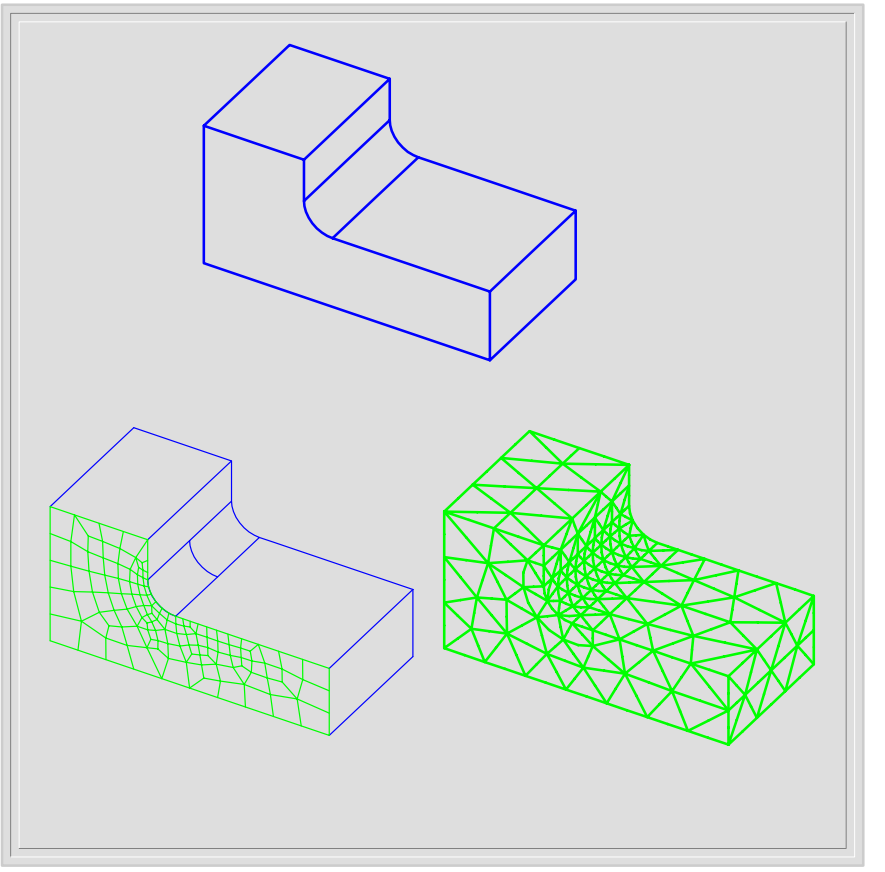


Free Meshing

I-DEAS™ Tutorials: Fundamental Skills

This tutorial covers free meshing techniques on surfaces and volumes.

Learn how to:

- mesh a surface
- refine the mesh on curved edges
- assign local element densities
- use maximum area plane meshing
- mesh a solid

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

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

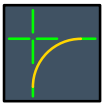
Set your units to mm.

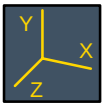
Options
Units

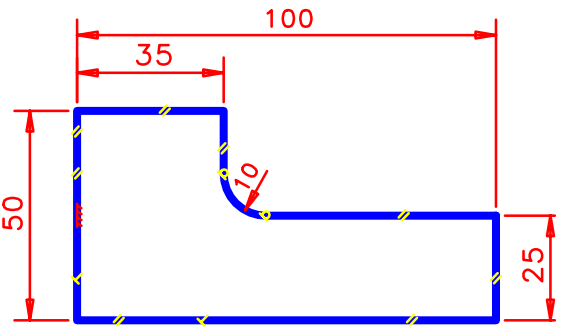
mm (*milli newton*)

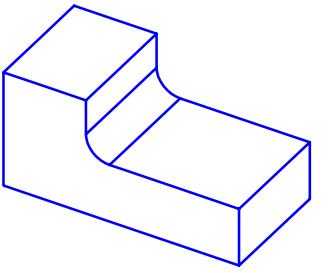
What: Sketch and extrude to 50mm the part shown.

Hint









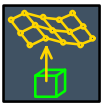
What: Name the part.

Hint

What: Create an FE model associated with the part.

How:

 *Meshing*



FE Model Create form

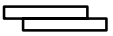
 *OK*

Things to notice

There's no obvious change to your part. But if you look at the Manage form (double-click the part name to expand it if there's an ellipsis after the part name), you'll see that the part now has an associated FE model.

Hint

Save your model file.



File
Save

Warning!
If you're prompted by I-DEAS to save your model file, respond:

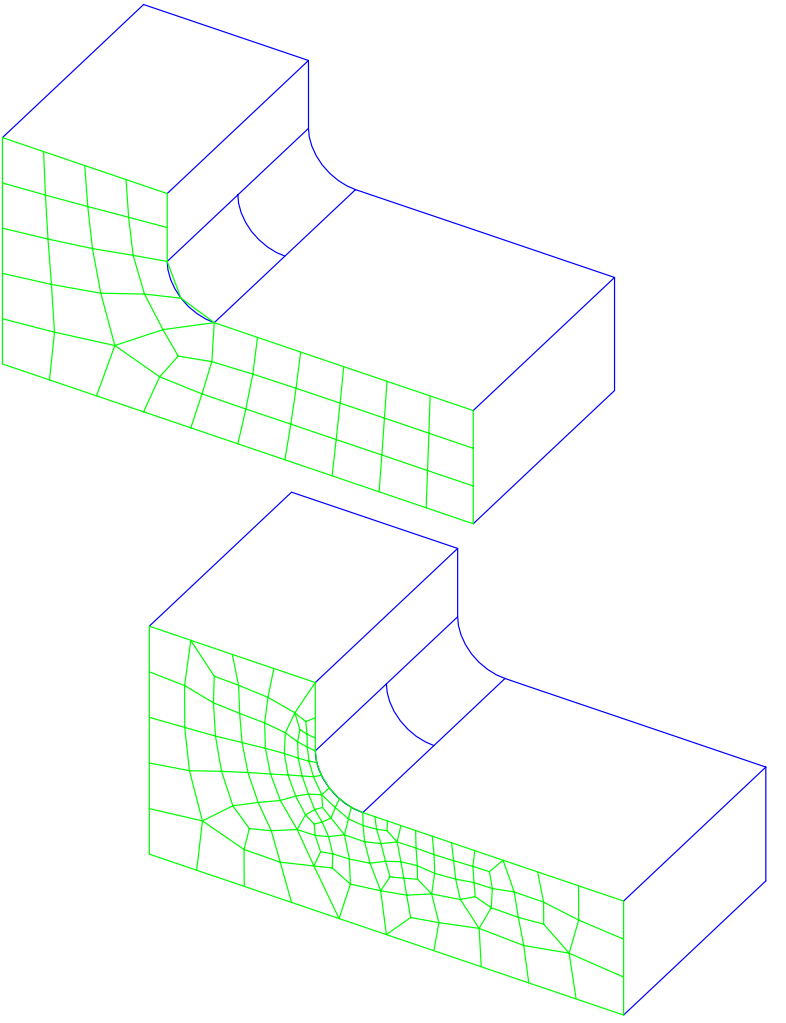
No

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.

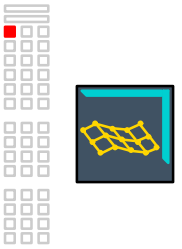
In this section, you'll create thin-shell elements on a surface of a part using mostly default parameters. In later sections, you'll use other parameters to control the sizes and locations of elements in the mesh.

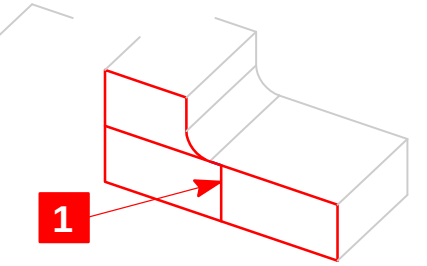


Nodes and elements meshed on part geometry will associatively change if you modify the part's dimensions.

What: Define mesh parameters on the front surface, using 10mm thin-shell quadrilateral elements.

How:





1 pick surface

Define Mesh form

 *Element Length*: 10

 *Free Options...*

Define Free Meshing Options form

Curvature Based Length:

 *None*

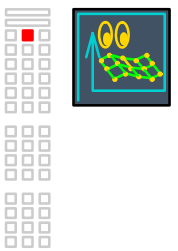
 *OK*

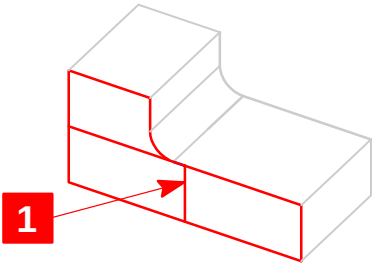
 *OK*

Why: An element size of 10mm creates at least two elements along the short side, which is 25mm. Curvature-based refinement was turned off so you can investigate the effect of this parameter in the next step.

What: Preview the distribution of nodes around the boundary of the surface.

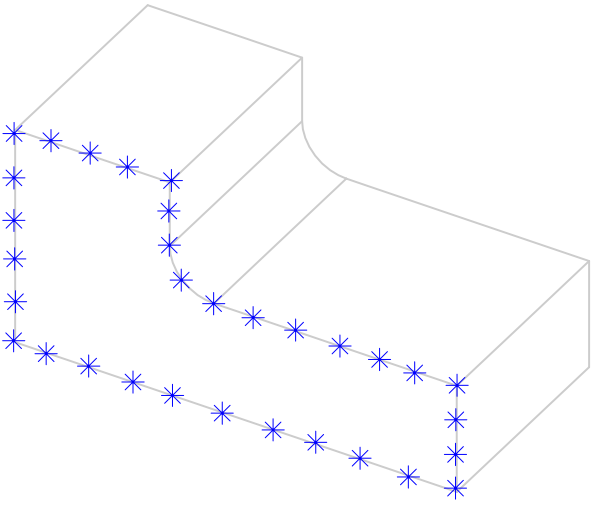
How:





1

Modify Mesh Preview form



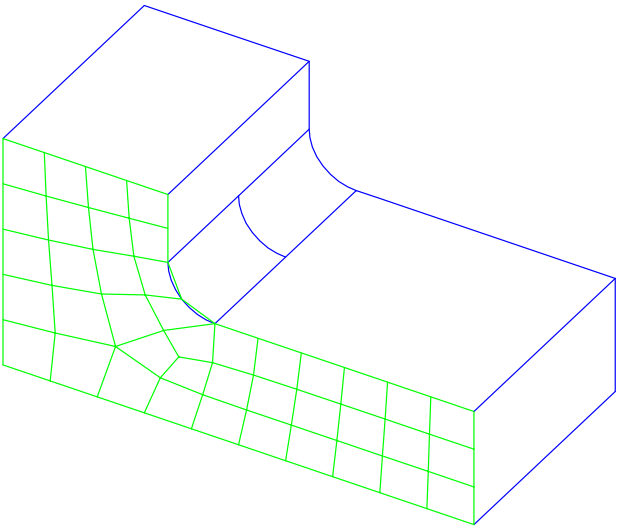
Why: Checking the nodes on the boundary gives a good preview before meshing.

The preview function is also available on the mesh definition forms.

What: Generate the mesh on the surface.

How:

Modify Mesh Preview form



Things to notice

Although the mesh is quite uniform, it won't necessarily give the best results, especially in the area near the fillet.

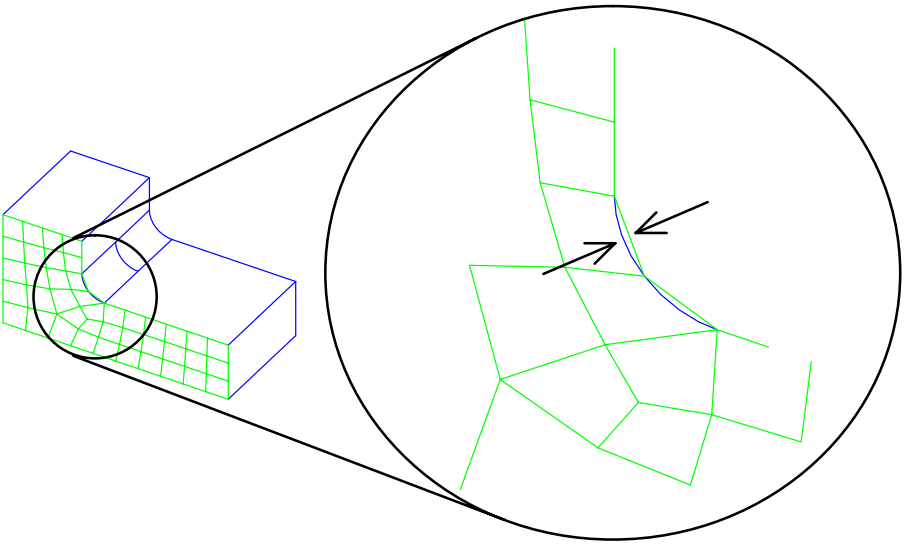
 *Cancel Mesh*

Refine the mesh on curved edges 1 of 3

Elements should generally be smaller at curved edges so that:

- the straight element edges more closely approximate the curved geometry, and
- stresses, which tend to be higher at internal corners, are more accurately modeled.

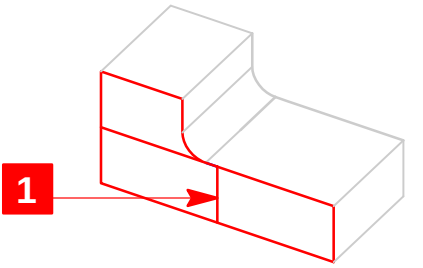
Curvature-based length refines the mesh density at all curved edges by controlling the deviation between the element edges and the curved part geometry. You may enter the maximum deviation either as a fixed absolute value or as a relative percentage of the curve radius.



Refine the mesh on curved edges 2 of 3

What: What value of percent deviation gives you about 4 elements around the fillet?

How:



1

Define Mesh form

Free Options...

Define Free Meshing Options form

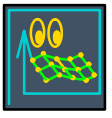
Curvature Based Length:

Percent Deviation

Percent Deviation: try values between 1 and 10

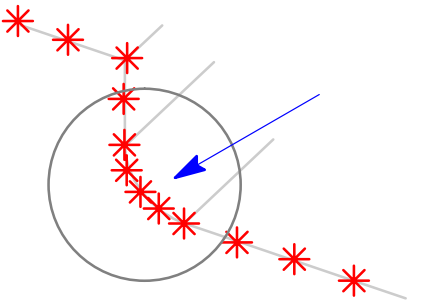
OK

Define Mesh form



Continued on next page...

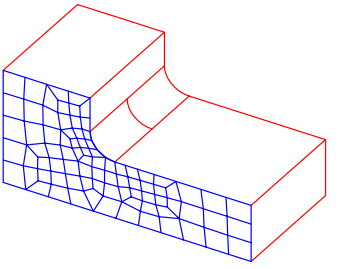
Modify Mesh Preview form



Things to notice

The node distribution has changed on the curved edge.

Modify Mesh Preview form



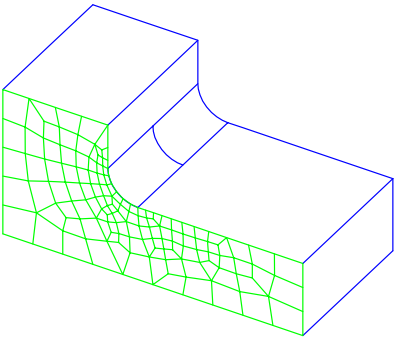
 *Cancel Mesh*

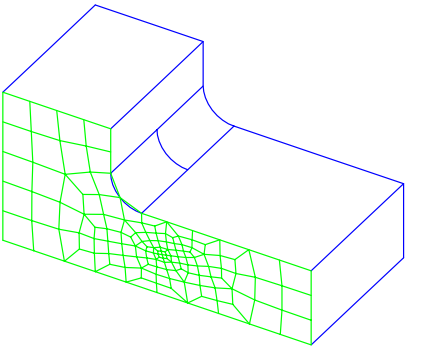
 *OK*

Assign local element densities 1 of 4

Curvature-based refinement automatically refines the mesh on all curved surfaces.

Defining local element densities gives you more control to refine the mesh locally at points or edges where you want more elements. It also gives more direct control over the number of elements on a curve.

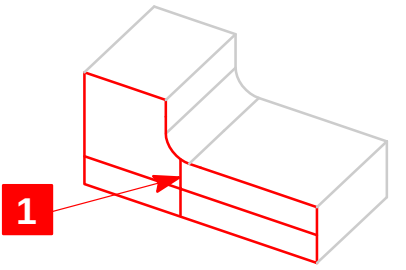




What: Assign a local element density on the curved edge to use exactly six elements.

Hint

1

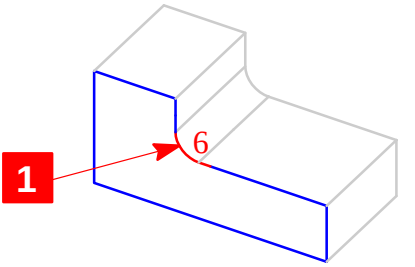


Free Options...

Curvature Based Length:

None

OK (all forms)



1 pick curve

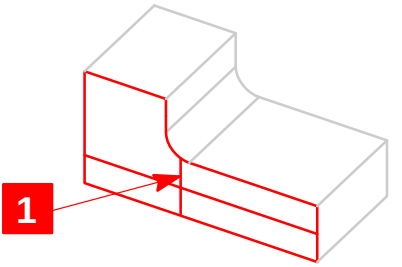
Check *I-DEAS Prompt*.

number of elements on edge: 6

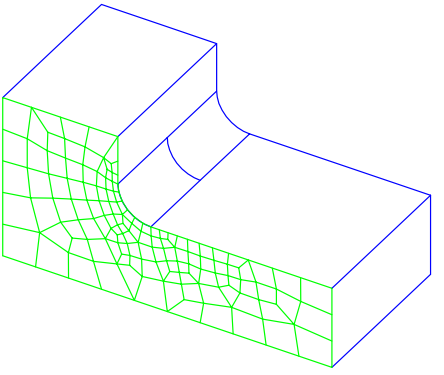
Assign local element densities 3 of 4

What: Generate the mesh.

Hint



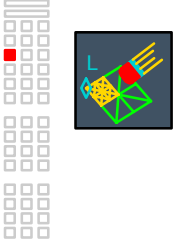
1



No

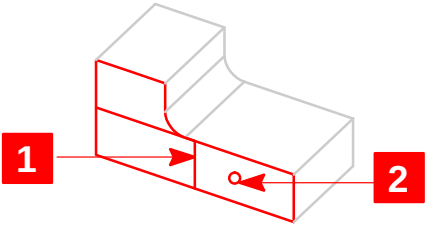
What: Delete the local element size setting on the curve.

Hint



What: Assign a local element density of 2 at a point on the surface.

Hint



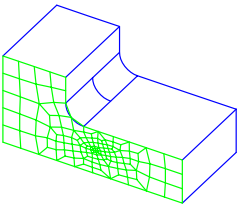
1

2

Check *I-DEAS Prompt*.

local element length: 2

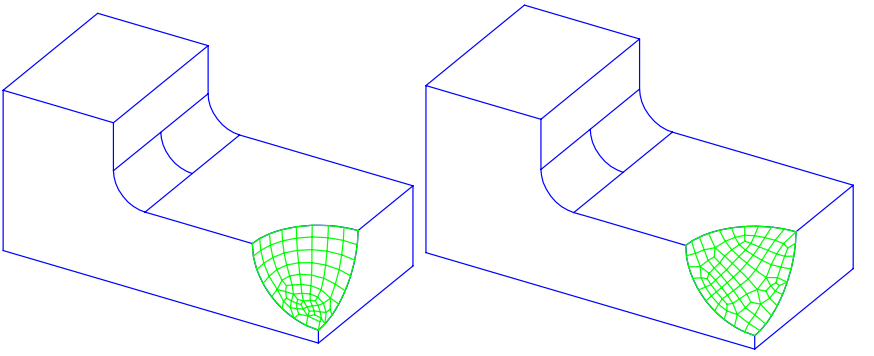
pick front surface



No

Use maximum area plane meshing 1 of 5

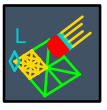
By default, the program uses the s and t parametric coordinates of the surfaces to calculate nodes for the mesh. In surfaces of revolution where the s and t coordinates degenerate to a point at the pole, like the north and south poles of the earth, this method may result in poor quality meshes, as shown on the left.



The maximum area plane meshing method projects the surface to a flat plane to calculate the mesh, and is not sensitive to poles. This method may require the surfaces to be partitioned so that they don't wrap more than 180 degrees.

What: Delete the local element definition.

Hint

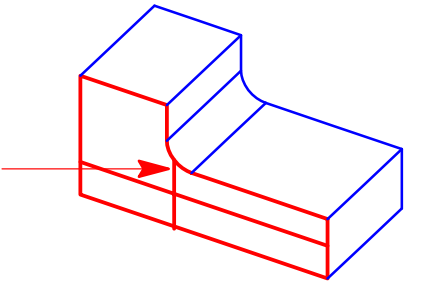


Use maximum area plane meshing 2 of 5

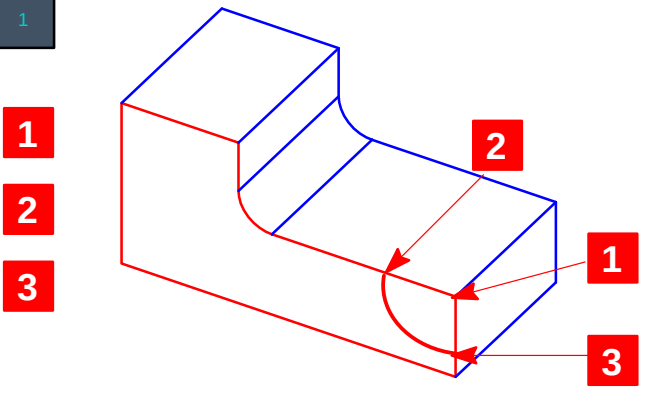
What: Create a revolved cutout.

How:

Master Modeler



pick surface

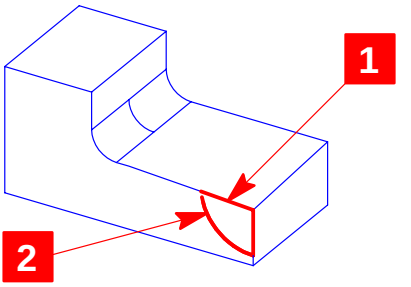


Use maximum area plane meshing 3 of 5

Section Options...

Stop at Intersections

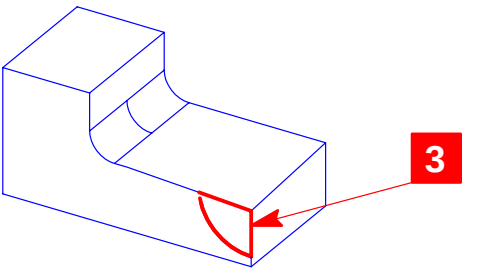
OK



1

2

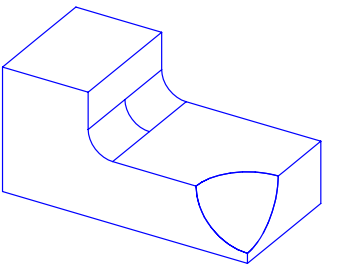
3



Cutout

Angle: 360

OK



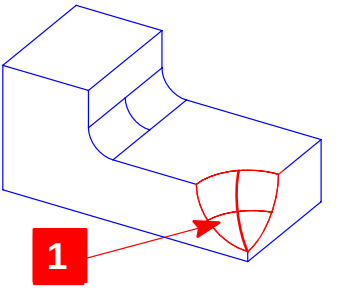
Use maximum area plane meshing 4 of 5

What: Using free meshing (with the default parameter space meshing), try to mesh the revolved surface.

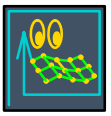
Hint

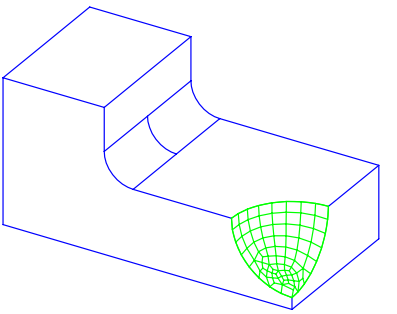
Meshing

1



Element Length: 3





Cancel Mesh

OK

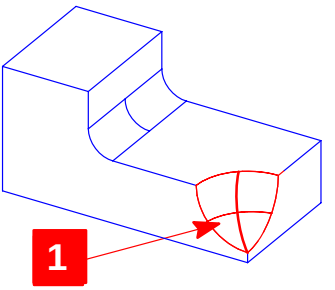
Things to notice

You may get a mesh that looks different, or you may get an error message that the meshing has failed. This can be a result of using parameter space meshing on a surface with a pole of revolution.

Use maximum area plane meshing 5 of 5

What: Mesh the surface with the maximum area plane option.

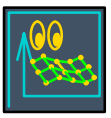
Hint

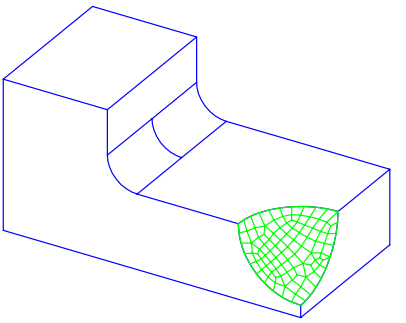


 *Free Options...*
Free Meshing Method:

 *Maximum Area Plane*

 *OK*





 *Cancel Mesh*

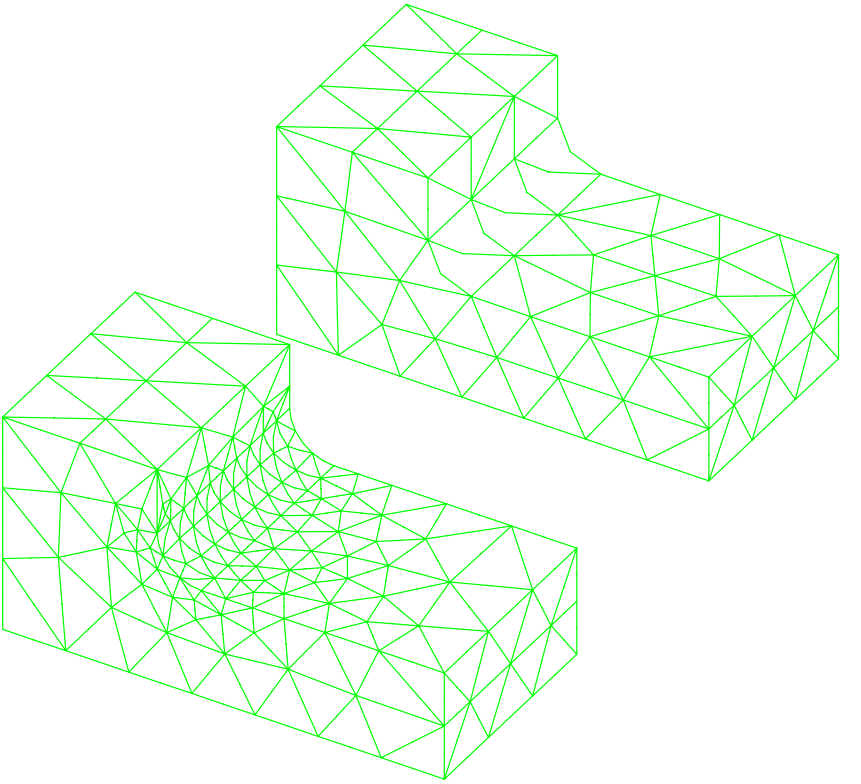
 *OK*

The third option *Automatic*, on the Define Free Meshing Options form, will automatically choose the most appropriate method.

The steps to generate a mesh of solid elements on a volume are the same as the steps to generate shell elements on a surface, except that you select part volumes instead of surfaces.

The global element size on the volume can be refined in local areas by the value of curvature-based refinement and by setting local element densities.

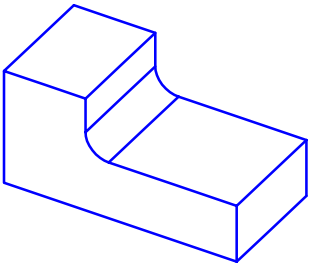
In this example, you'll first generate a solid mesh on the part with the default parameters. Then you'll generate the mesh using local settings to refine the mesh at the radius.



What: Get your original part, without the revolved cutout.

Hint

Control–Z



What: Set curvature-based lengths on all surfaces to *None*.

Why: Although you will generate a solid mesh in the next step, some parameters are entered on surfaces.

Hint

 *All*

 *Done*

 *Free Options...*

Curvature Based Length:

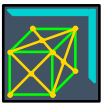
 *None*

 *OK*

 *OK*

What: Define a solid mesh to generate a solid tetrahedral mesh on the part volume with an element length of 15mm.

How:

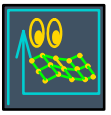


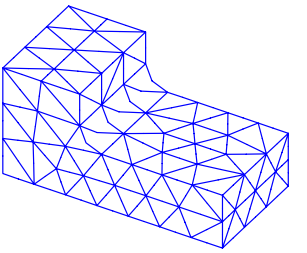
 *All*

 *Done*

Define Mesh form

 *Element Length*: 15





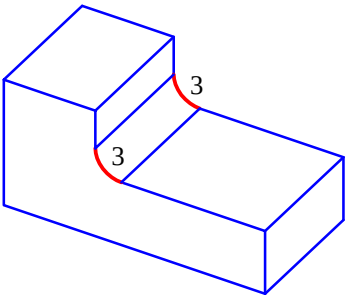
(The illustration here uses a hidden line removed display for clarity.)

 *Cancel Mesh*

 *OK*

What: Define a local element density of three elements per curve on the two curved edges.

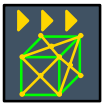
Hint
Use F2 to zoom in to pick the curves. Use shift-pick to select the second curve.



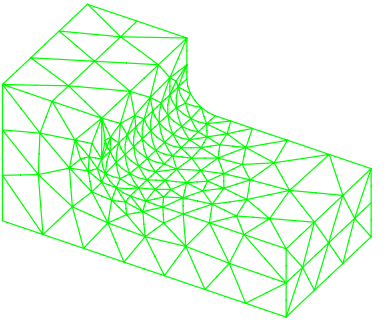
Check *I-DEAS Prompt*.

☐ *number of elements on edge*: 3

☐ *number of elements on edge*: 3



All



No

You have completed the Free Meshing tutorial.

Delete the FE model, then the part. This part is not used in any other tutorial.

Hint

See also...

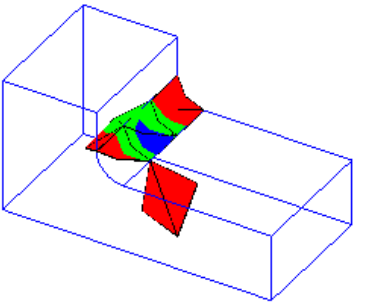
For additional information on the concepts covered in this tutorial, see the following:

Help, Manuals, Table of Contents

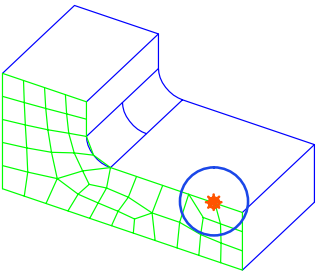
Simulation: Finite Element Modeling User's Guide
- Meshing a Model
  - Icon Overview for Meshing
  - Creating a Mesh
  - Modifying a Mesh

What's next?

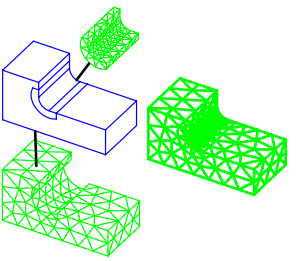
The tutorial on checking element quality will show how to compute element quality checks. This tutorial is important for advanced users of Simulation performing complex analyses.



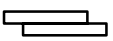
The tutorials on boundary conditions cover related information on ways to control node locations.



The tutorial on preparing a part for analysis includes partitioning a part into multiple volumes to control meshing.



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

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