

Mapped Meshing on Surfaces

I-DEAS™ Tutorials: Fundamental Skills

This tutorial covers mapped meshing techniques on surfaces.

Learn how to:

- split surfaces
- mesh surfaces
- use biasing
- use unequal number of elements

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

—or—

Quick Tips to Using I-DEAS

—and—

Creating Parts

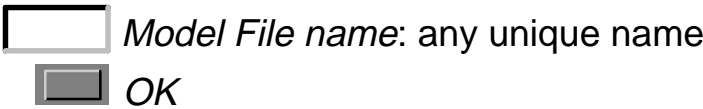
- Introduction to Simulation
- Managing Parts in Model Files
- Free Meshing

This tutorial on mapped meshing builds on the concepts introduced in the Free Meshing tutorial: defining mesh parameters and generating free meshes on surfaces.

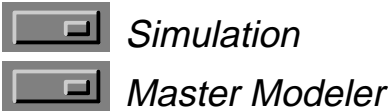
If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.



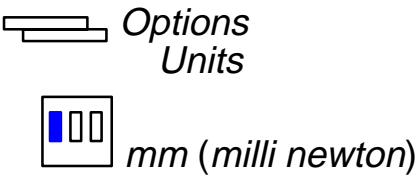
Open Model File form



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

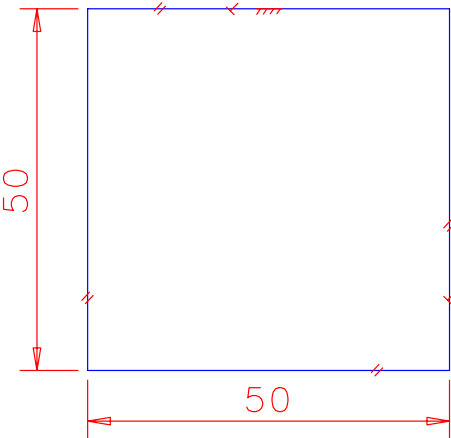
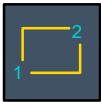


Set your units to mm.



What: Sketch this square to the dimensions shown.

Hint



What: Create a surface bounded by the square.

How:



Options...

Surface by Boundaries Options form



Stitch/Join Edges



Autochain Wireframe



OK



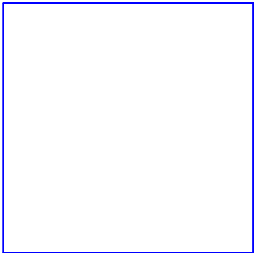
pick square



Done



Yes



What: Name this part.

Hint



Name: Mapped Surface

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:

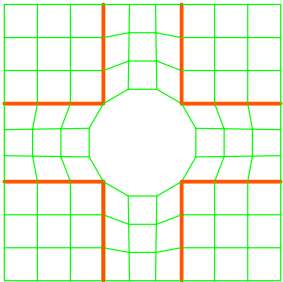
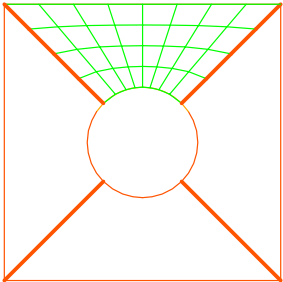
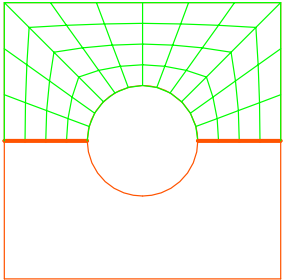
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

Hint

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

Unlike free meshing, mapped meshing doesn't allow holes in the interior of a mesh. Because of this, surfaces have to be split or trimmed to create 3- or 4-sided areas without interior holes.

Here are 3 different ways a square plate with a hole can be split, along with the resulting mesh patterns.

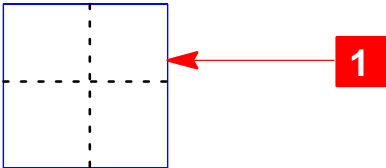


What: Sketch in place on the surface and draw a circle.

How:

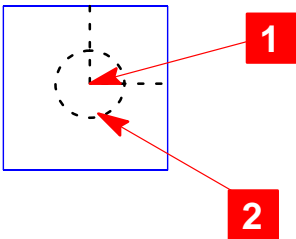


1 pick plane to sketch on



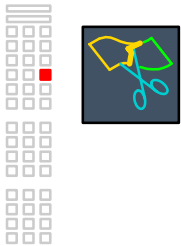
1 locate center

2 locate point on edge



What: Trim a hole in the surface. Keep the area outside the hole.

How:



- 1** pick surface
- 2** pick trimming curve

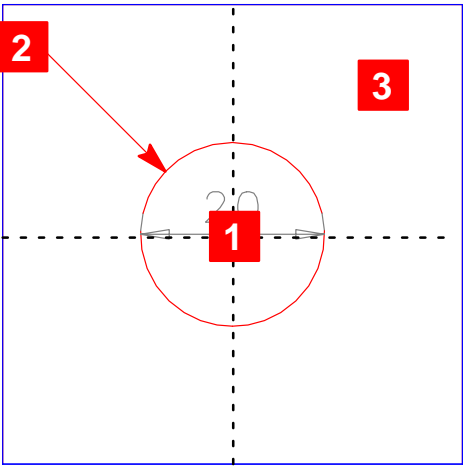


Done

- 3** indicate region of surface to keep



Done



Things to notice

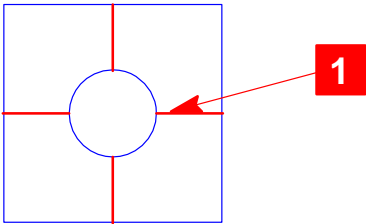
If you display the part in shaded mode, you'll see that there is now a hole in the center.

What: Sketch in place on the surface and sketch a line from the center of one side to the other.

How:



1

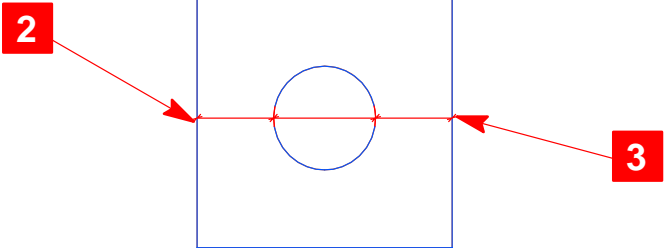


2 locate start

3 locate end



Done



Recovery Point





The circle and the line were sketched with *Sketch in place* on the surface – a good practice. You can associatively dimension and constrain trimming geometry to existing edges. This makes it easier to modify, and to use techniques like optimization, which uses part dimensions as variables to change the mesh.

What: Divide the surface with the line. In one step, you can extrude the line into a plane and split surfaces intersected by the plane.

How:



Section Options...



Stop at intersections (off)

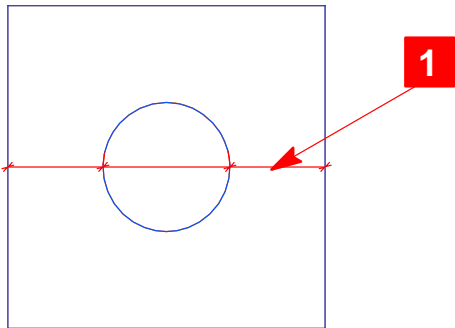


OK



Split Surface

1



Done

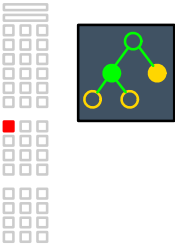
Extrude Section form



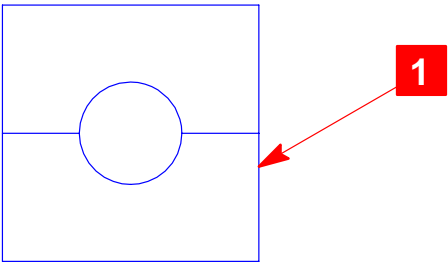
Recovery Point



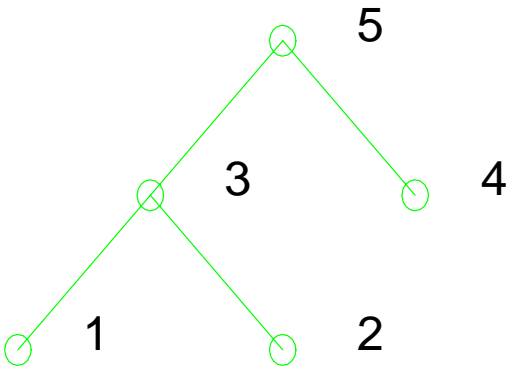
What: Display the part history tree to review the steps.



1



History Access form



- 1. Surface by boundary
- 2. Trim information
- 3. Trim operation
- 4. Extrude
- 5. Split operation



Dismiss

Things to notice

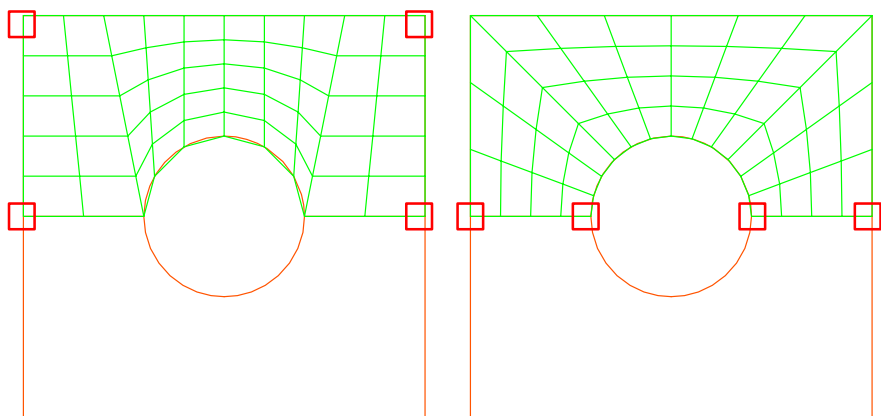
Since each step is a separate node of the history tree, each one can be modified or deleted.

With mapped meshing, element size and local element densities are not defined, as they are in free meshing.

The mapped mesh is controlled by:

- defining corners of the mesh (3 or 4)
- defining the number of elements per side
- optionally defining edge biasing – making the elements smaller at the ends or the center of an edge

The first decision is how to define the mesh corners.



Picking the physical corners may seem logical, but picking the corners as shown on the right above minimizes the distortion. This is especially true around the hole, where stresses may be important.

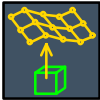
This tutorial follows the example mesh on the right.

What: Create an FE model to reference this part.

How:



Meshing



FE Model Create form



OK

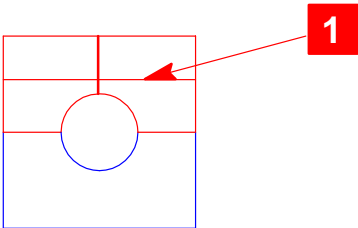
What: Define thin-shell mapped mesh parameters on the top region. Then, define the corners.



1



Done



Define Mesh form

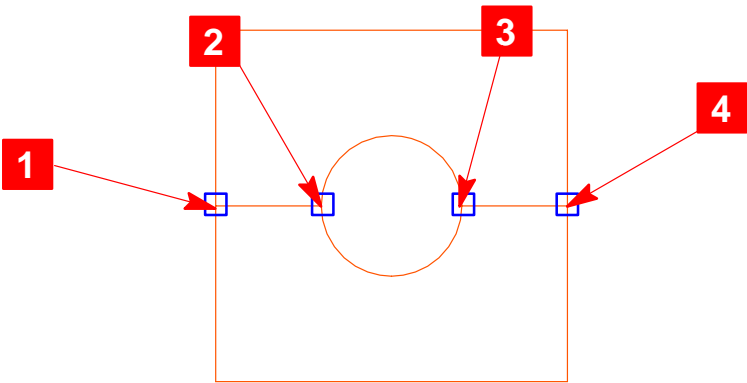
 *Mapped*

 *Mapped Options...*

Mapped Meshing Options form

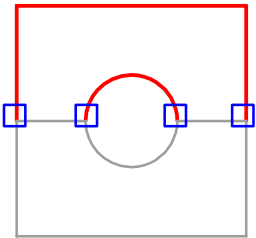
 *Define Corners*

 (shift-pick 1-4)



Mapped Meshing Options form

 *Define Elements/Side*



What: Define the number of elements per side.

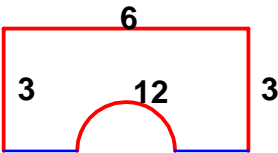
How:



Check I-DEAS Prompt.



number of elements for highlighted sides: 12

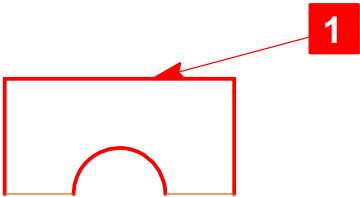


The software determines the distribution of elements per side based on the length of the edges. If you used different dimensions to create your part, your default distribution may differ.



Change distribution

1



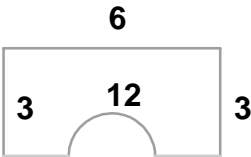
number of elements on the edge: 3

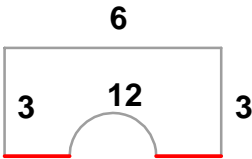


number of elements of edge: 6



Done

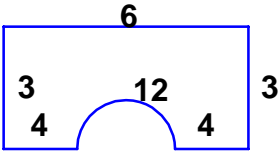




number of elements for highlighted sides: 4



Done

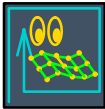


Dismiss

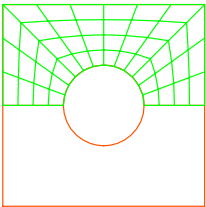
What: Generate the mesh.

How:

Define Mesh form

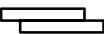


Modify Mesh Preview form



Keep Mesh

Recovery Point



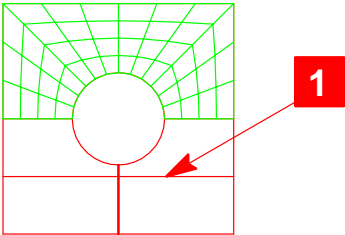
File
Save

What: Define and generate a mesh on the other surface.

How:



1 pick surface



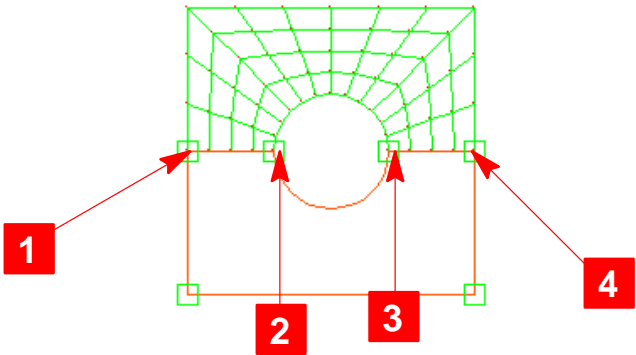
Define Mesh form

 *Mapped*

 *Mapped Options*

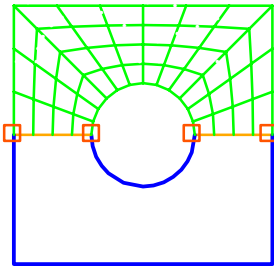
Mapped Meshing Options form

 *Define Corners*




Mapped Meshing Options form

 Define Elements/Side



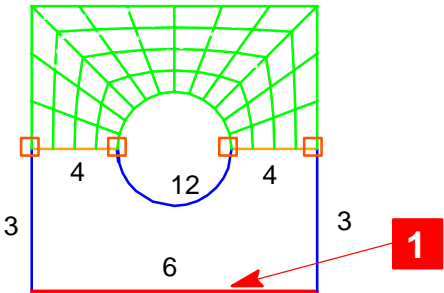
Check I-DEAS Prompt.

 number of elements for highlighted sides: 12



Change distribution

1



 number of elements on the edge: 3

 number of elements of edge: 6



Done

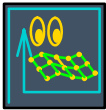
Things to notice

There are no prompts for the elements per side on edges that are shared with the first mesh.

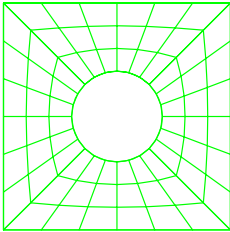
Mapped Meshing Options form



Define Mesh form



Modify Mesh Preview form

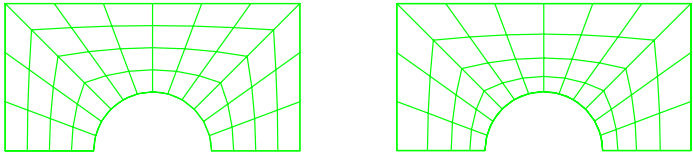


Recovery Point



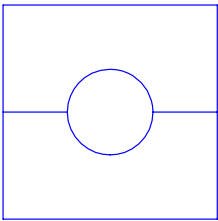
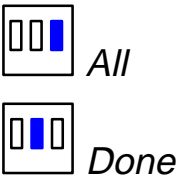
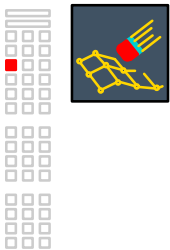
With a radial mesh pattern like this (shown on the left), element lines get closer together in the center toward the hole, causing the elements to become stretched.

One solution to this is to bias the mesh (shown on the right).



What: Delete the mesh on all surfaces. (Don't delete the mesh definition.)

How:

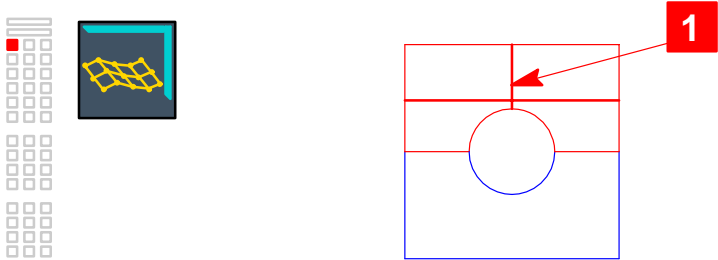


Recovery Point



What: Define edge biasing near the hole so the elements are smaller by a factor of 2.

How:



1 pick surface



Define Mesh form

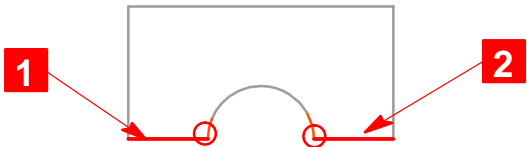


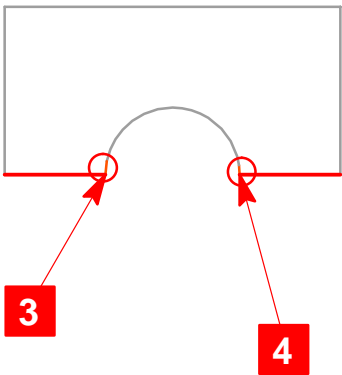
Mapped Meshing Options form



1 pick

2 shift-pick





3

bias: 2

☐

Yes

4

2

☐

Yes

Mapped Meshing Options form

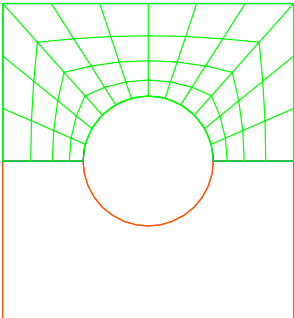
Dismiss

Define Mesh form

OK

What: Generate the mesh.

Hint



Recovery Point



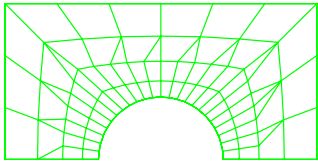
Things to notice

Now element lengths are shorter in the radial direction near the hole, which keeps element aspect ratios more constant.

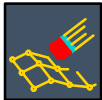
Use unequal number of elements

1 of 3

Another method is to set an unequal number of elements on opposite sides.



First, delete the mesh on all surfaces. (Don't delete the mesh definition.)

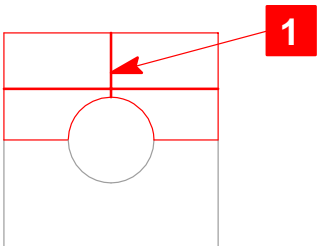


What: Define the elements per side, using an unequal numbers of elements.



1

pick surface



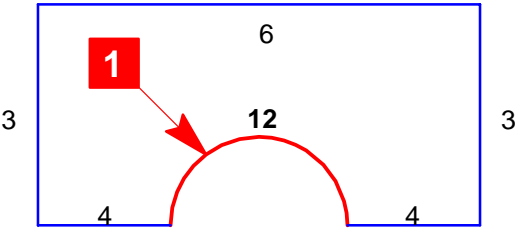
Define Mesh form



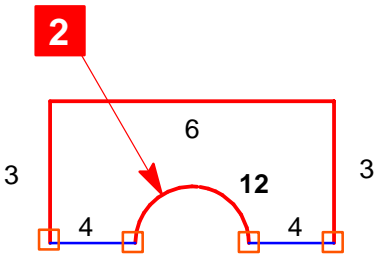
Mapped Meshing Options form




1

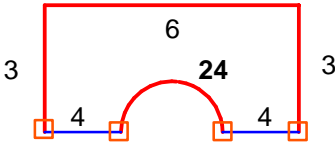


2





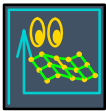
number of elements for highlighted sides: 24



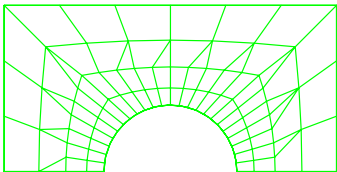
Mapped Meshing Options form



Define Mesh form




Modify Mesh Preview form




Things to notice


Triangular elements are added to the mesh to transition between the unequal number of elements on opposite sides.


For more information on meshing surfaces, see the tips on the next page.

 There are both surface and solid methods to create a part that can be used for meshing surfaces. However, it may be easier to extrude to create a solid part rather than to create a single surface.

 When working with individual surfaces for meshing, create the outer boundary first, then trim or split the surface into regions.

 Remember to use *Sketch in Place*.

 When using *Trim*, remember to pick all the regions to keep.

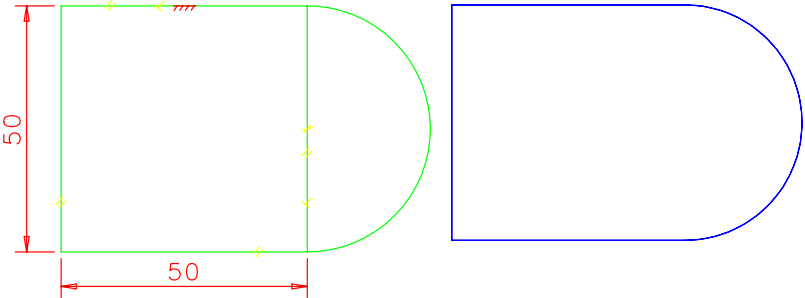
 Use the *Split* option of *Extrude* to divide surfaces. The *Split* icon is not as useful for finite element modeling, because it requires you to create a splitting part first.

On your own...

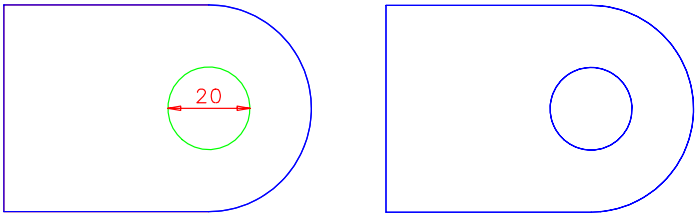
Here is a typical 2D meshing example to try as a review of the techniques in this tutorial.

Hint

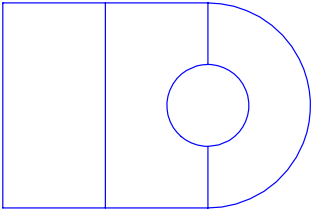
Surface by Boundary.



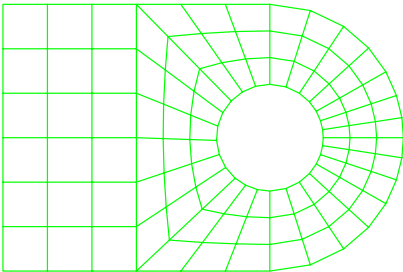
Sketch in Place, Trim



Sketch in Place, Extrude→ Split Surface



Define Shell Mesh, Shell Mesh



You've completed the Mapped Meshing on Surfaces tutorial.

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

Hint



See also...

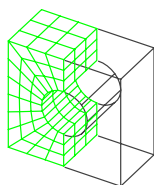
For additional information on the concepts covered in this tutorial, see the following:

 *Help, Manuals, Table of Contents*

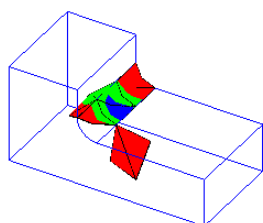
- Simulation: Finite Element Modeling User's Guide
 - Simulation Techniques and Examples
 - Meshing; Creating Nodes and Elements
 - Meshing a Model
 - Icon Overview for Meshing
 - Creating a Mesh

What's next?

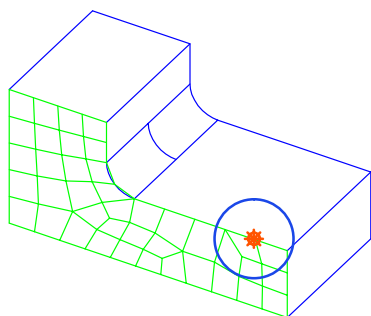
The tutorial "Mapped Meshing of Volumes" expands on the surface techniques just completed.



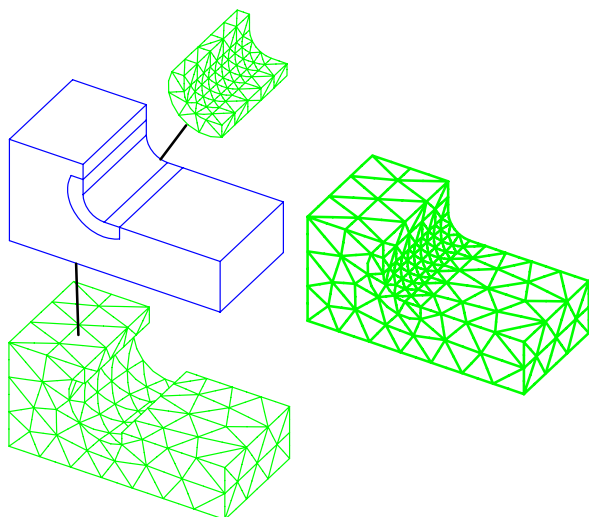
The tutorial on checking element quality shows how to compute element quality checks.



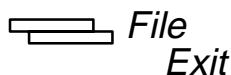
The Boundary Condition Sets tutorial covers information on ways to control node locations.



The Preparing a Part for Analysis tutorial covers 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