

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—

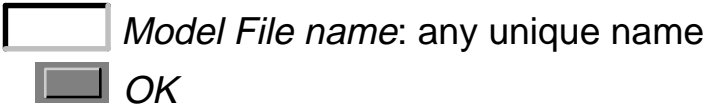
Creating Parts

- Introduction to Simulation
- Managing Parts in Model Files

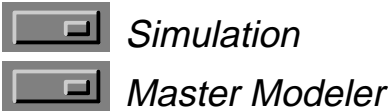
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



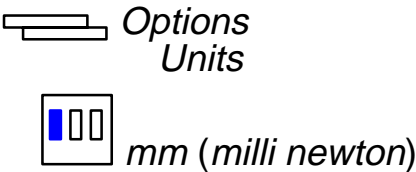
Open Model File form



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

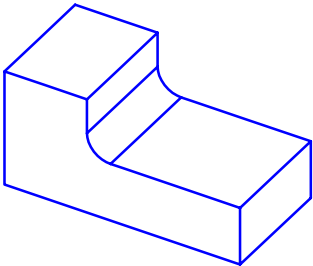
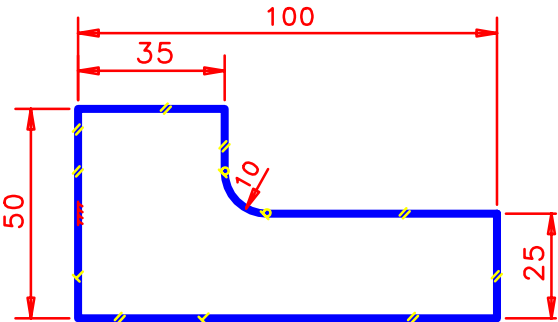
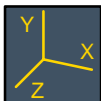


Set your units to mm.



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

Hint



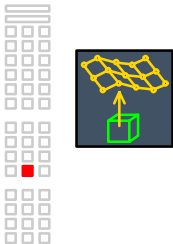
What: Name the part.

Hint



What: Create an FE model associated with the part.

How:



FE Model Create form



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.**Warning!**

If you're 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.

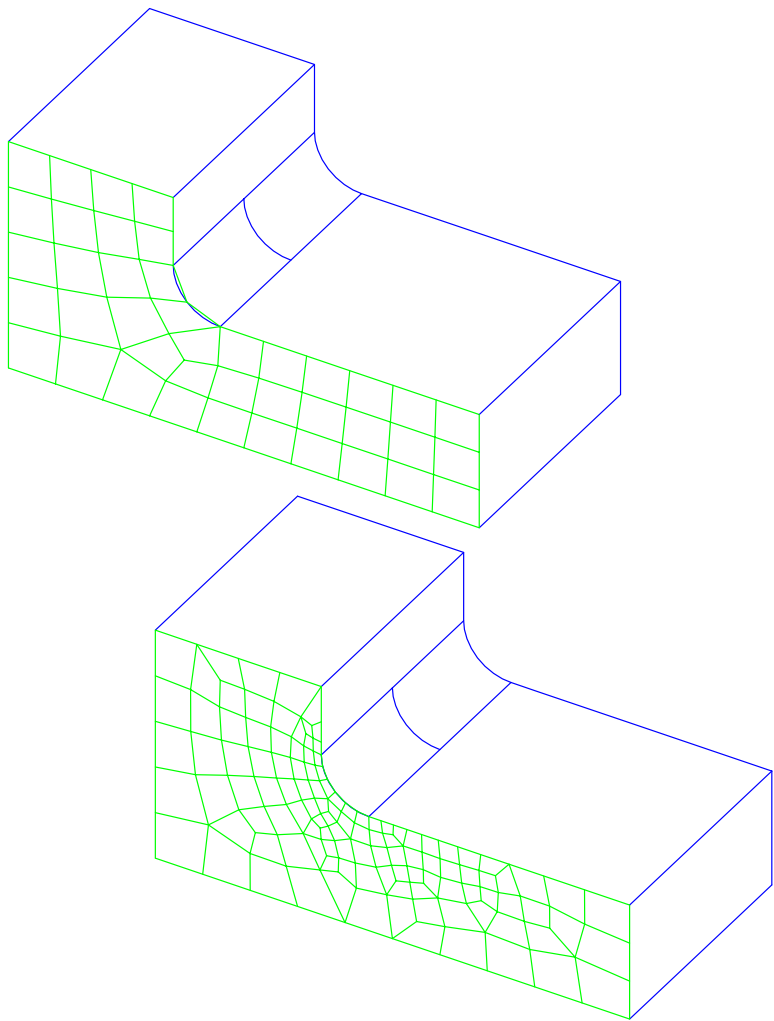
Why:

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.

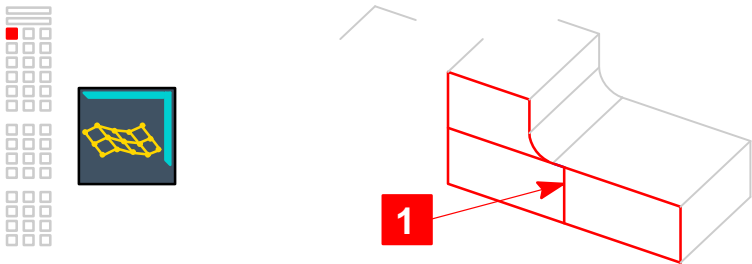
In this section, you'll create thin-shell elements on a surface of a part using mostly default parameters. In later sections, you'll use other parameters to control the sizes and locations of elements in the mesh.



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*

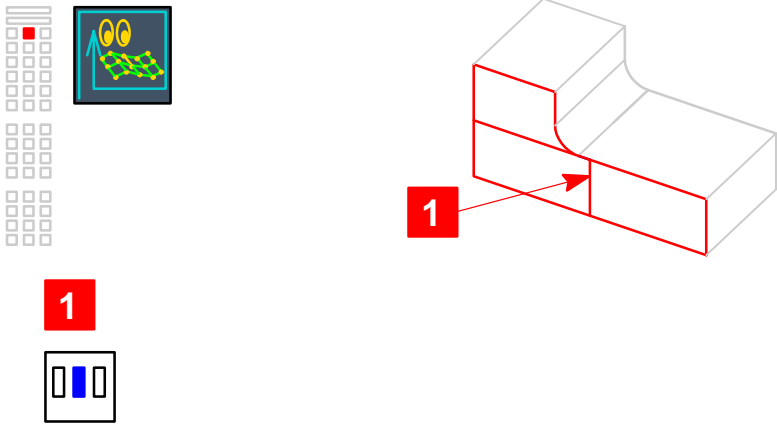
☐ *OK*

☐ *OK*

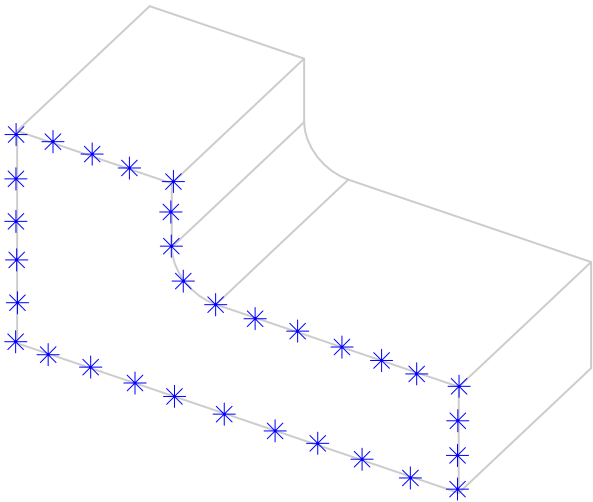
Why: An element size of 10mm creates at least two elements along the short side, which is 25mm. Curvature-based refinement was turned off so you can investigate the effect of this parameter in the next step.

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


How:



Modify Mesh Preview form



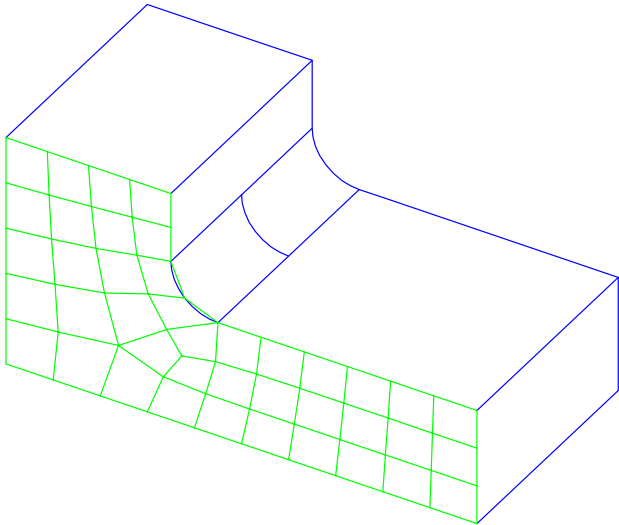
Why: Checking the nodes on the boundary gives a good preview before meshing.

 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.



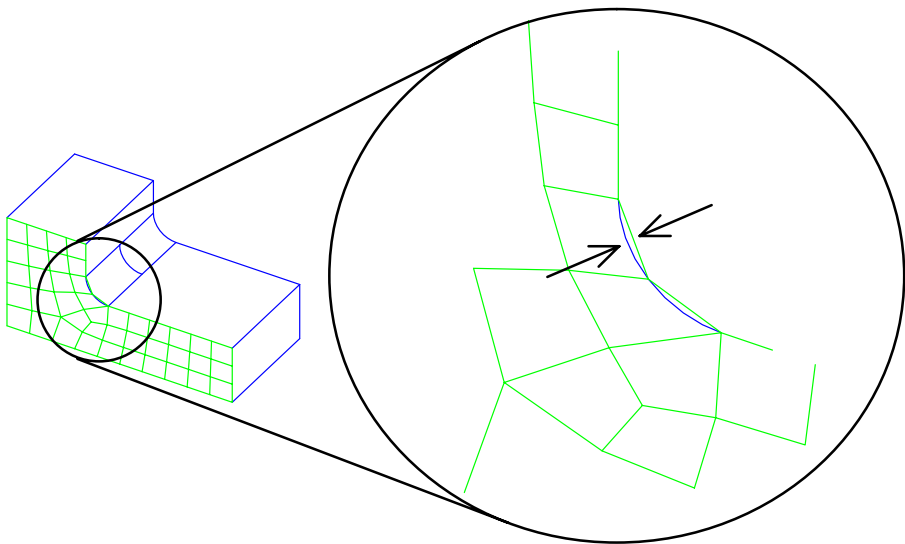
Cancel Mesh

Refine the mesh on curved edges 1 of 3

Elements should generally be smaller at curved edges so that:

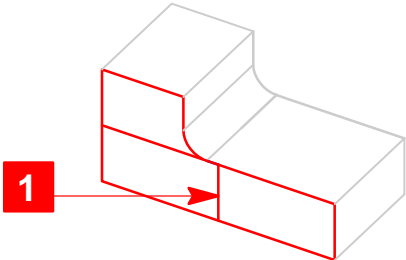
- the straight element edges more closely approximate the curved geometry, and
- stresses, which tend to be higher at internal corners, are more accurately modeled.

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.



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

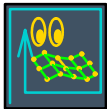


Percent Deviation: try values between 1 and 10



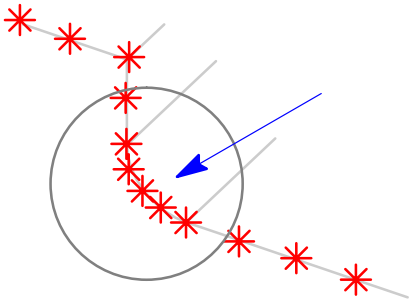
OK

Define Mesh form



Continued on next page...

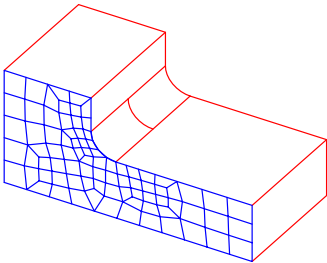
Modify Mesh Preview form



Things to notice

The node distribution has changed on the curved edge.

Modify Mesh Preview form



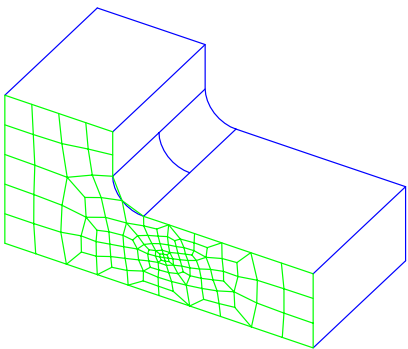
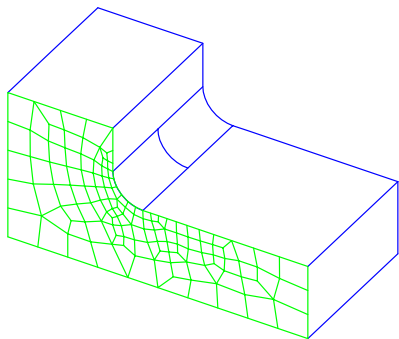
Cancel Mesh



OK

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.

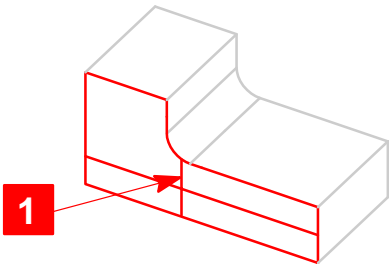


What: Assign a local element density on the curved edge to use exactly six elements.

Hint



1



Free Options...

Curvature Based Length:



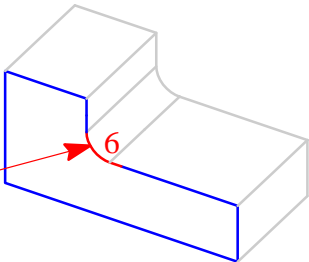
None



OK (all forms)



1



1

pick curve



Check I-DEAS Prompt.

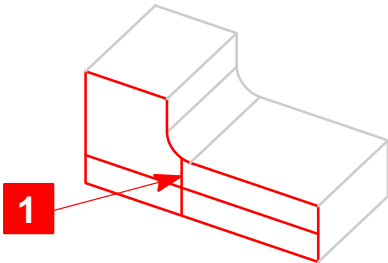


number of elements on edge: 6

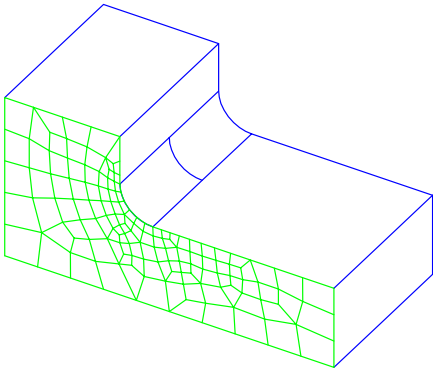


What: Generate the mesh.

Hint



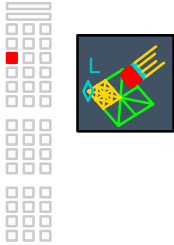
1



No

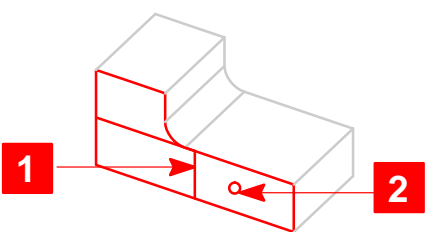
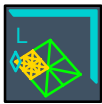
What: Delete the local element size setting on the curve.

Hint



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

Hint



1



2



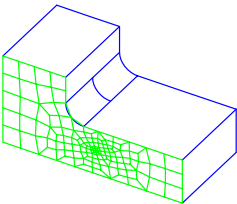
Check I-DEAS Prompt.



local element length: 2



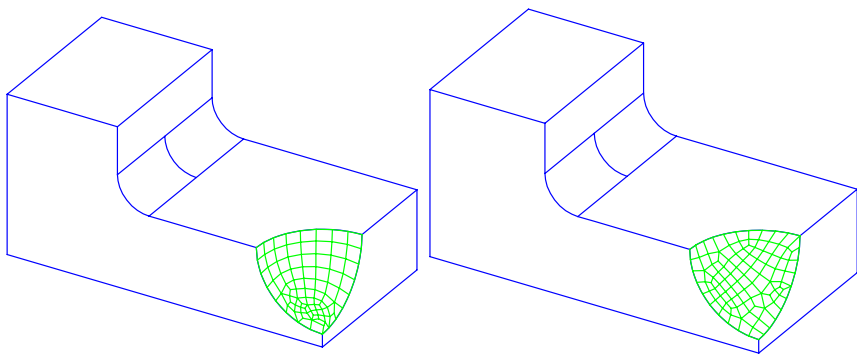
pick front surface



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



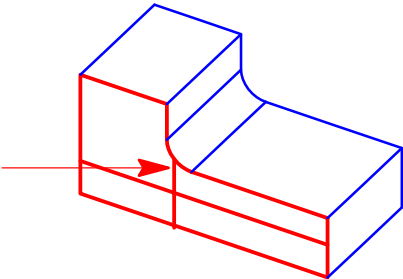
Use maximum area plane meshing 2 of 5

What: Create a revolved cutout.

How:



Master Modeler



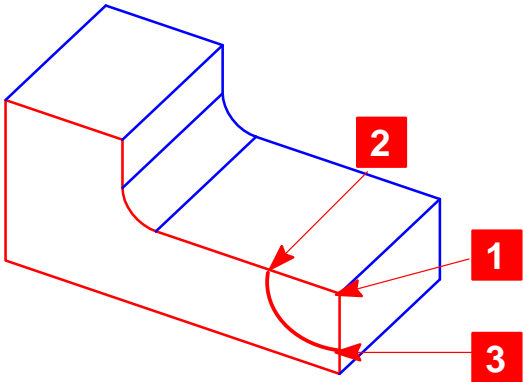
pick surface

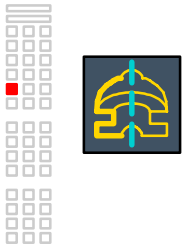


1

2

3





Section Options...



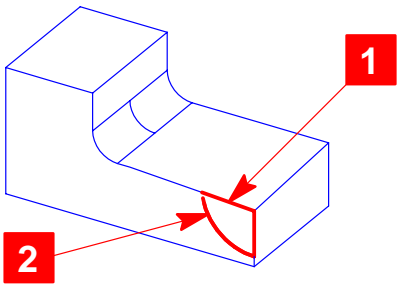
Stop at Intersections



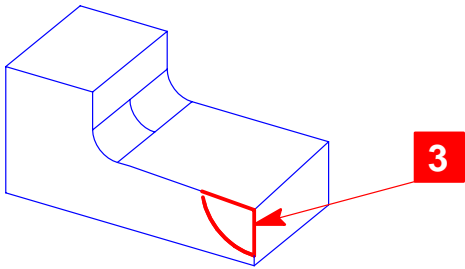
OK

1

2



3



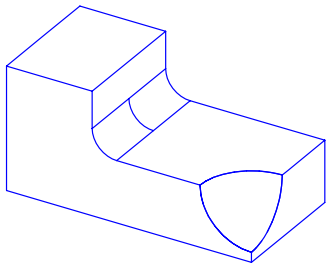
Cutout



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.

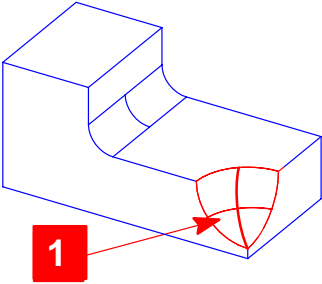
Hint



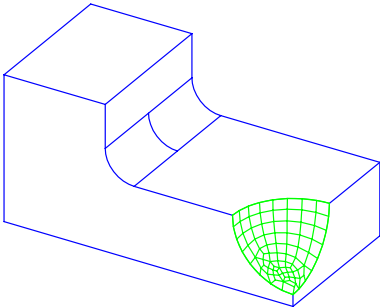
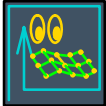
Meshing



1



Element Length: 3



Cancel Mesh



OK

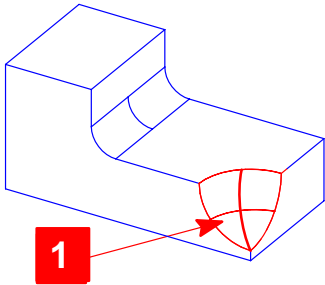
Things to notice

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



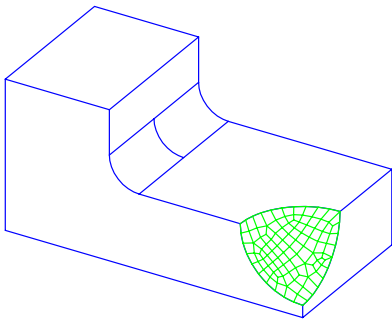
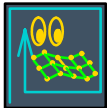
Free Options...
Free Meshing Method:



Maximum Area Plane



OK



Cancel Mesh



OK

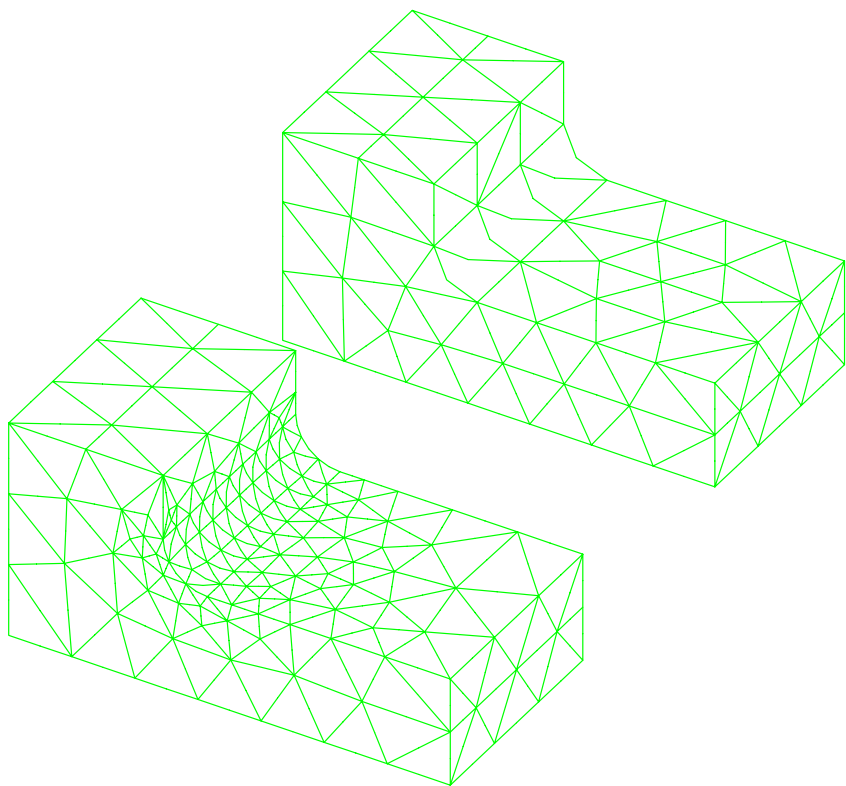


The third option *Automatic*, on the Define Free Meshing Options form, will automatically choose the most appropriate method.

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.

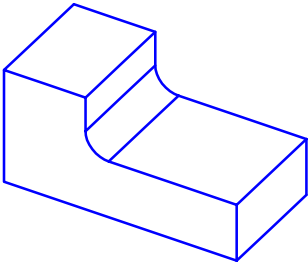
In this example, you'll first generate a solid mesh on the part with the default parameters. Then you'll generate the mesh using local settings to refine the mesh at the radius.



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



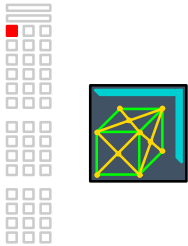
OK



OK

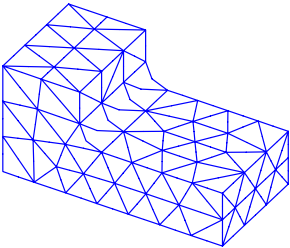
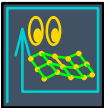
What: Define a solid mesh to generate a solid tetrahedral mesh on the part volume with an element length of 15mm.

How:



Define Mesh form

Element Length: 15

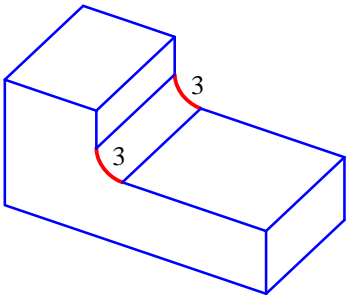
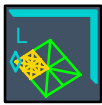


(The illustration here uses a hidden line removed display for clarity.)

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



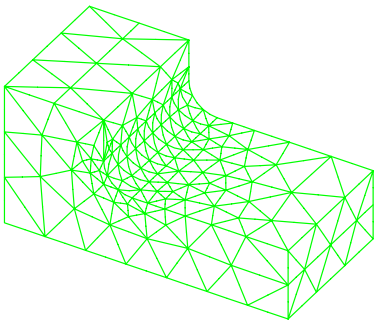
number of elements on edge: 3



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...

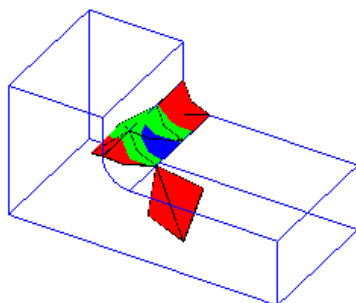
For additional information on the concepts covered in this tutorial, see the following:

 *Help, Manuals, Table of Contents*

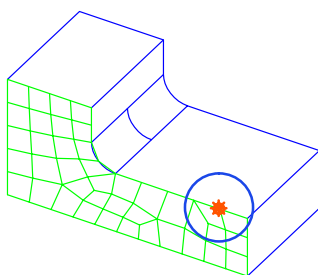
Simulation: Finite Element Modeling User's Guide
 Meshing a Model
 Icon Overview for Meshing
 Creating a Mesh
 Modifying a Mesh

What's next?

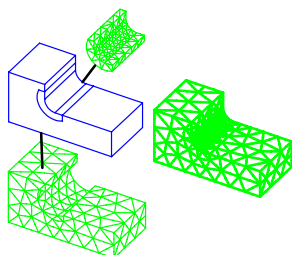
The tutorial on checking element quality will show how to compute element quality checks. This tutorial is important for advanced users of Simulation performing complex analyses.



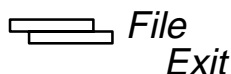
The tutorials on boundary conditions cover related information on ways to control node locations.



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



To exit this tutorial, select:



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