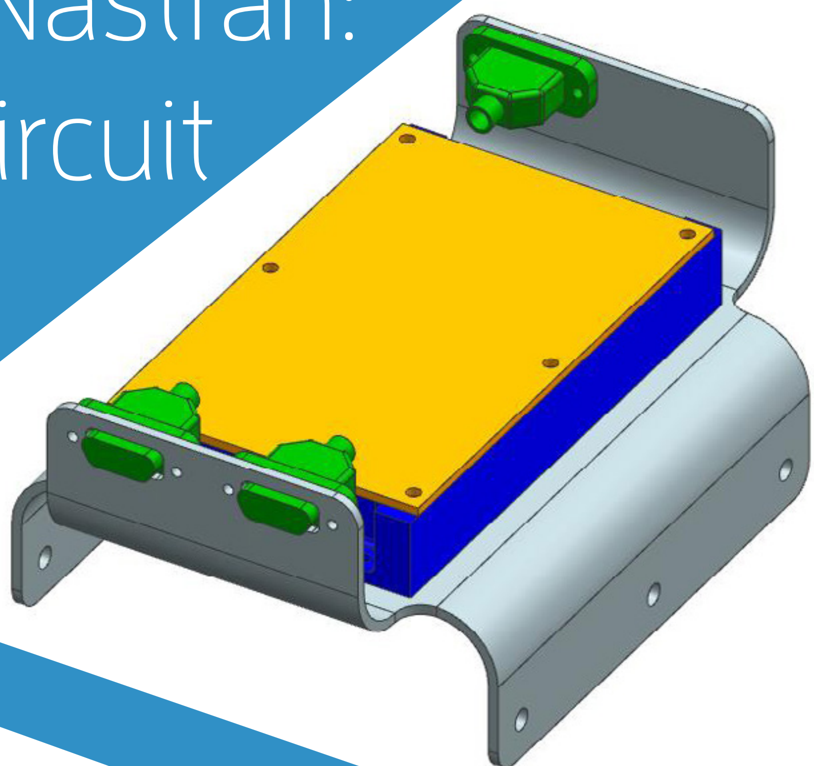
NX 10

NX **TUTORIAL**

NX with NX Nastran: Meshing a Circuit Card



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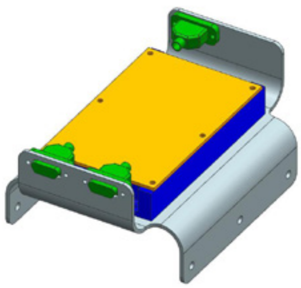
Circuit Card Assembly Tutorial in NX



Software:
NX 10.0

Difficulty Level:
Intermediate

Preceding Tutorial:
None

Input Files Required:
electronics_box_assy.prt
housing_proe.prt
bracket_proe.prt
d_connector_proe.prt

Overview

This document provides a written description of a demonstration of various NX meshing capabilities. The model will be made from a simple generic circuit card geometry file.

This tutorial is part one of a three-part NX CAE tutorial series.

This tutorial is part of a series of free Siemens PLM Software training resources provided by ATA. For more tutorials, whitepapers, videos, and macros, visit ATA's PLM Software website: http://www.ata-plmsoftware.com/resources.

Concepts

- NX 10.0 basics
- Geometry idealization
- Midsurfacing
- Tetrahedral meshing
- Brick meshing
- Shell meshing
- 1D and 0D meshing
- Mesh controls
- Model checks

Contents

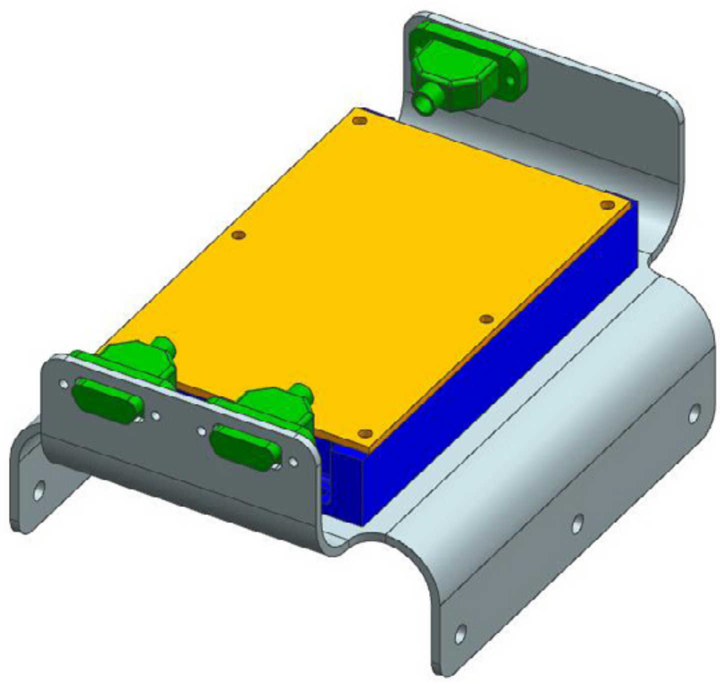
Tutorial

1. Introduction

This document provides a written description of a demonstration of advanced computer-aided engineering (CAE) for NX 10. The model is a simple generic circuit card mounted to a housing with a bracket and D-connectors. The housing mounts to a sheet metal bracket, which also supports three identical D-connectors.

This introductory section is provided for background only. Sections 1.1 and 1.2 demonstrate creating a new part file and importing geometry, while the remainder of section 1 explains view manipulation and introduces the user interface. Alternatively, the assembly part can be directly loaded by opening “electronics_box_assy.prt,” and the user can advance to section 2.

Figure 1-1: ▸
NX demonstration model.



1.1. Create a New File

1. Open NX and click the New icon on the upper toolbar or go to Menu: File → New
2. In the Model tab, set units to inches.
3. Change the name to electronics_box_assy.prt.
4. Change the folder to whatever is desired.
5. Click OK.

1.2. Import an Assembly

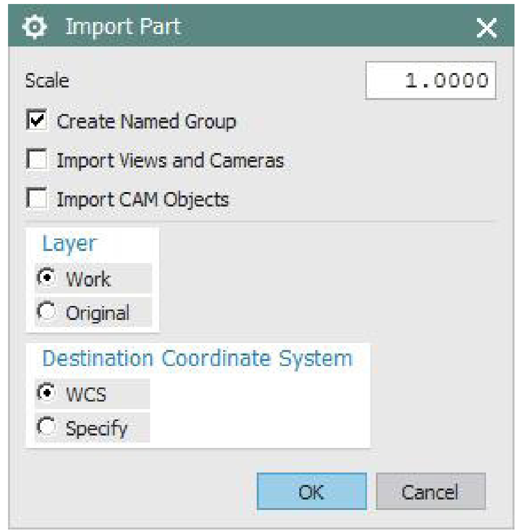


Figure 1-2: ▶
Import Part dialog box.

1. Menu → File → Import → Part
2. Click OK and browse for the electronics box assembly part and select the latest revision.
3. Click OK in the Point dialogue box to place the assembly at the origin.
4. Click cancel to exit the Point dialogue box.

1.3. Change to Full Screen Mode

1. Menu: View → Full Screen OR Alt + Enter OR click the icon just above the graphics window on the right. The same commands can be used to exit full screen mode.

1.4. Manipulate the Model

- Pan – The view will pan using either of these methods:
  1. Holding down the Shift key and the mouse scroll wheel while moving the mouse.
  2. Holding down the mouse scroll wheel and the right mouse button while moving the mouse.
- Zoom – The view will zoom in and out using any one of these three methods:
  1. Rotating the mouse scroll wheel.
  2. Holding down the Ctrl key and the mouse scroll wheel while moving the mouse.
  3. Holding down the mouse scroll wheel and the left mouse button while moving the mouse.

- Rotate – The view will rotate using any one of these three methods:
    1. Holding down the mouse scroll wheel while moving the mouse.
    2. Holding down the Alt key and mouse scroll wheel while moving the mouse.
    3. F8 key – Pressing F8 will orient the model to the nearest orthographic view.

1.5. Bring Up the Command Finder Tool

Because NX contains many commands across a number of various interfaces, the Command Finder can be a very useful tool when you know what tool you want to use but are unsure of where to find it.

1. Menu → Help → Command Finder OR type in Find a Command in the top right of the screen.

Figure 1-3: ▸
Command Finder tools.

For example:

2. Type "point" into the Search box and hit Enter.

NX returns results relevant to the search in the same dialog box with a description of each command found, a path to where each command can be located, and—if a command is not available in the current application—which application(s) it can be used in.

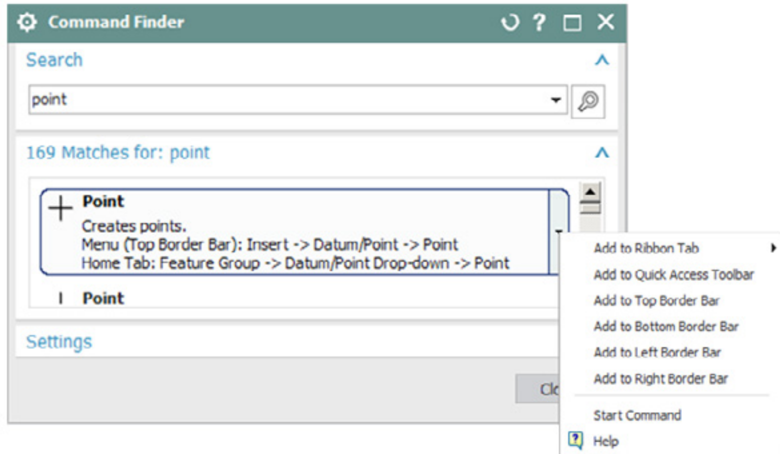


Figure 1-4: ▸
Command Finder "Point" search results.

3. Hovering over the command will show where it is located in the menu (if the command is available in the current application).
4. Clicking the dropdown menu will give options to add the command to different toolbars. This feature can be used to add shortcuts that are not on the default toolbar.
5. Clicking Start Command will start the command. This can also be done by simply clicking on the result without going through the dropdown list.
6. Clicking Help will bring up the NX Help document in your browser, allowing you to get more information on the particular command.
7. Close the Command Finder dialog box.

1.6. Know the Assembly Navigator

The assembly navigator shows every part that has been added to the assembly, including multiple occurrences of the same part. You can show or hide any part in the assembly simply by clicking the checkbox next to the part's name.

For assemblies that contain many components, it may be beneficial to combine multiple instances of the same part in the Assembly Navigator on a single row by right-clicking in the navigator and selecting Pack All.

You can also search for a particular component by name in the Assembly Navigator by right-clicking in the navigator and selecting Find Selected Components.

Figure 1-5 is an example of what the Assembly Navigator will look like for this demo.

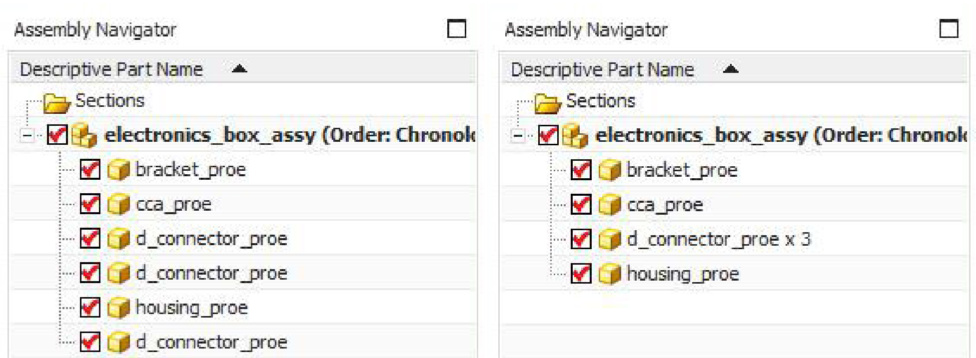


Figure 1-5: ▶
Assembly Navigator: unpacked (left) and packed (right).

1.7. Save the Individual Parts

1. File → Save → Save All
2. File → Close → All Parts

2. Meshing the Housing Part

2.1. Open the Housing Part

1. Open housing_proe.prt.
2. Application → Simulation → Advanced or File → Start → Advanced Simulation
3. Resource Bar → Simulation Navigator → Right-click on housing_proe.prt and select New FEM…
4. The default name appends _fem1 to the part file and uses the same location as its parent. For this demonstration, these defaults are acceptable.

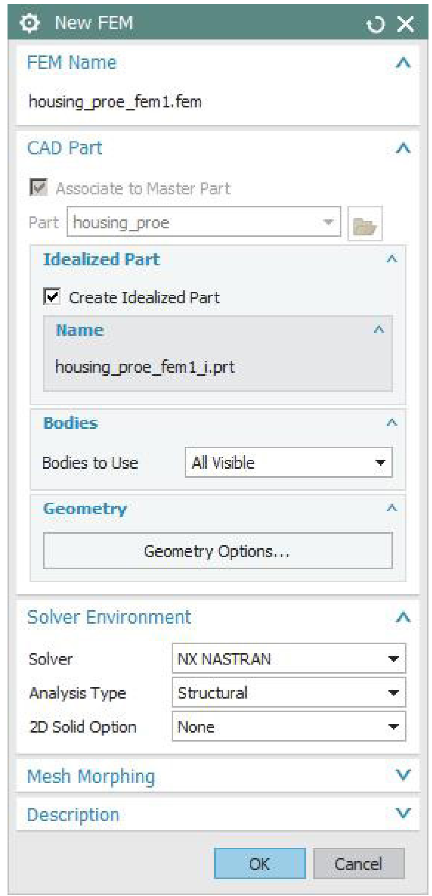


Figure 2-1: ▶
New FEM dialog box.

5. Click OK.
6. Make sure Create Idealized Part is checked.
7. Under CAD Part – Bodies, Bodies to Use is set to All Visible by default. The dropdown list also shows All, Select, and None. All Visible accepts all bodies that are not suppressed in the model. In this part (and all others in this demonstration), there is only one body that has been imported from the part model and it is by default not suppressed, so All Visible is acceptable to use.
8. Click OK.

2.2. Assign a Material to the Part

1. Menu → Tools → Materials → Assign Materials or Home → Properties → Assign Materials
2. Select the housing body.
3. Select Aluminum 6061 from the list of Library Materials.
4. Click OK.

You can also build your own local library which can be easily reused across models if needed. You can create a new material from scratch or you can export an existing material in the NX material library and edit individual properties. If you want to export an existing material, begin at step 5 below.

1. Menu → Tools → Materials → Manage Materials
2. Click Create Material under Material List – New Material.
3. Edit the name and properties to what is desired and click OK.
4. Close the Manage Materials dialog box.
5. Menu → Tools → Materials → Manage Library Materials
6. Set Type to Export Material to Library.
7. Set Target Material Library to Single MatML File and enter the path where you would like the material library to be located, followed by the name you would like the file to have.
8. Set Source Material List to Local Materials and select the newly created material from the list if you created a new material. If you are exporting an existing material, set Source Material List to Library Materials and check the box for NX Material Library. Select the material from the list that you would like to export.
9. Click OK.

This custom material can now be reused across models by loading the MatML Library into the material list in that session. If you would like to edit a material that has been exported to a user library, continue with the following steps.

1. Menu → Tools → Materials → Manage Library Materials
2. Set Type to Edit Library Material.
3. Check Library Material List – User MatML Library and browse to the .xml file created in the above steps.
4. The material should now appear in the Materials list in the window. Select it and click OK.
5. In the Isotropic Material dialog box, change any properties as desired and click OK.

2.3. Promote the Idealized Part

1. In the Simulation Navigator under Simulation File View, double-click on housing_proe_fem1_i.
2. Click OK on the Idealized Part Warning.

This is the idealized part that will be used to manipulate the geometry to simplify the model and make it easier to mesh. Before you can modify the geometry, any parts that are going to be changed must either be promoted or WAVE linked.

3. Home → Start → Promote.

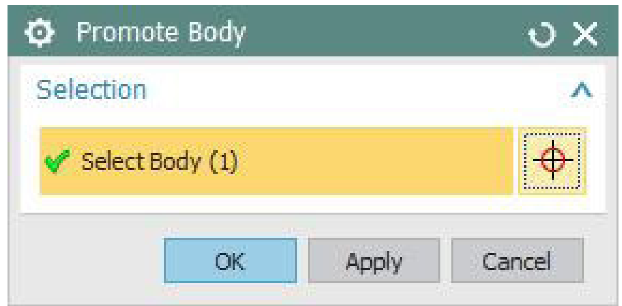


Figure 2-2: ▶
Promote Body dialog box.

4. Select the housing body.
5. Click OK.

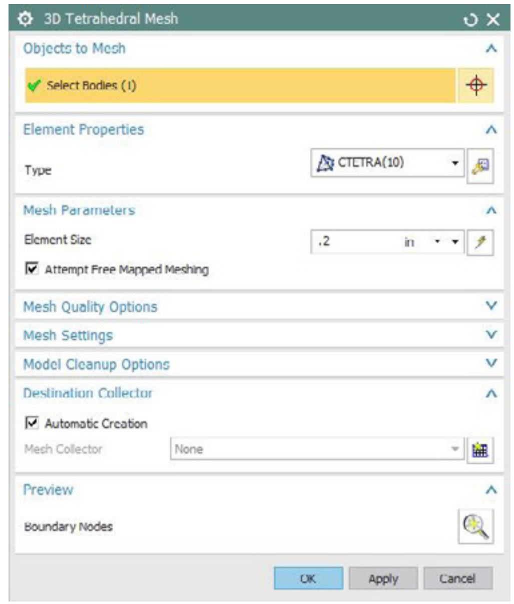
Promoting or WAVE linking geometry of the idealized part creates an associative link between the idealized part and the original part. This provides an editable linked copy of the original part without affecting the original part, meaning that any changes made to the geometry of the idealized part will not change the geometry of the original part. However, updates of the original geometry will be reflected in the idealized part.

2.4. Tetrahedral Mesh the Part As Is

Before making any changes to the idealized part, we will see how the mesher deals with the small holes and fillets in the original geometry and how to make quick changes in the FEM.

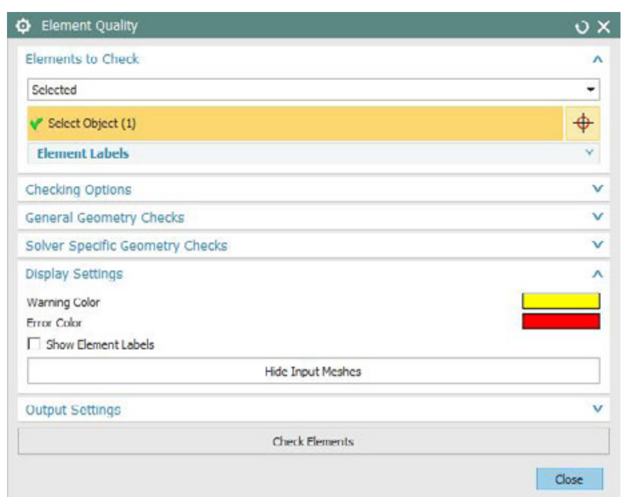
1. In the Simulation Navigator under Simulation File View, double-click on housing_proe_fem1.
2. Home → Mesh → 3D Tetrahedral Mesh

Figure 2-3: ▸
3D Tetrahedral Mesh dialog box.



3. Select the housing body.
4. Set the Mesh Parameters – Element Size as 0.2 in.
5. Click OK.
6. In the Simulation Navigator, click the checkmark next to Polygon Geometry to hide the geometry and view just the mesh.

Figure 2-4: ▸
Model Check dialog box.



7. Home → Checks and Information → Element Quality
8. Expanding the dropdown list shows a number of checks that NX can perform on the model. Under Output Settings, change Report to Failed and Warning.
9. Select the mesh and click Check Elements.

Figure 2-5: ▶
Housing tetrahedral mesh, 0.2 in.

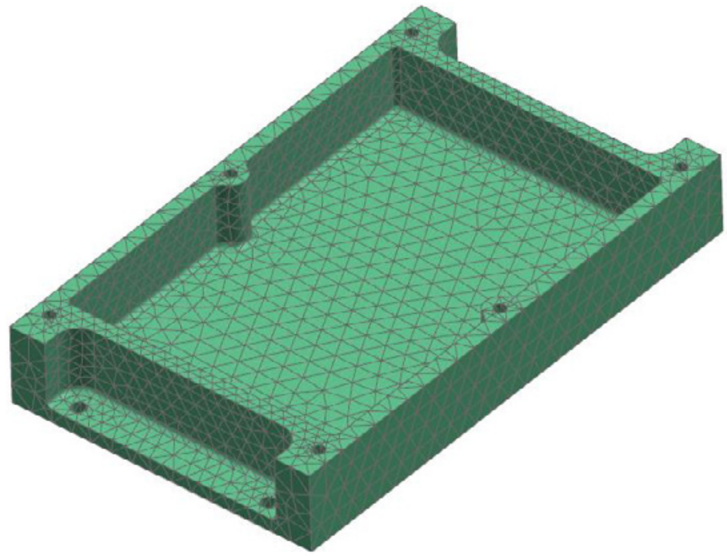


Figure 2-6: ▶
Results of the element shape check, 0.2 in.

Overview

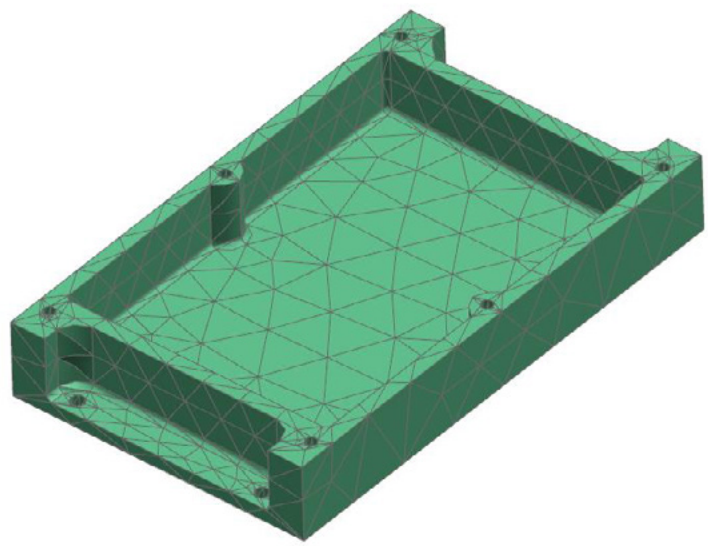
	Number Failed	Number Warning	Number Checked
Elements	0	7706	7706

Check	Number Failed	Number Warning	Worst Value
Jacobian Sign	0	0	1.00000
Jacobian Zero	0	7706	0.00005
Volume	0	7706	0.00002
Axisymmetric Consistent Y	0	0	-N/A-
Axisymmetric +X	0	0	-N/A-
Aspect Ratio	0	1	11.21430
Skew Angle	0	0	-N/A-
Maximum Interior Angle	0	0	-N/A-
Minimum Interior Angle	0	0	-N/A-
Taper	0	0	-N/A-
Warp Factor	0	0	-N/A-
Face Warp Coefficient	0	0	-N/A-
Edge Point Length Ratio	0	0	0.70710
Element Offset Length Ratio	0	0	-N/A-

As can be seen in Figure 2-6, the element shape check should have returned that no bad elements were generated among the 7706 elements checked. Now we will generate a coarser mesh, i.e., one with fewer elements.

1. In the Simulation Navigator, expand 3D Collectors to view the mesh collector and the actual mesh.
2. Double-click 3d_mesh(1).
3. Change the Element Size to 0.6 in.
4. Click OK.

Figure 2-7: ▶
Housing tetrahedral mesh, 0.6 in.



5. Home → Checks and Information → Element Quality
6. Select the mesh and click Check Elements.

Figure 2-8: ▶
Results of the element shape check, 0.6 in.

Overview

	Number Failed	Number Warning	Number Checked
Elements	0	2012	2012

Check	Number Failed	Number Warning	Worst Value
Jacobian Sign	0	0	1.00000
Jacobian Zero	0	2012	0.00002
Volume	0	2008	0.00001
Axisymmetric Consistent Y	0	0	-N/A-
Axisymmetric +X	0	0	-N/A-
Aspect Ratio	0	42	18.42249
Skew Angle	0	0	-N/A-
Maximum Interior Angle	0	0	-N/A-
Minimum Interior Angle	0	0	-N/A-
Taper	0	0	-N/A-
Warp Factor	0	0	-N/A-
Face Warp Coefficient	0	0	-N/A-
Edge Point Length Ratio	0	0	0.70679
Element Offset Length Ratio	0	0	-N/A-

As can be seen in Figure 2-8, the element shape check again returns that no bad elements were generated. This mesh contains roughly a quarter of the elements of the previous mesh. Although no elements failed the check, this coarse mesh is not suitable to analyze the model due to the shape and number of elements in what would be the high-stress areas.

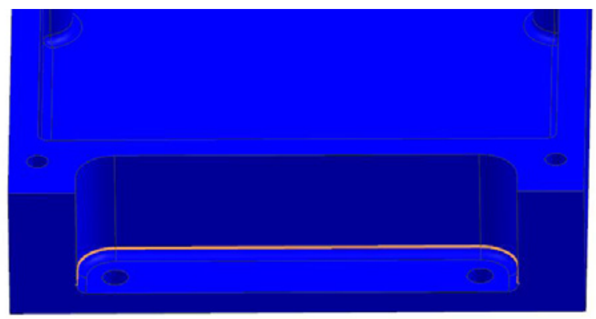
7. Undo the coarse mesh by pressing Ctrl + Z.

2.5. How NX Surface Coats the Volume

In NX, solid mesh quality can be controlled if desired by surface meshing or by using mesh controls. Surface meshes can be toggled so they will not be used for export to the solver, just for meshing technique. It is also possible to quickly create a mesh with fewer elements by glossing over unimportant details in the geometry by using model cleanup tools on the FEM rather than doing this with the design geometry or the idealized part.

1. In the Simulation Navigator, hide the tetrahedral mesh and show the polygon geometry.
2. Home → Polygon Geometry → Merge Face
3. Select the horizontal curves that run along the vertical walls of the open slots on the outside of the part as shown in Figure 2-9.

Figure 2-9: ▶
Merge face selected curves.



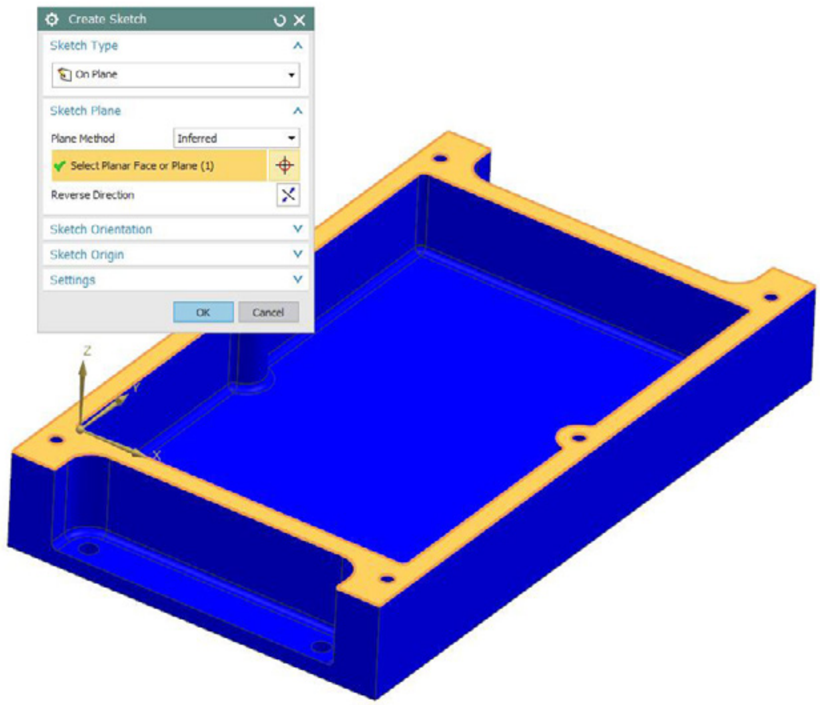
4. Click OK.
5. In the Simulation Navigator, show the tetrahedral mesh by clicking the checkbox next to 3D Collectors.
6. Home → Context → Update
7. Running an element quality check will show that around 175 elements were eliminated by using the Merge Face command on those sets of curves. This is a quick and easy way to reduce the number of elements but may result in unfavorable meshes around fillets, curves, and holes.
8. In the Simulation Navigator, right-click the Solid(1) mesh collector under 3D Collectors and select Delete.
9. In the Simulation Navigator, right-click housing.prt.1 (1) under Polygon Geometry and select Recreate and Update to return the model to its original geometry.

2.6. Sketch Circles at the Six Top Bolt Holes

This is done to be sure to have local areas to connect the model with rigid elements easily.

1. In the Simulation Navigator under Simulation File View, double-click on housing_proe_fem1_i.
2. Menu → Insert → Sketch in Task Environment…
3. Select the top face, shown in Figure 2-10, as the sketch plane. Click OK.

Figure 2-10: ▶
Top face as the sketch plane.



4. Home → Curve → Circle
5. Create six circles using the centers of the bolt holes on the same plane as the sketch as the circle centers. Each circle will have a diameter of 0.31 in.
6. Sketch → Finish

2.7. Use Idealize Geometry to Remove Extraneous Features

1. Home → Geometry Preparation → Idealize Geometry

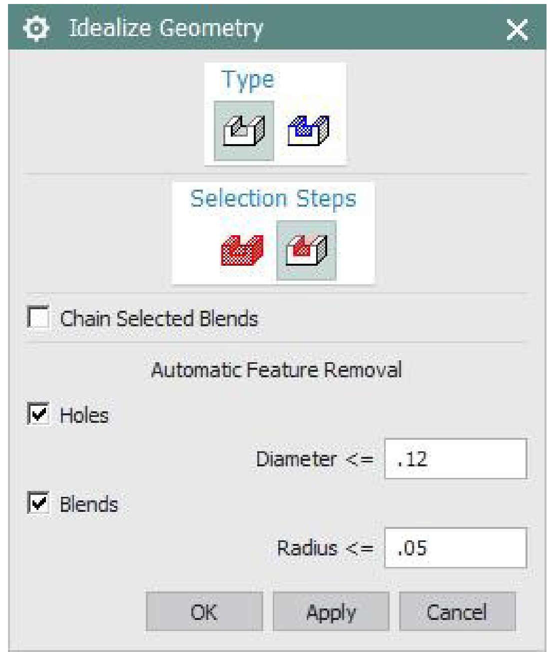


Figure 2-11: ▸
Idealize Geometry dialog box.

2. Select the housing body.
3. Remove the small holes and fillets by checking the appropriate boxes and entering the above diameter and radius values.
4. Click OK.

Your part should look like the one in Figure 2-12.

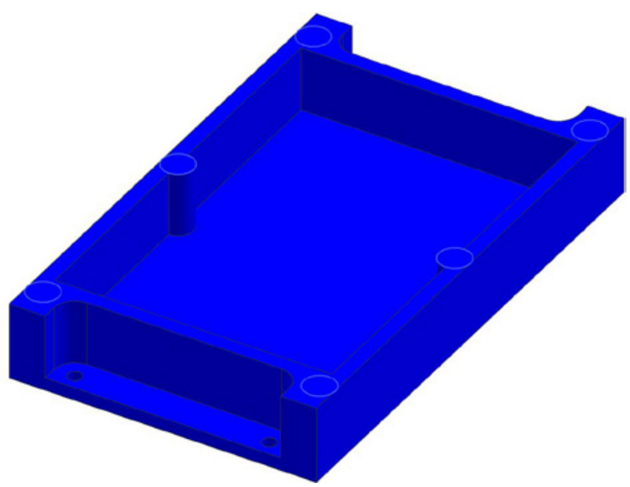


Figure 2-12: ▸
Housing part after idealization.

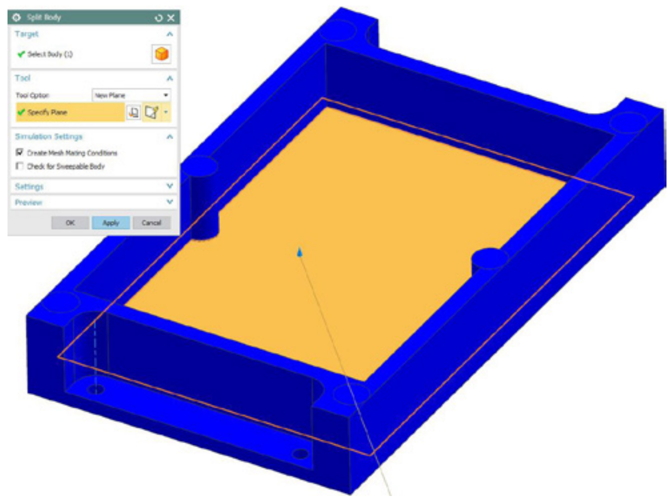
2.8. Divide the Top Face

1. Home → Geometry Preparation → Divide Face
2. Select the top face of the housing body as the Faces to Divide and the six circles sketched earlier as the Dividing Objects.
3. Click OK.

2.9. Split the Housing Body

1. Home → Geometry Preparation → Split Body

Figure 2-13: ▸
Splitting the housing body.



2. Select the housing body as the Target.
3. Select New Plane from the dropdown list for Tool - Tool Option and specify the orange face in Figure 2-13 as the tool plane.
4. Make sure Simulation Settings - Create Mesh Mating Conditions is checked and click OK.

2.10. Generate Connection Elements for the Bolted Joints

1. In the Simulation Navigator under Simulation File View, double-click on housing_proe_fem1.
2. Home → Connections → 1D Connection
3. Set Type to Point to Face.
4. Select the center point of one of the circles created in section 2.6 as the Source and the face enclosed by the circle as the Target.
5. Set Connection Element – Element Properties – Type to RBE2.
6. Click Apply.
7. Repeat until each of the six center points are connected to the housing. Uncheck Connection Element – Destination Collector – Automatic Creation after creating the first connection elements so that all the connections are stored in the same mesh collector.

8. Repeat this process also for the four through-holes on the lower body of the housing. Choose the center point of the circle defining each hole on the bottom face of the housing part as the Source and the inner wall of the hole as the Target.

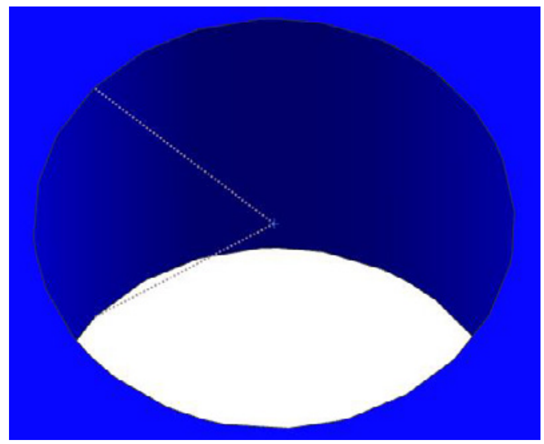


Figure 2-14: ▶
Through-hole connection recipe.

2.11. Mesh the Upper Housing Body

Swept meshing, often referred to as brick meshing, is more powerful than just extruding plate elements (as it is associative to the geometry for boundary conditions, follows the shape of the side faces, updates with geometry updates, etc.).

1. Hide the lower body of the housing that was made in step 2.9. This can be done by setting the selection filter to Polygon Body, selecting the lower body, and pressing Ctrl-B.
2. Home → Mesh → 3D Swept Mesh
3. Set Type to Until Target.
4. Select the eight faces shown in orange in Figure 2-15 as the Source Faces.

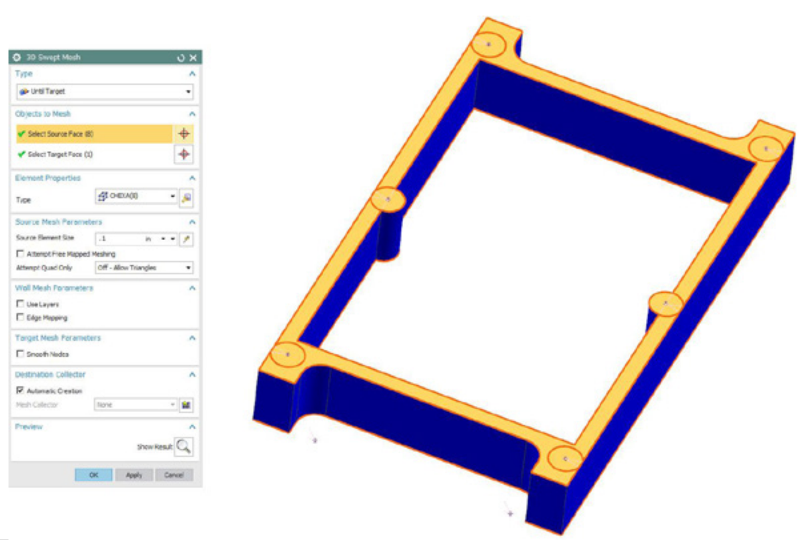


Figure 2-15: ▶
Swept meshing the upper body.

5. For the Target Face, select the bottom face.
6. Make sure Element Properties – Type is set to CHEXA(8).
7. Enter 0.1 in. for Source Mesh Parameters – Source Element Size.
8. Make sure Source Mesh Parameters – Attempt Free Mapped Meshing is unchecked and Attempt Quad Only is set to Off – Allow Triangles.
9. Make sure Wall Mesh Parameters – Edge Mapping and Target Mesh Parameters – Smooth Nodes are also both unchecked.
10. Click OK.

2.12. Fix the Upper Housing Mesh

Looking at either of the bosses in the middle of the upper body, you can notice little slivers from the divided face where the boss meets the wall (shown in Figure 2-16). These small slivers create problems with the mesh, so it is desirable to remove them.

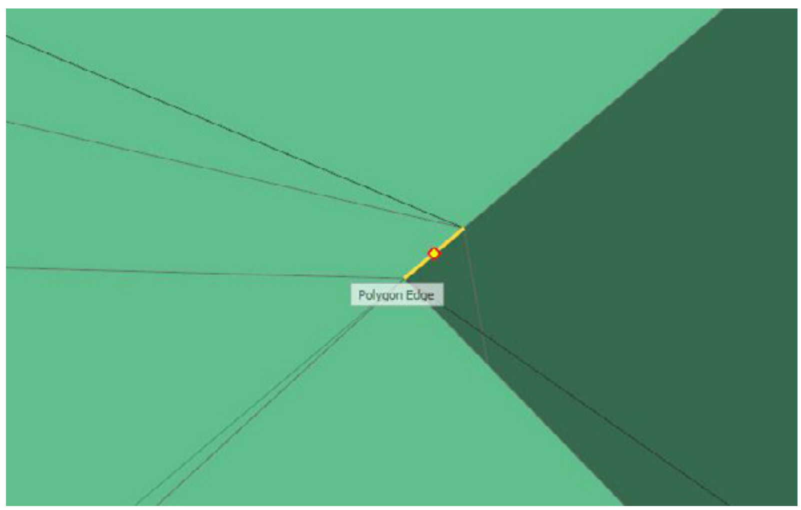
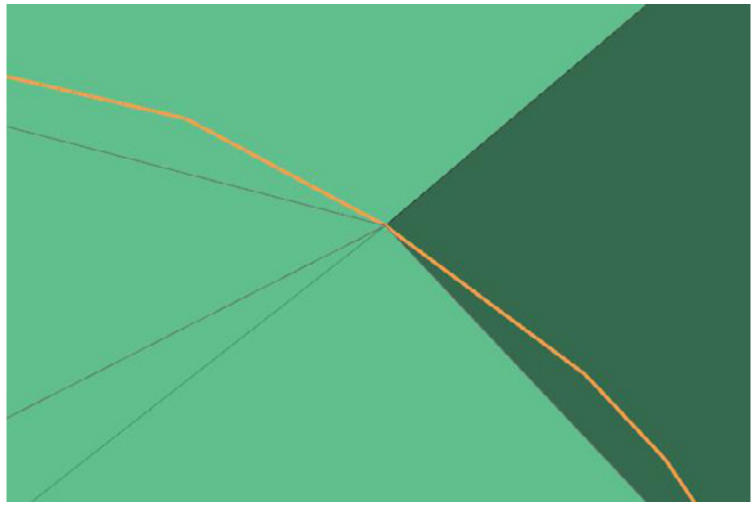


Figure 2-16: ▸
Middle boss slivers.

1. Polygon Geometry → Collapse Edge
2. Select a point on the line of each sliver where the wall meets the boss for all four instances.
3. Click OK.
4. Advanced Simulation → Update Finite Element Model

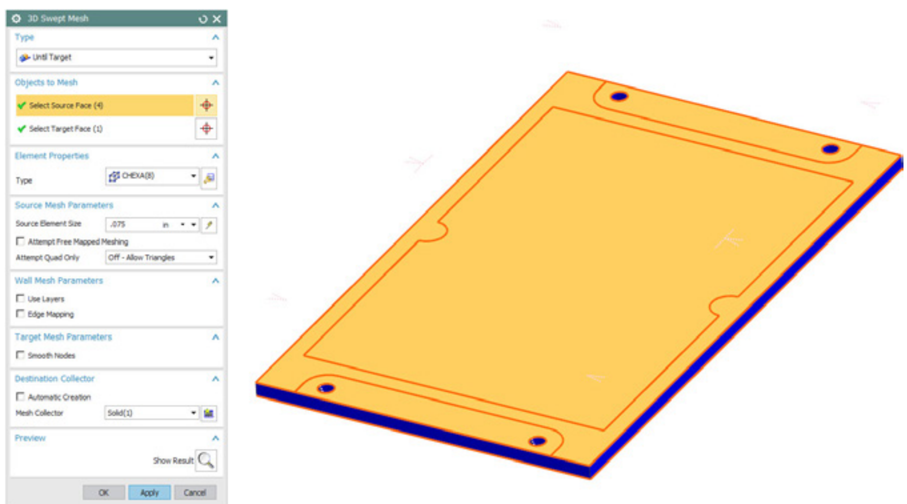
Figure 2-17: ▶
Slivers removed after collapsing the edge.



2.13. Mesh the Lower Housing Body

1. Hide the mesh and upper body of the housing, and show the lower body.
2. Home → Mesh → 3D Swept Mesh
3. Select the four faces on the orange plane shown in Figure 2-18 below as the Source Faces.

Figure 2-18: ▶
Swept meshing the lower body.



4. Select the bottom face of the lower body as the Target Face.
5. Make sure Element Properties – Type is set to CHEXA(8).
6. Enter 0.075 in. for Mesh Parameters – Element Size.
7. Make sure Source Mesh Parameters – Attempt Free Mapped Meshing is unchecked and Attempt Quad Only is set to Off – Allow Triangles.
8. Make sure Wall Mesh Parameters – Edge Mapping and Target Mesh Parameters – Smooth Nodes are also both unchecked.

9. Uncheck Destination Collector – Automatic Creation.
10. Click OK.
11. Save the housing FEM file.

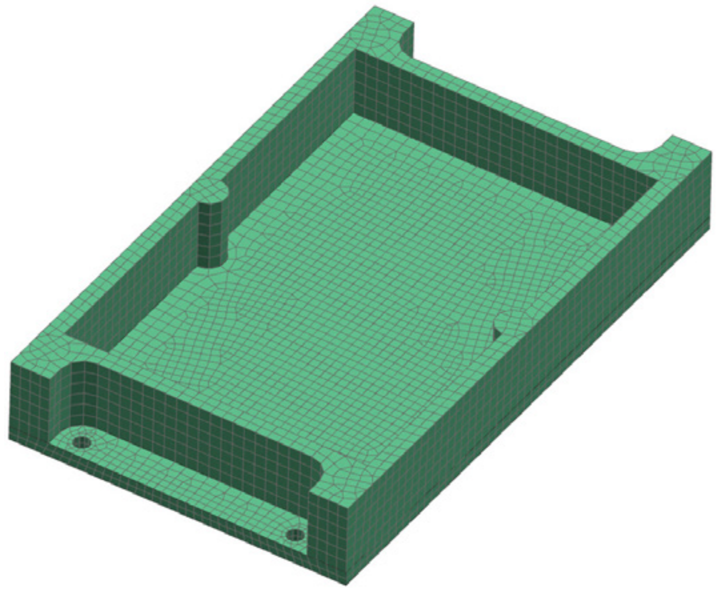


Figure 2-19: ▶
Final meshed housing.

3. Meshing the Bracket

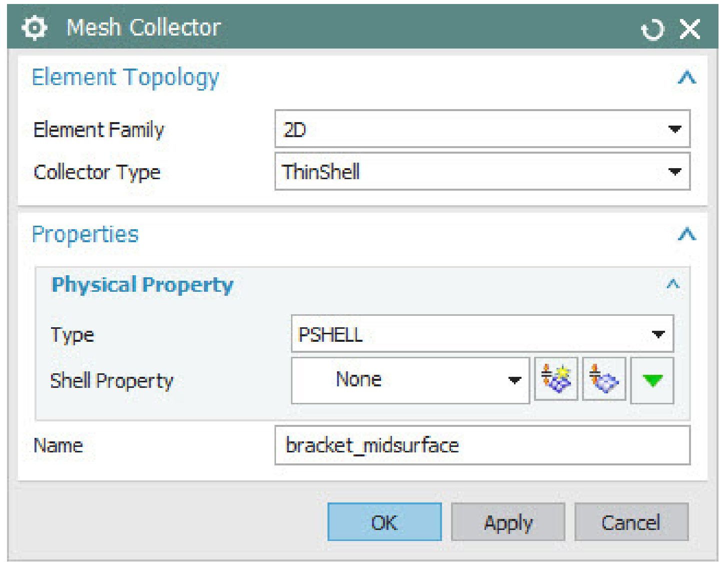
3.1. Open the Bracket Part

1. Open bracket_proe.prt.
2. NX should still be in Advanced Simulation mode; if it is not, enter Advanced Simulation mode now.
3. In the Simulation Navigator, right-click on bracket_proe.prt and select New FEM…
4. Click OK.
5. Make sure Create Idealized Part is checked and Bodies to Use is set to All Visible and click OK.

3.2. Assign a Material to the Mesh

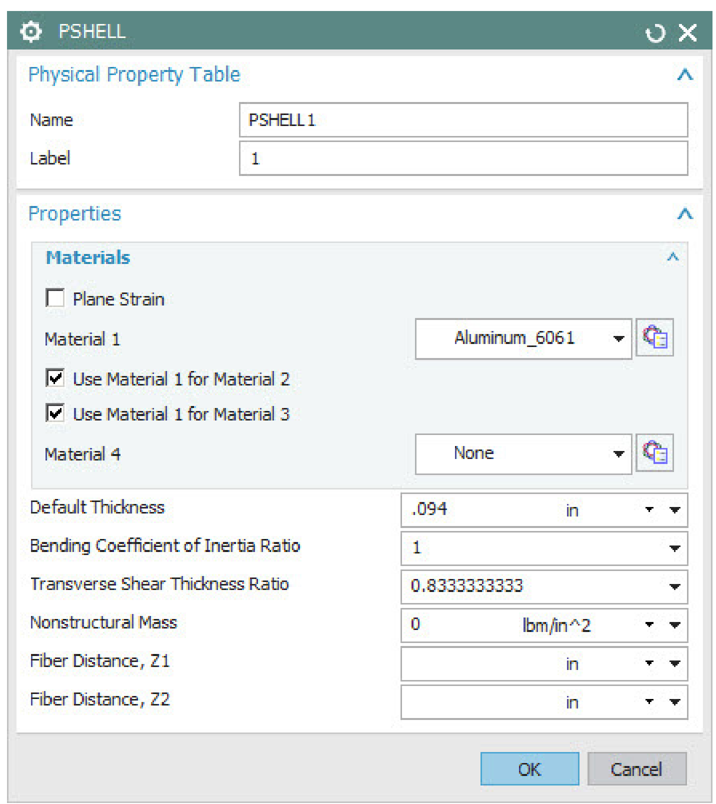
1. Properties → Mesh Collector

Figure 3-1: ▶
Mesh Collector dialog box.



2. Select 2D for Element Topology – Element Family.
3. Select ThinShell for Element Topology – Collector Type.
4. Make sure Properties – Type is set as PSHELL.
5. Click Create Physical… on the Properties – Shell Property line.

Figure 3-2: ▶
Physical Shell Properties dialog box.



6. Click Choose Material on the Properties – Materials – Material 1 line.
7. Choose Aluminum_6061 from the list and click OK.
8. Enter 0.094 in. for Properties – Default Thickness.
9. Click OK on both windows.

3.3. Generate the Midsurface

1. In the Simulation Navigator under Simulation File View, double-click on bracket_proe_fem1_i.
2. Advanced Simulation → WAVE Geometry Linker.
3. Set Type to Body and select the bracket part.
4. Click OK.
5. Geometry Preparation → Midsurface by Face Pairs.

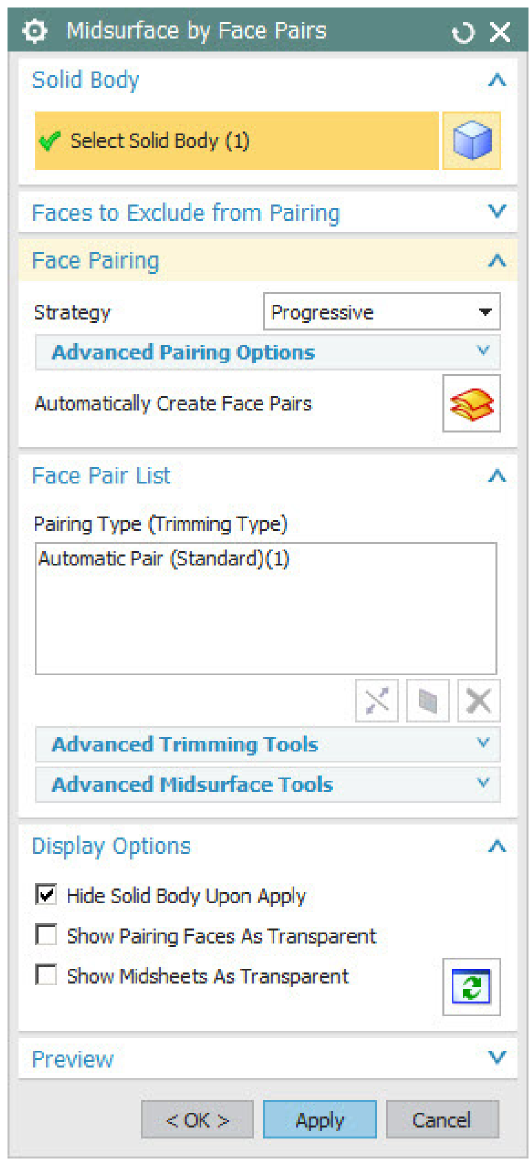


Figure 3-3: ▶ Midsurface dialog box.

6. Select the bracket as the Solid Body.
7. With Face Pairing - Strategy set to Progressive, click the Automatically Create Face Pairs button.
8. Make sure Hide Solid Body Upon Apply is checked and click OK.

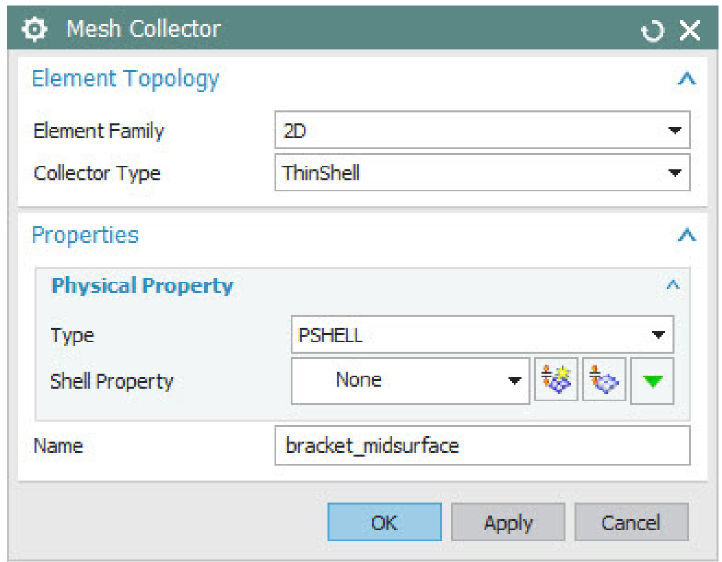


Figure 3-4: ▶
The midsurfaced bracket.

3.4. Generate Circular Imprints Around Bolt Holes

1. In the Simulation Navigator under Simulation File View, double-click on bracket_proe_fem1.
2. Polygon Geometry → More → Circular Imprint

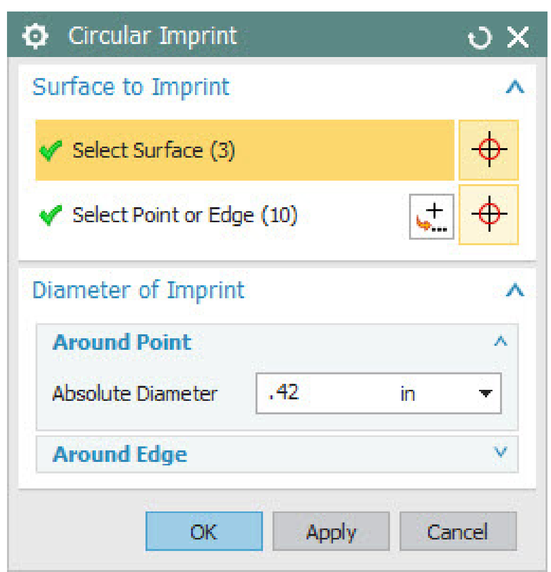
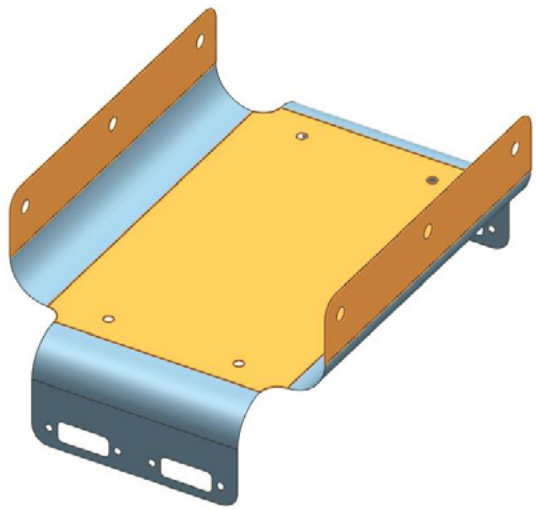


Figure 3-5: ▶
Circular Imprint dialog box.

3. With just the midsurface shown, select the highlighted faces in Figure 3-6 for Surfaces to Imprint – Select Surface and select the center point of each of the holes on the highlighted surfaces for Surfaces to Imprint – Select Point or Edge.

Figure 3-6: ▶
Surfaces to circular imprint.

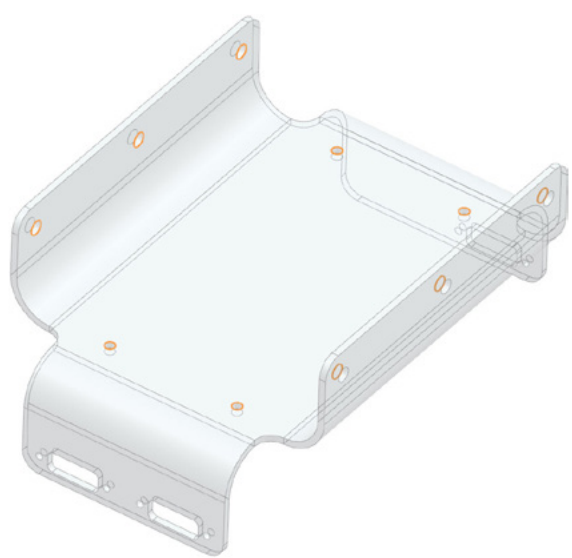


4. Enter 0.42 in. for the absolute diameter of the circular imprint.
5. Click OK.

3.5. Generate Connection Element Definitions

1. In the Simulation Navigator, expand Polygon Geometry and show the original geometry by clicking the checkbox corresponding to bracket.prt.4.
2. Utilities → Point
3. With the original geometry shown, create a point at the center point of the highlighted holes on the inner side of the part as shown in Figure 3-7.

Figure 3-7: ▶
Point creation guide.



4. Hide the original geometry so just the midsurface is visible.
5. Connections → 1D Connection
6. With Type set to Point to Face, select one of the points just created on the horizontal surface as the Source and the face of the corresponding circular imprint as the Target.
7. Set Connection Element – Element Properties – Type to RBE2 and click Apply.
8. Uncheck the Automatic Creation box and create the three remaining connection elements on the horizontal plate.
9. Change Type to Point to Edge.
10. Select one of the six remaining points just created as the Source and the edge of the corresponding hole as the Target. Click Apply.
11. Create the five remaining connection elements to be used to attach the D-connectors to the bracket.

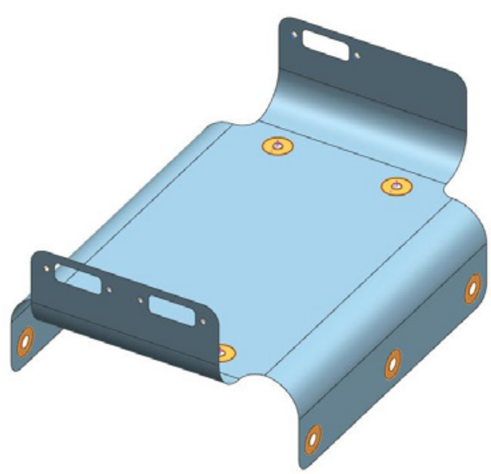
Because a mesh has not been created for the bracket yet, the connections just created exist as temporary recipes. Once a mesh is created, these recipes will automatically create connection elements between the source points and the nodes on the respective face/edge.

3.6. Create Mesh Controls

Sometimes to create a better mesh, it is desirable to control how elements are created in certain areas of a part, typically holes and fillets. For the bracket, we will define how many elements are to be created around all of the holes, the circular imprints, and the fillets on the horizontal face.

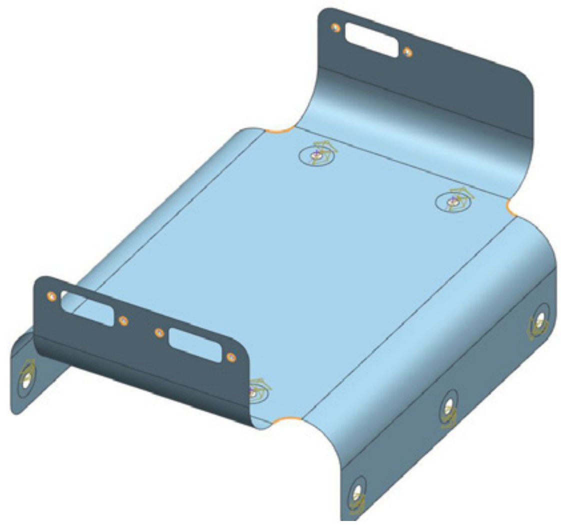
1. Mesh → Mesh Control
2. Set Density Types to Number on Edge.
3. Select the faces between holes and their circular imprints as shown in Figure 3-8 as the Targets. There should be ten selections.

Figure 3-8: ▸
Mesh control selections, 1.



4. Set Number on Edge – Number of Elements to 8.
5. Click Apply.
6. Select the edges of the six small holes on the vertical faces and the four corner fillets on the horizontal face as shown in Figure 3-9 as the Targets. There should be ten selections.

Figure 3-9: ▸
Mesh control selections, 2.

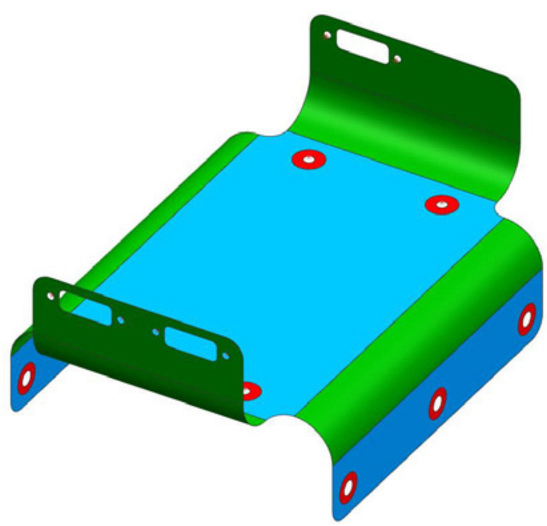


7. Set Number on Edge – Number of Elements to 6.
8. Click OK.

3.7. Generate the Shell Mesh

To create an optimal mesh, the bracket will be meshed in three sections which have been divided by color in Figure 3-10. It is important to mesh the sections in the following order: green → red→ blue. The order is important because the second and third meshes are seeded, i.e., their boundary nodes are defined, by the previous meshes.

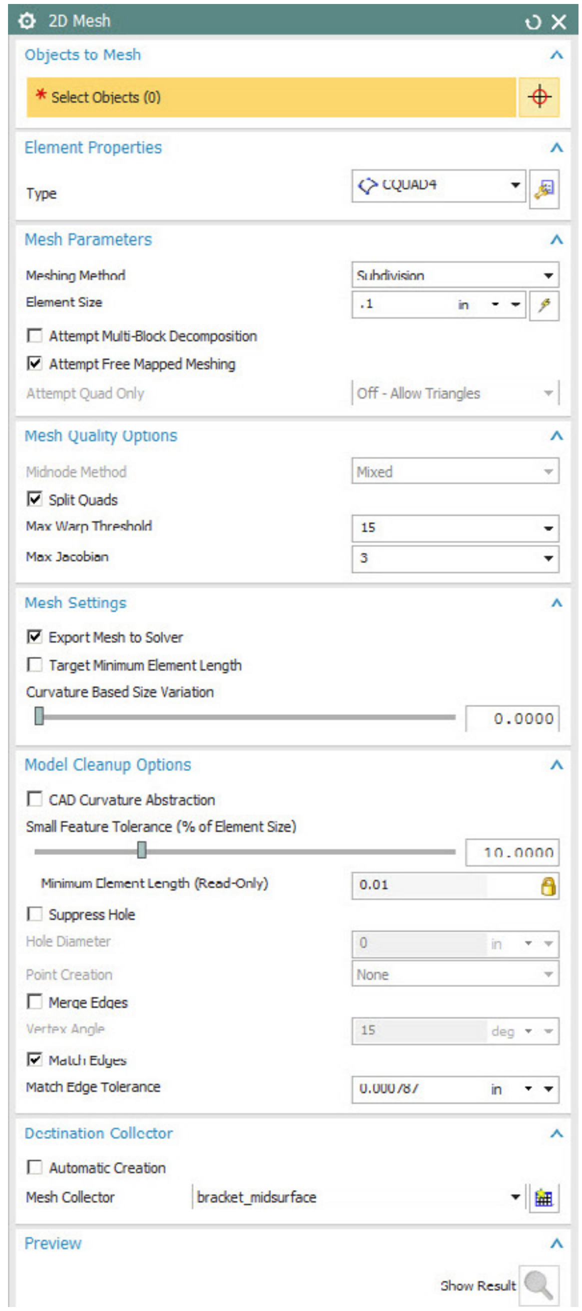
Figure 3-10: ▸
Separations for meshing the brackets.



1. Mesh → 2D Mesh

For each section, the properties of the mesh should be the same as seen in Figure 3-11. The only exception is that the Mesh Parameters – Meshing Method will be set to Subdivision for the green and blue sections while it will be set to Paver for the red section.

Figure 3-11: ▸
Bracket mesh properties.



2. Select the six faces of the green section in Figure 3-10 as the Objects to Mesh with the properties set to match those in Figure 3-11. Click Apply.

3. Select the ten faces of the red section in Figure 3-10 as the Objects to Mesh, changing Mesh Parameters – Meshing Method to Paver and keeping all the other properties the same. Click Apply.
4. Select the three faces of the blue section in Figure 3-10 as the Objects to Mesh, changing Mesh Parameters – Meshing Method back to Subdivision and keeping all the other properties the same. Click OK.

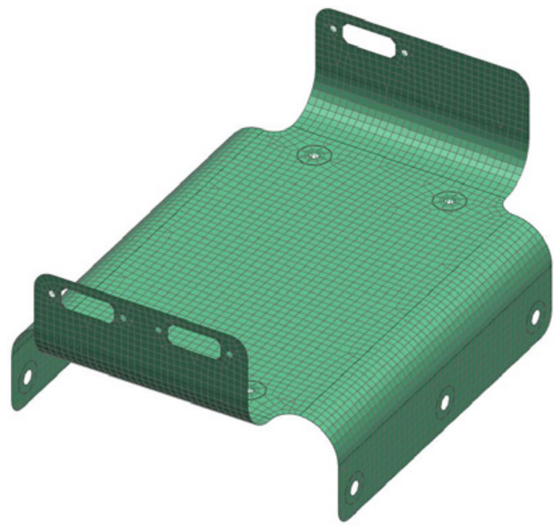


Figure 3-12: ▸
2D meshed bracket.

3.8. Turn On Shell Element Thickness Display

1. In the Simulation Navigator, right-click on the mesh collector containing the three sectioned meshes.
2. Click Edit Display.
3. Check the box corresponding to Display 2D Element Thickness and Offset.
4. Click OK.
5. Save the bracket FEM file.

Figure 3-13: ▸
Final meshed bracket.

4. Meshing the D-Connector

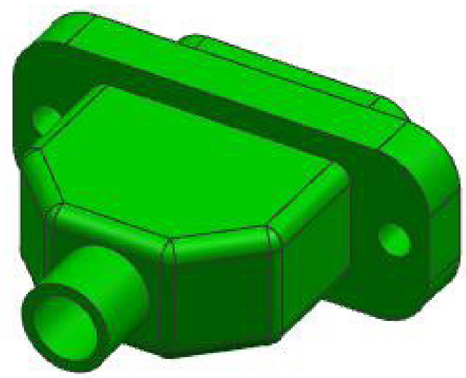


Figure 4-1: ▶
D-connector.

4.1. Open the D-Connector Part

1. Open d_connector_proe.prt.
2. NX should still be in Advanced Simulation mode; if it is not, enter Advanced Simulation mode now.
3. In the Simulation Navigator, right-click on d_connector_proe.prt and select New FEM…
4. Click OK.

For the D-connector, an idealized part is not needed because no modifications will be made to the geometry. Instead, the D-connector will be modeled as a concentrated mass with rigid body elements connecting it to the bolt locations.

5. Uncheck Create Idealized Part and make sure Bodies to Use is set to All Visible and click OK.

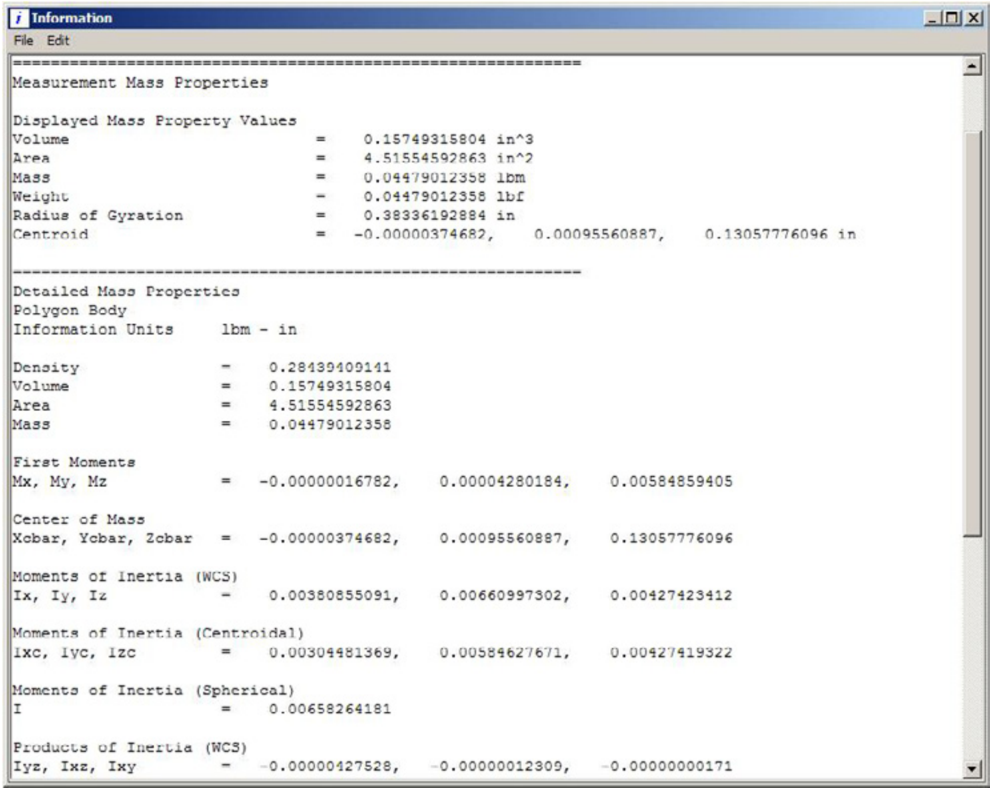
4.2. Assign a Material to the Part

1. Properties → Assign Materials
2. Select the D-connector body.
3. Select AISI_Steel_1005 from the list of Library Materials.
4. Click OK.

4.3. Compute Mass Properties for the Part

1. Menu → Analysis → Measure Bodies…
2. Select the D-connector body.
3. Check the box corresponding to Results Display – Show Information Window

Figure 4-2: ▶
D-connector mass properties.

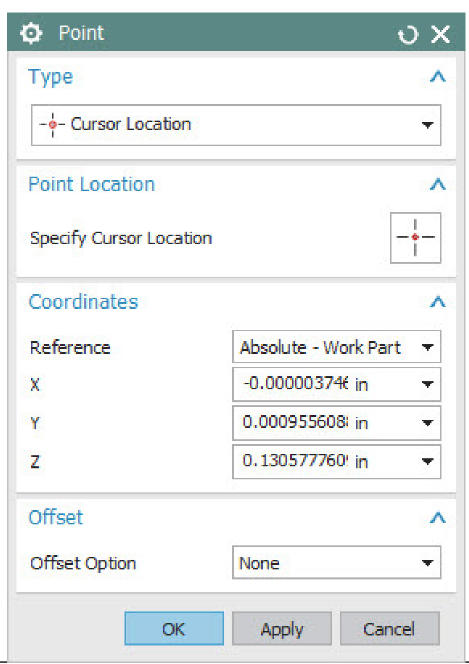

```
Information
File  Edit
==============================================================
Measurement Mass Properties

Displayed Mass Property Values
Volume                          =    0.15749315804 in^3
Area                            =    4.51554592863 in^2
Mass                            =    0.04479012358 lbm
Weight                          =    0.04479012358 lbf
Radius of Gyration              =    0.38336192884 in
Centroid                        =   -0.00000374682,    0.00095560887,    0.13057776096 in

--------------------------------------------------------------
Detailed Mass Properties
Polygon Body
Information Units     lbm - in

Density               =    0.28439409141
Volume                =    0.15749315804
Area                  =    4.51554592863
Mass                  =    0.04479012358

First Moments
Mx, My, Mz            =   -0.00000016782,    0.00004280184,    0.00584859405

Center of Mass
Xcbar, Ycbar, Zcbar   =   -0.00000374682,    0.00095560887,    0.13057776096

Moments of Inertia (WCS)
Ix, Iy, Iz            =    0.00380855091,    0.00660997302,    0.00427423412

Moments of Inertia (Centroidal)
Ixc, Iyc, Izc         =    0.00304481369,    0.00584627671,    0.00427419322

Moments of Inertia (Spherical)
I                     =    0.00658264181

Products of Inertia (WCS)
Iyz, Ixz, Ixy         =   -0.00000427528,   -0.00000012309,   -0.00000000171
```

4. We are interested in the mass and centroid of the part as well as the moments and products of inertia, which are shown in Figure 4-2. It is easiest to leave this information window open for the next steps, allowing you to copy and paste values from the window.

4.4. Generate a Point at the CG

1. Utilities → Point

Figure 4-3: ▶
Create point dialog box.



2. Enter the X, Y, and Z coordinates of the centroid into the appropriate boxes and click OK.

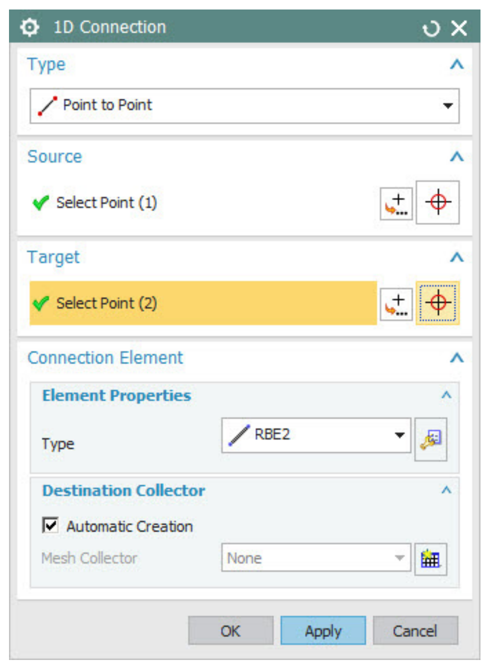
4.5. Generate a Concentrated Mass Element at the CG

1. In the Simulation Navigator, uncheck the box corresponding to Polygon Geometry to view the point just created.
2. Mesh → More → 0D Mesh
3. Select the point created at the centroid as the Objects to Mesh.
4. Select CONM2 for Element Properties – Type. Click the Edit Mesh Associated Data button in the same section of the dialog box.
5. Input the mass, mass moments of inertia, and mass products of inertia found in section 4.3 in the appropriate boxes. Click OK.
6. Click OK to create the 0D Mesh.

4.6. Generate a Connection Element Definition

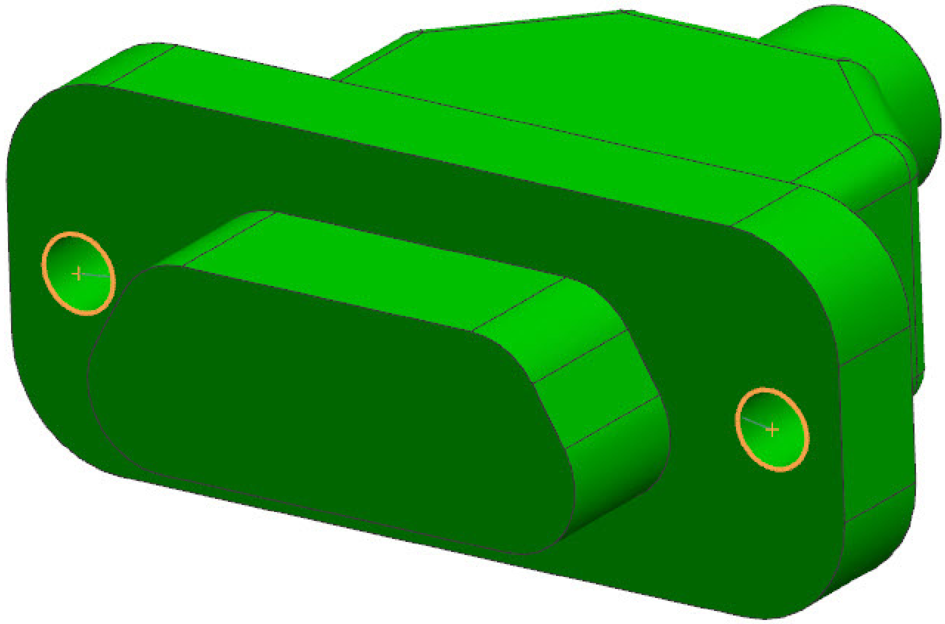
1. Connections → 1D Connection

Figure 4-4: ▶
1D Connection.



2. Change Type to Point to Point from the dropdown list.
3. Select the point created at the centroid in section 4.4 as the Source.
4. Select the arc-centers shown in Figure 4-5 as the Target.
5. Make sure Connection Element – Element Properties – Type is set to RBE2 and click OK.

Figure 4-5: ▶
Arc-center locations.



6. Save the connector FEM file.

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