point

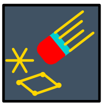
 *File*

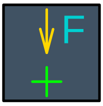
Save

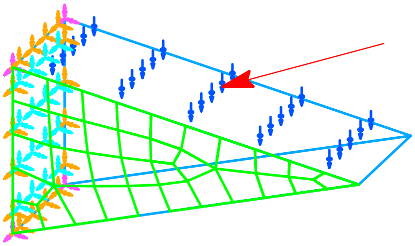
What: Delete the face pressure, and create an edge load.

Hint

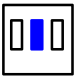
Boundary Conditions







1 pick edge

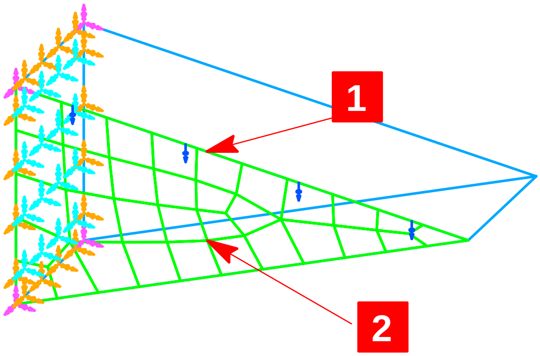


2 pick front face

Total force

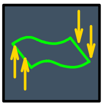
In Plane Force: 1000

OK



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

Hint



What: Solve the model.

Hint

 *Model Solution*

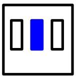
Remember

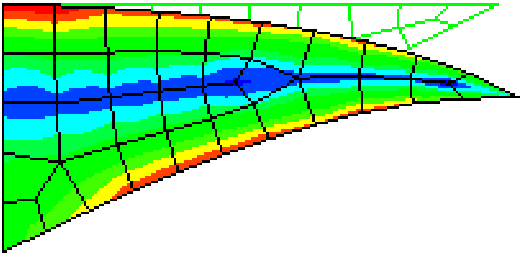
A solution set already exists, since the model was copied.

What: Display the results.

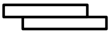
Hint

 *Post Processing*





Recovery Point

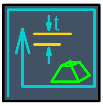
 *File*

Save

What: Try to modify physical properties.

Hint

Meshing



Directory

Thin Shell

List Owning Model Entities

OK

Yes

Things to notice

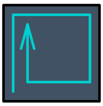
The physical property table can’t be modified. It’s used in models that contain stored results, which lock the models.

Continue

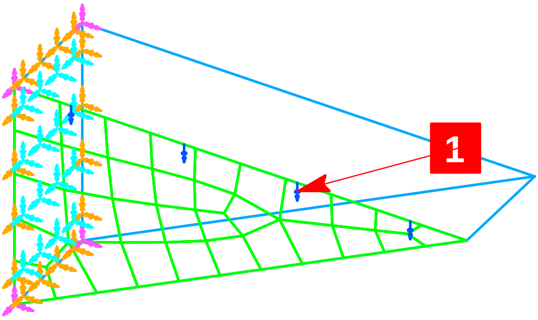
What: Try to modify the load.

Hint

Boundary Conditions



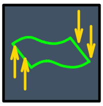
1 pick load



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.

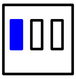
OK

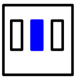
Cancel

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.

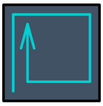
There is a hierarchy of results that prevents you from making changes:

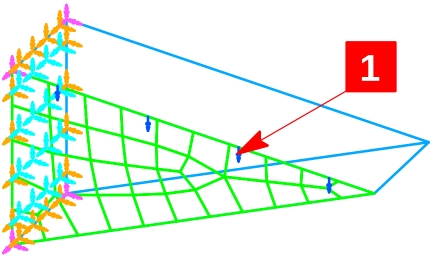
- Result sets
  - Solution sets
    - Boundary condition sets
      - Load sets
      - Restraint sets
  - Elements
    - Physical property tables
    - Material property tables

What: Try to modify the load.

Hint

Boundary Conditions





1 pick load

Force on Edge form

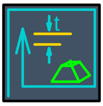
OK

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

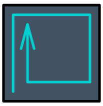
Things to notice

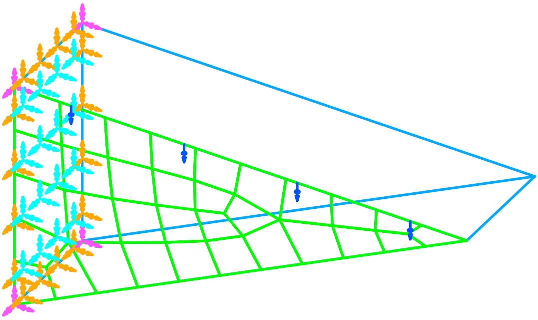
You’re still not allowed to modify the physical properties because they’re used in Fem1, which still contains results.

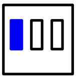
What: Try to modify the part dimensions.

Hint

 *Master Modeler*



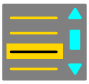


 pick part

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)

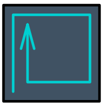
Things to notice

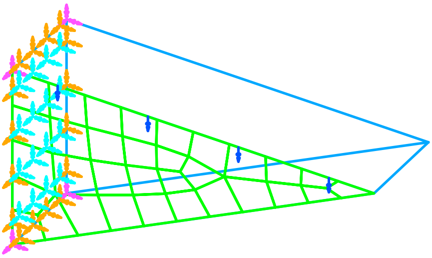
Fem1 has been deleted.

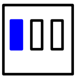
 *Dismiss*

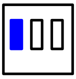
What: Modify the part length dimension.

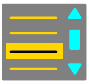
Hint





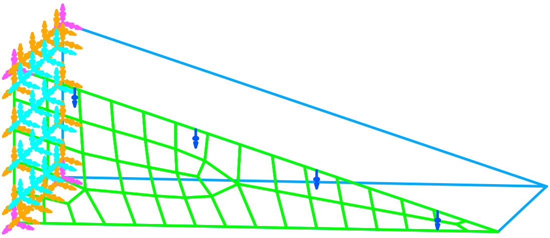
 pick part

 *Dimension Values*

 100

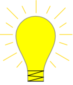
 150

 *OK*



Things to notice

Both the part and mesh are updated.

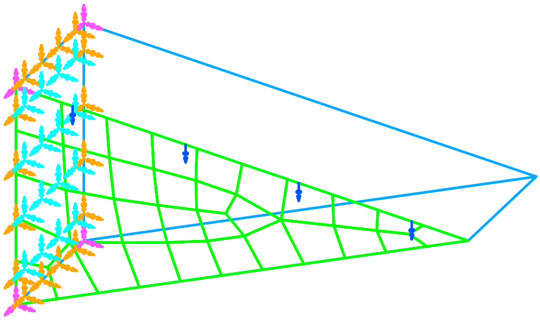


To speed up multiple part changes, put away the FE model before modifying the part. This avoids the time of re-meshing with each change. The mesh will be updated when you get the FE model onto the workbench.

What: Open your model file to the last save.

Hint

Control–Z



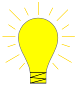
What: Slightly move an edge of the Graphics Window.

Things to notice

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

What: Re-display the graphics.

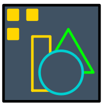
Hint



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...

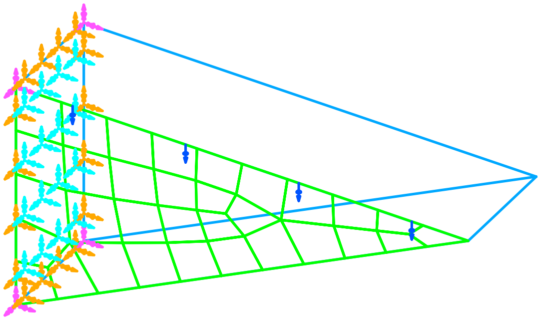
Node (toggle off)

OK (all forms)

What: Try to list the coordinates of a node.

Hint

Boundary Conditions



Things to notice

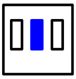
Nodes aren’t pickable because they aren’t displayed.

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

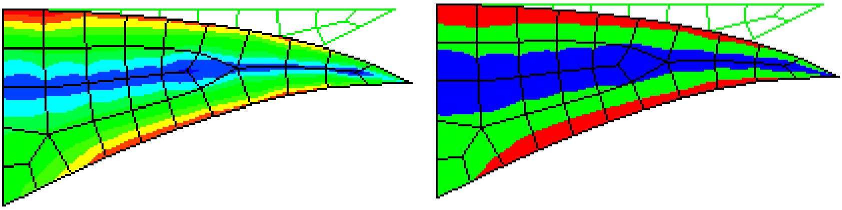
What: Display the stress results.

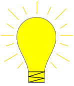
Hint

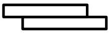
 *Post Processing*

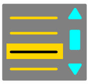


Your display will look like either:



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

 *Options*
Preferences

 *Display*

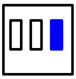
 *Double buffering*

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.

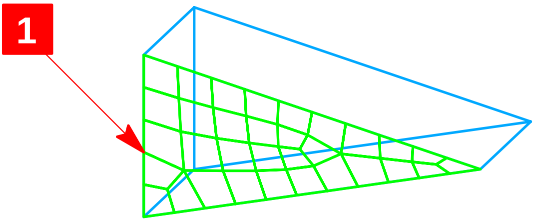
How to use “Related to” options

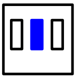
What: Display the stresses of all elements related to the left edge.

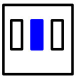
How:

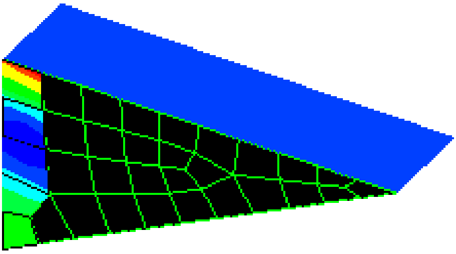
 *Related To*

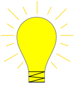
1











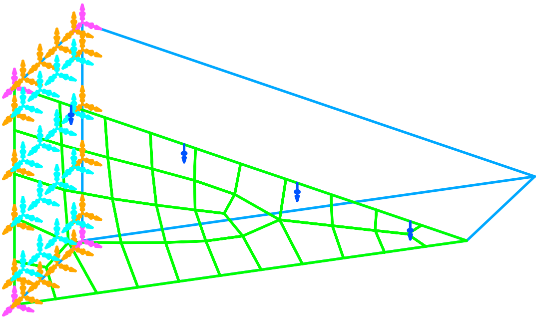
You can use this function to pick all elements or nodes on a surface to apply boundary conditions.

How to use the selection filter

What: Set the selection filter to pick only elements.

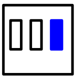
Hint

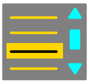
 *Meshing*



Things to notice

As you move the cursor, all entities are selectable.

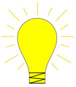
 *Filter...*

 *Element*

 *Pick Only*

Things to notice

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



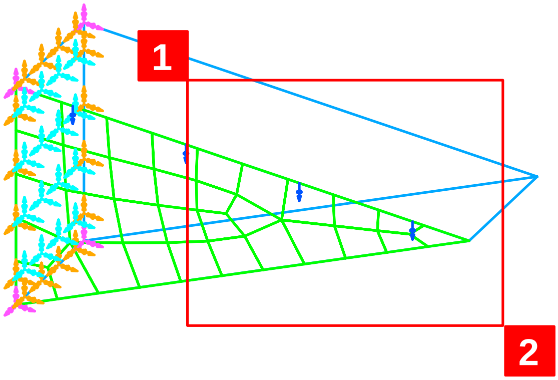
You can use the filter to limit what you want the software to select.

What: Preselect a group of elements in a rectangle.

How:

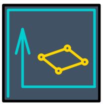
1

2 drag a box



What: Modify the color of the selected elements.

How:

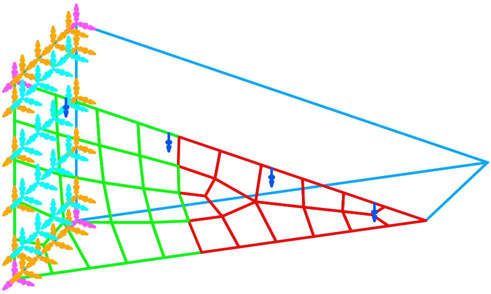


Color

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

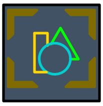
Green

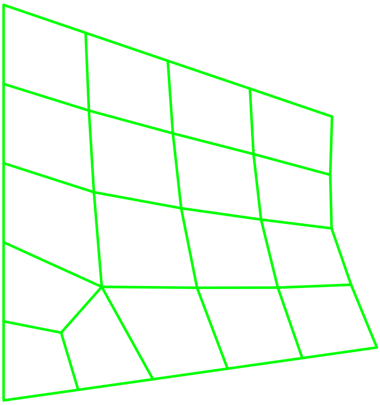
Things to notice There are many other attributes you can use to select elements, such as element type, material, and other values.

OK

Pick Only

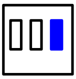
All

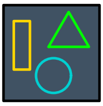


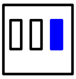


What: Select all part edges that are less than 30mm.

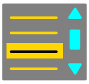
How:

 *Deselect All*



 *Filter...*

 *Select All* (*Non–Pickable*)

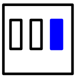
 *Edge...* (scroll down)

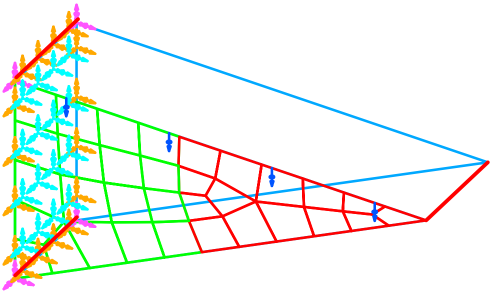
 *Attributes...*

 *Max length*: 30

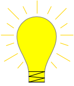
 *OK*

 *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.

Run verification checks

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.

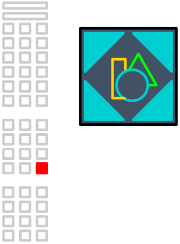
Things to notice

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.

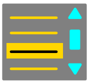
To solve a large model with limited disk space, you can split this file into two equal size files on different disks.

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.

Tutorial wrap-up

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

Structural Dynamics Research Corporation
2000 Eastman Drive
Milford, Ohio 45150
(513) 576−2400