MFG500024

Combining Solid, Shell, and Line Elements with Inventor Nastran

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Learning Objectives

- Explain the difference between solid, shell, and line elements.
- Efficiently simplify CAD geometry into Nastran idealizations.
- Bond shell element structures using a continuous mesh.
- Connect elements using rigid body connectors and face splits.

Description

Using one element type is not always feasible when performing stress analysis on large, complex CAD assemblies. Mixing solid, shell, and line elements together can produce a mesh that's more efficient and more accurate. This approach can drastically reduce analysis time and allow for more automation in the digital prototyping process. In this class, Ed Gillman will explain the key differences between the finite element types and where they are best utilized. He will then demonstrate the process of converting assemblies into mixed element models and what techniques can be used to properly connect the mesh.

Speaker(s)

Ed Gillman is an Manufacturing Applications Expert at IMAGINiT Technologies, an Autodesk Reseller serving North America. He has extensive experience with Inventor Nastran, Inventor CAM, Fusion 360, FeatureCAM, and PowerMill. Over his career as a Mechanical Engineer, Ed developed spacecraft structural components, consumer products, and a patented manufacturing method for creating custom molded spinal orthotics from 3D-Scan data. Due to his experience in advanced manufacturing and product development, he provides a unique perspective.

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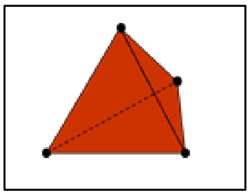
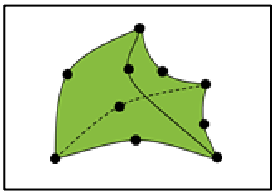
Inventor Nastran Element Types

Inventor Nastran has three different types of finite elements that can be applied to a model. Each element type has it's own set of pros and cons. They can be combined together to form an efficient and accurate stress analysis solution.

Solid Elements – 3D

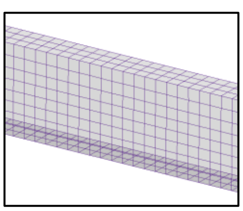
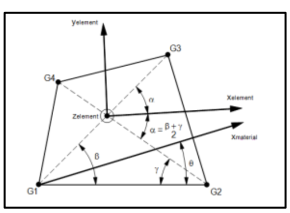



Solid Elements are three-dimensional tetrahedrons. Inventor Nastran supports Linear Tetrahedrons with 4 nodes and Parabolic Tetrahedrons with 10 nodes. Solid Elements have 3 translational degrees of freedom (Tx, Ty, Tz). This means each node can translate in the X, Y, or Z direction – but can't rotate.

Solid elements are often preferred because the CAD model requires little to no preparation. Complex geometry details are accurately captured and boundary conditions can be applied to any entity (face, edge, or vertex).

The primary disadvantage of solid elements is they can greatly increase analysis processing time in larger models. When working with big, multi-part assemblies this can really start to add up. Another issue with solid elements is they can be too stiff for thin-walled components. The mesh will need 2-3 solid elements through the thickness of thin sections to accurately capture bending results.

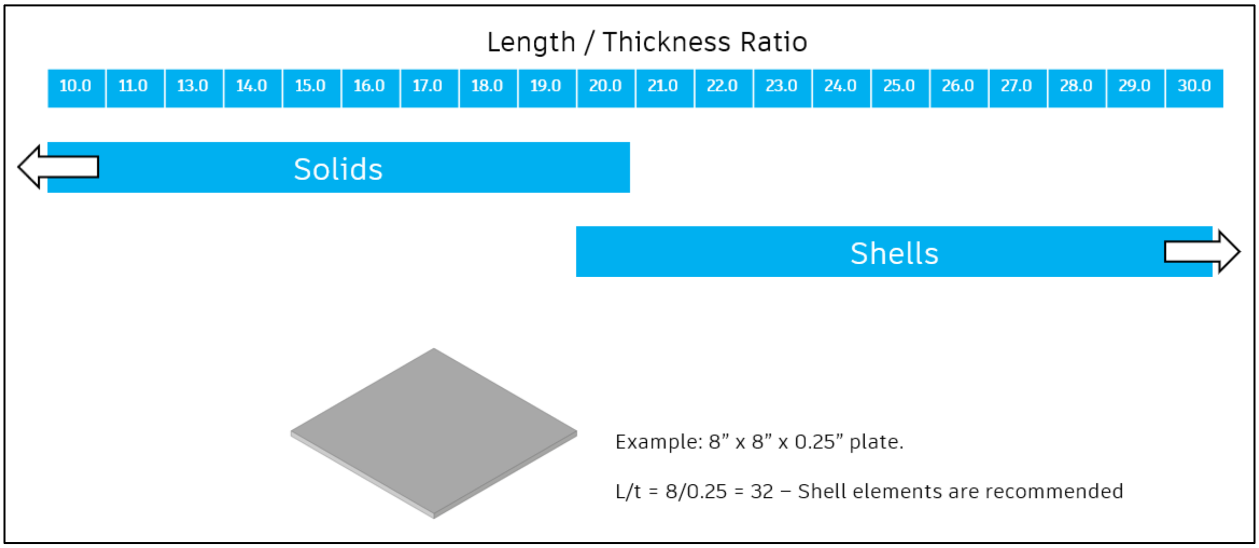
Shell Elements – 2D

Shell elements are two-dimensional triangle or quadrilateral surfaces. Inventor Nastran supports Linear and Parabolic triangles with 3 and 6 nodes, respectively. Linear and Parabolic quadrilaterals with 4 and 8 nodes are also supported. Shells have 5 degrees of freedom (Tx, Ty, Tz, Rx, Ry). An additional sixth degree of freedom (drilling DOF) can be added by editing the element advanced settings. Shell element idealizations can be created by converting the faces of existing solid bodies or applied directly to surface bodies.

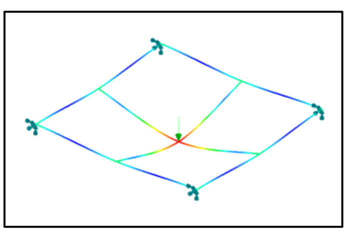
Shell elements are commonly used to idealize thin-walled components with a consistent cross-sectional thickness. They can capture bending and buckling of thin-walled bodies with high accuracy. Since they use fewer nodes and elements, the processing time is much faster than solids. Additionally, shells can be bonded to other shells with continuous meshing. This feature is excellent for analyzing welded frames and requires fewer contacts.

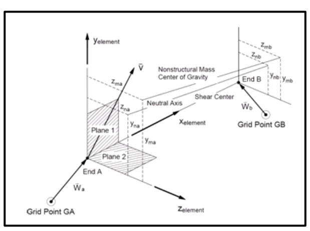
Use the following chart to decide when to apply shell or solid elements:



The biggest disadvantage of shell elements is the time required to prepare a CAD model for this type of idealization. Inventor Nastran's "Offset Surface" and "Midsurface" tools are fast, but leave gaps between components – making continuous meshing impossible. The native surfacing tools also struggle to convert "I" and "C" structural shapes with multiple fillets. Both Inventor and Fusion 360 have tools to generate, trim, extend, and stitch surfaces to build a model ready for continuous shell meshing. Another issue with shell elements is they can't mesh complex parts like hubs, shafts, bearings, etc.

Line Elements – 1D





Line Elements are one-dimensional entities with nodes along their length. Each node has 6 degrees of freedom (Tx, Ty, Tz, Rx, Ry, Rz). The strength and stiffness at each node of a line element is defined by the cross sectional properties of the shape. These properties can be entered manually, imported from the Inventor Frame Generator environment, or calculated from a dimensioned shape.

Line elements only require a 2D or 3D sketch, making them fast and easy to work with. Solve times are typically 30 seconds or less. Since traditional beam bending formulas are used to calculate stresses – line elements make it easy to compare with hand calculations. Complex connections (e.g. pins, bearings, clamps) can't be modeled with line elements, so they work best with primarily welded structures.

Benefits of Mixed Element Modeling

Mixed element modeling is intended for large, complex models with hundreds of thousands, maybe millions of elements. The problem with using an exclusively solid element mesh is not just solve time – other things start to slow down too (e.g. adding constraints, changing material properties, viewing results).

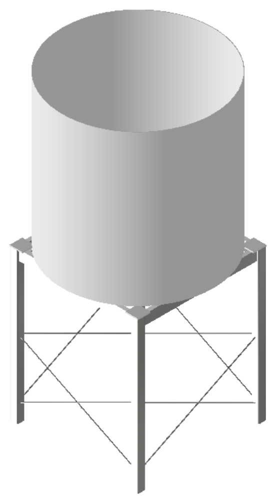


FIGURE 1 – WATER TANK FRAME ASSEMBLY

Take for example this large water tank frame shown in Figure 1. The assembly is 20 feet tall and has around 30 components. Using only solid elements is the simplest approach, but not always the most efficient. By simplifying features, and including shell and line elements in the mesh, the model can be drastically improved. As shown in Table 1 below, the total number of elements, contacts, and overall solve time is significantly different.

	Solid Elements Only	Mixed Element Mesh
# of Elements	674,331	29,721
# of Contact Sets	43	2
Solve Time	10 minutes	30 seconds

TABLE 1 – WATER TANK FRAME DETAILS

Creating a mixed element model certainly takes more time on the front end, but that time is repaid during the Inventor Nastran setup and analysis. In this example, the difference was approximately 10 minutes. In larger structural models, this can be hours.

Tips for Analyzing Large Structures

Performing finite element analysis on large structures is hard. Here are some tips to handle these types of analyses more effectively. You don't have to use all of them, but even using one of these can be significant.

1. Apply a mixed element mesh to limit the total number of nodes. Focus your attention on the simplifying the largest components first.

2. Apply continuous meshing to the shell mesh to reduce total number of contact pairs. Nothing slows down an analysis more than contacts. Try to eliminate them wherever possible.

3. Apply ½ and ¼ symmetry to limit overall model size. If your model is symmetric this is a quick way to simplify.

4. Simplify non-structural masses to Forces, Moments, or Concentrated Mass. Examples of these would be things like fuel tanks, batteries, motors, and electronics.

Connecting Shells to Solids

Shell elements have more degrees of freeom than solid elements. When connecting them with Inventor Nastran the DOF mismatch can cause problems. As shown in Figure 2, shell edges that are bonded to a solid will act link hinge.

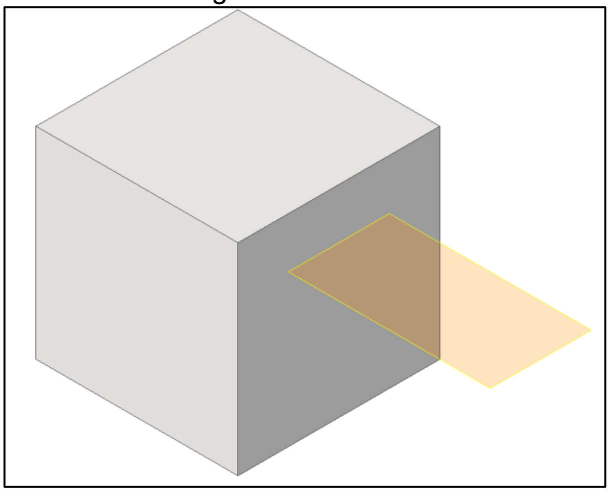


FIGURE 2 – SHELL ELEMENTS CONNECTED TO SOLIDS BY THEIR EDGE ROTATE LIKE A HINGE

Applying a bonded contact in this scenario will not work. When solving, the result will either be an E5000 error or a very large displacement (e.g. 70,000 in). There are 3 different ways to resolve this problem and connect the shell mesh:

Option 1 – Offset Bonded Contact

Apply an Offset Bonded contact between the solid face and the shell edge (Figure 3). This is the simplest method for connecting the degrees of freedom. When creating the contact make the max activation distance 10-20% larger than the mesh size. The downside to this approach is it's difficult to control the contact area. Contacts typically create stress singularities at the corners of the shell elements, so a finer mesh might be required.

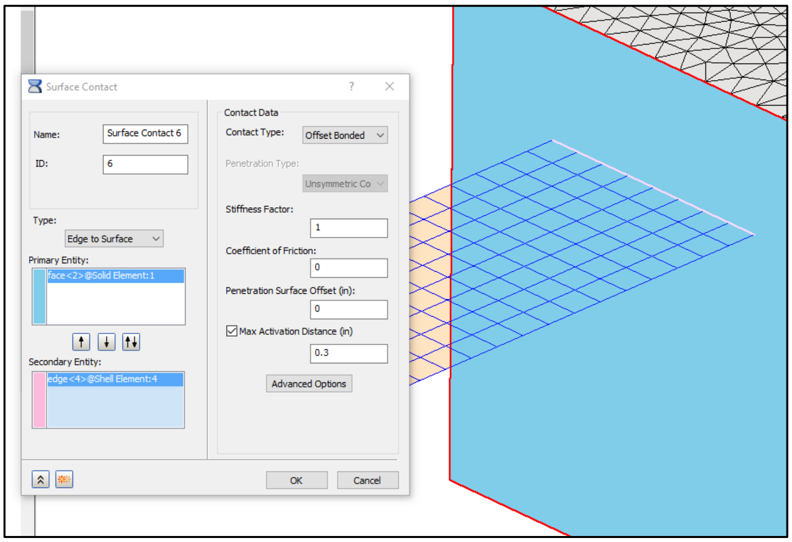


FIGURE 3 – ADDING AN OFFSET BONDED CONTACT BETWEEN THE SOLID FACE AND SHELL EDGE

Option 2 – Weld Geometry

Add a Solid or Shell weld geometry to the assembly (Figure 4). Try to match the size and shape of the weld when creating the CAD. This requires more prep time to create and assemble the additional components. In Nastran, use a Bonded contact between the weld body and the shell elements. Using solid elements to idealize the weld body can create artificial stress concentrations, so use a surface to represent the weld if possible (Figure 5).

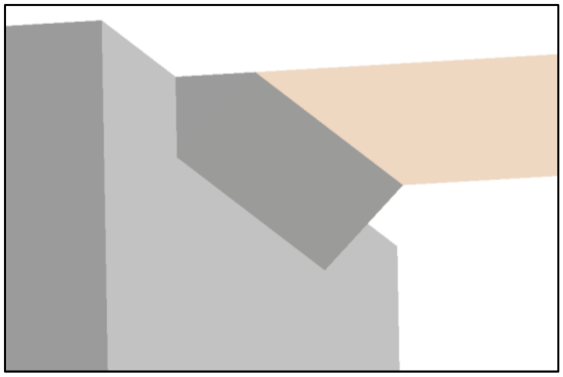


FIGURE 4 – SOLID BODY USED TO SUPPORT THE SHELL MESH

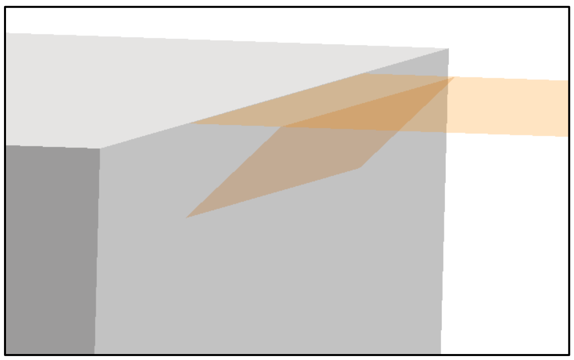


FIGURE 5 – SURFACE BODY USED TO SUPPORT THE SHELL MESH

Option 3 – Perpendicular Surface

Create a "T" connection with an additional perpendicular surface (Figure 6). The size and shape of the perpendicular surface should represent the cross section of the 3D geometry.

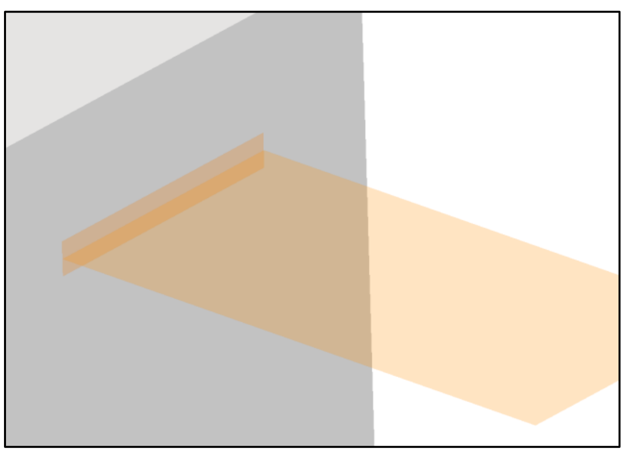


FIGURE 6 – "T" SURFACE ADDED PERPENDICULAR TO THE SHELL BODY

Create a new shell element idealization for the "T" surface. Make the thickness very small (e.g. 0.001") so that it doesn't add too much stiffness. Activate Continuous Meshing in the mesh settings to create a welded connection between the two surface bodies (Figure 7).

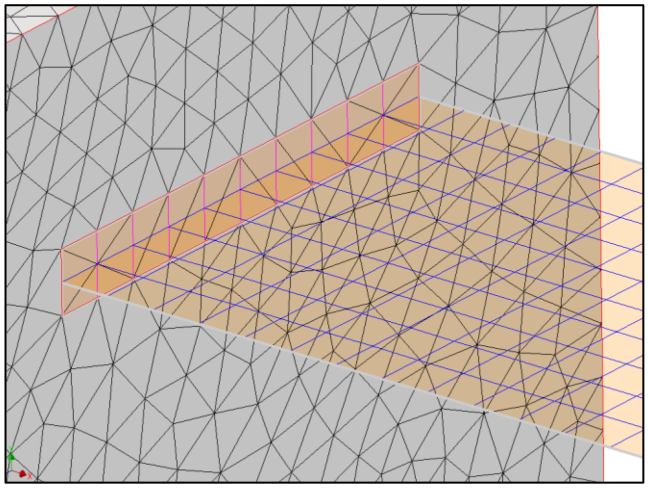


FIGURE 7 – CONTINOUS SHELL MESH APPLIED BETWEEN THE TWO SURFACE BODIES

This approach provides the most control over the size of the contact area and how it loads the solid element face. Split the solid face around the "T" to get a better mesh alignment.

All of these methods will yield similar results as shown in Figure 8. In general, option 2 and 3 are preferred because you have the most control over the contact area.

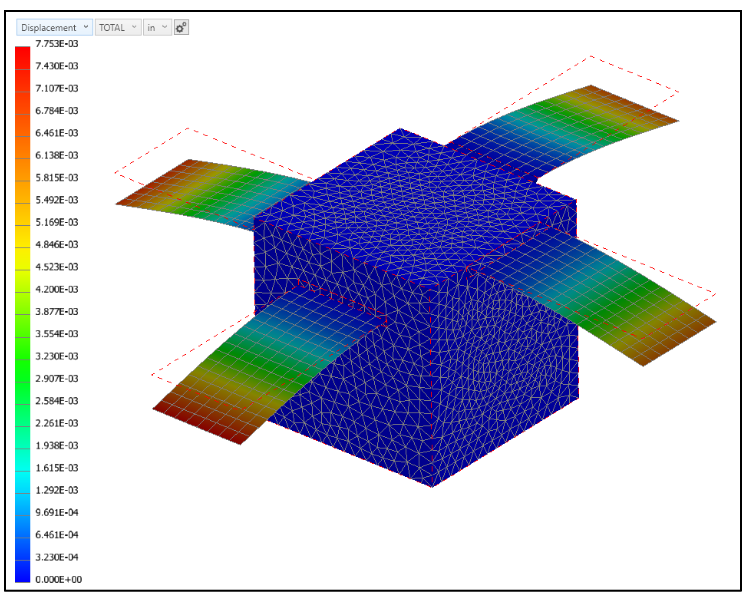


FIGURE 8 – RESULTS USING ALL THE DIFFERENT SHELL TO SOLID CONNECTION METHODS

Connecting Lines to Shells / Solids

Line elements can be connected with shell or solid elements in number of different ways. It greatly depends on the orientation of the components. Each scenario requires a different approach:

Scenario 1 – Parallel

As shown in Figure 9, the line element is running parallel to the face of the shell or solid element.

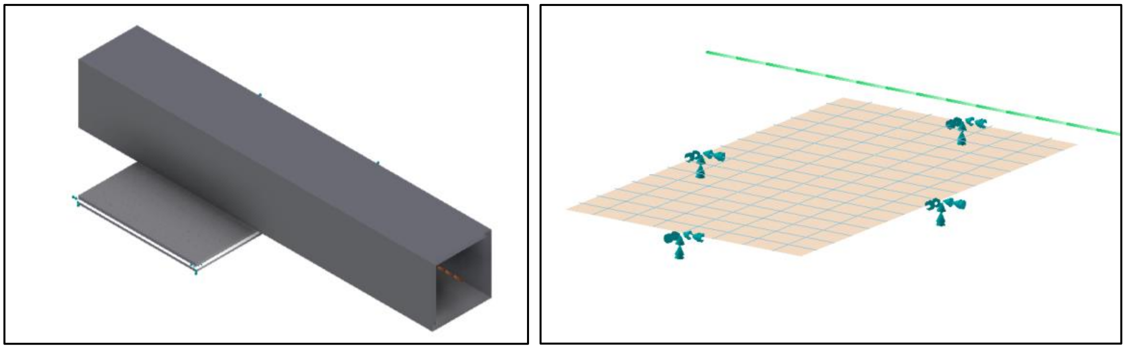


FIGURE 9 – PARALLEL ORIENTATION OF LINE ELEMENT RELATIVE TO THE SHELL ELEMENT FACE

To bond them together, apply an Edge-to-Surface Bonded contact. The primary entity will be the face of the shell / solid. The secondary entity will be the line element. In the contact settings, you will need to increase the Max Activation Distance to span the gap between the elements. Typically this value will be ½ the cross sectional thickness of the line element shape. The results of the stress analysis will look similar to Figure 10.

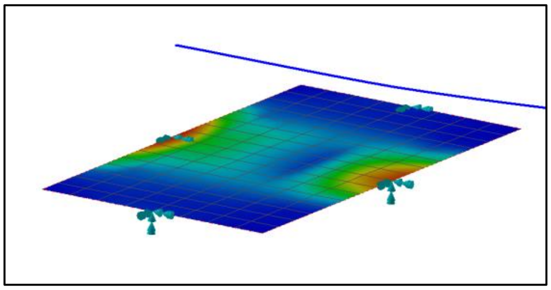


FIGURE 10 – STATIC STRESS ANALYSIS RESULT WITH LINE ELEMENT BONDED TO SHELL FACE

Scenario 2 – Axial

As shown in Figure 11, the line element is meeting the adjacent face in a perpendicular (or axial) orientation. This is fairly common in welded and fabricated structures. Surface contacts can't be applied to points, so a rigid body connector needs to be used.

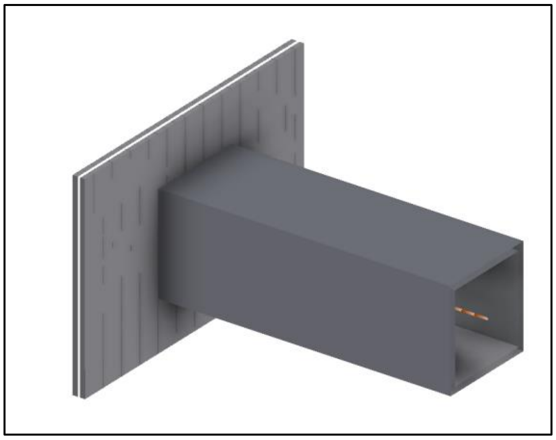


FIGURE 11 – PERPENDICULAR ORIENTATION OF LINE ELEMENT TO SHELL FACE

First, you'll need to split the surface (or solid) body with the silhouette of the line element cross section. As shown in Figure 12 – the split section should match the 2D cross section (e.g. Tube, Pipe, Ibeam, Channel).

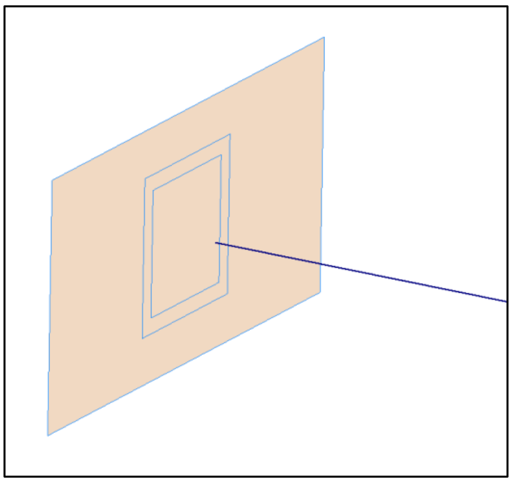


FIGURE 12 – SURFACE BODY SPLIT TO MATCH THE 2D CROSS SECTION OF THE LINE ELEMENT

Next, in Nastran, create a Rigid Body Connector (RBE2) between the idealizations. The "Dependent Entity" will be the split face of the surface. The "Independent Vertex / Point" will be the end point of the line element (see Figure 13).

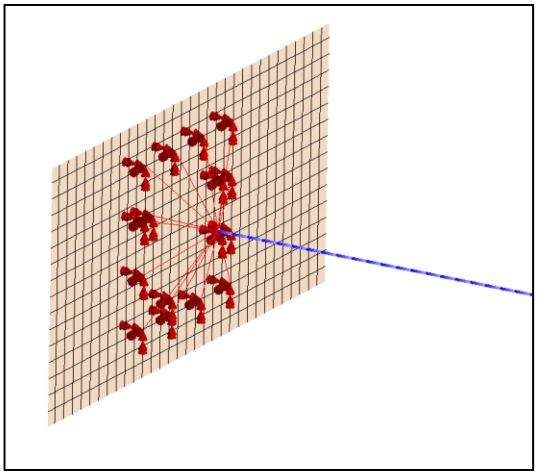


FIGURE 13 – RIGID BODY CONNECTOR BETWEEN SPLIT FACE AND LINE ELEMENT END POINT

There must be a small gap between the end point and the surface body. The dependent and independent entities of a Rigid Body Connector can't share nodes. After running a linear static stress study, the results will look similar to Figure 14.

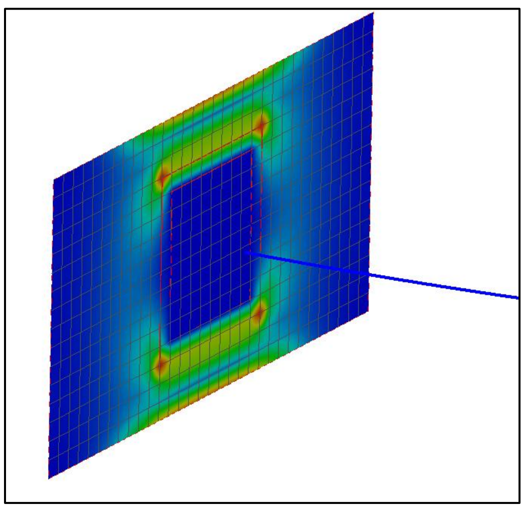


FIGURE 14 – STATIC STRESS RESULT WITH LINE ELEMENT ORIENTED PERPENDICULAR TO SHELL

Scenario 3 – Edge

As shown in Figure 15, the line element is running through the center of holes in the surface bodies. In this scenario, the pipe would be welded to the plates. Surface contacts can't be applied between two edges, so rigid body connectors need to be applied.

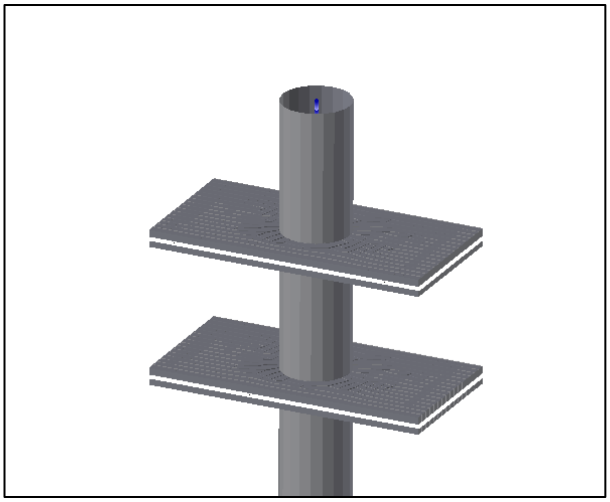


FIGURE 15 – LINE ELEMENT RUNNING THROUGH SHELL ELEMENT EDGES

First, you will need to split the line element where it passes through the hole. This will give the Rigid Body Connector something to attach to. This can be achieved by adding a "Split" to the sketch line as shown in Figure 16. Note – the line element idealization will need to updated after modifying the sketch to include both segments.

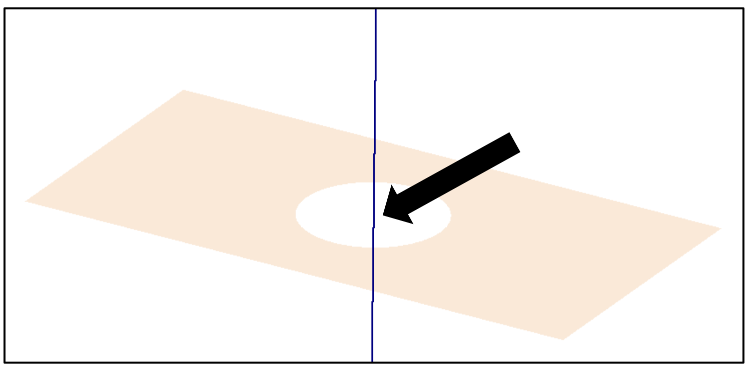


FIGURE 16 – SPLIT THE SKETCH LINE AT THE CENTER OF THE HOLE (SHOWN BY ARROW)

Next, in Nastran, create a Rigid Body Connector (RBE2) between the idealizations. The "Dependent Entity" will be the edge of the shell element hole. The "Independent Vertex / Point" will be the split point on the line element (see Figure 17).

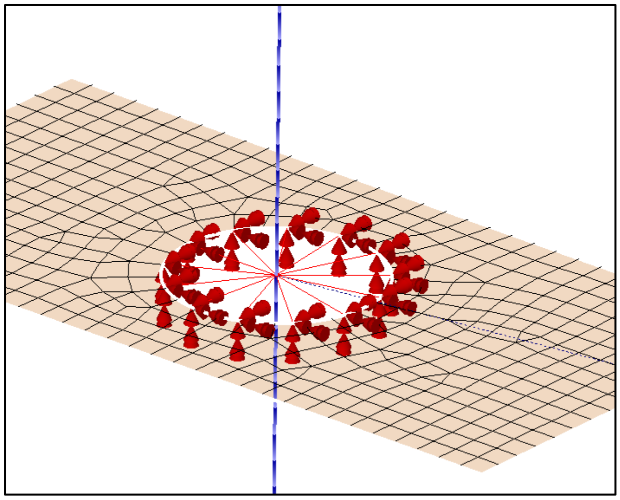


FIGURE 17- RIGID BODY CONNECTOR BETWEEN SKETCH POINT AND SHELL EDGE

The same approach can be used to connect the line element to a solid element face. After running a linear static stress study, the results should look similar to Figure 18.

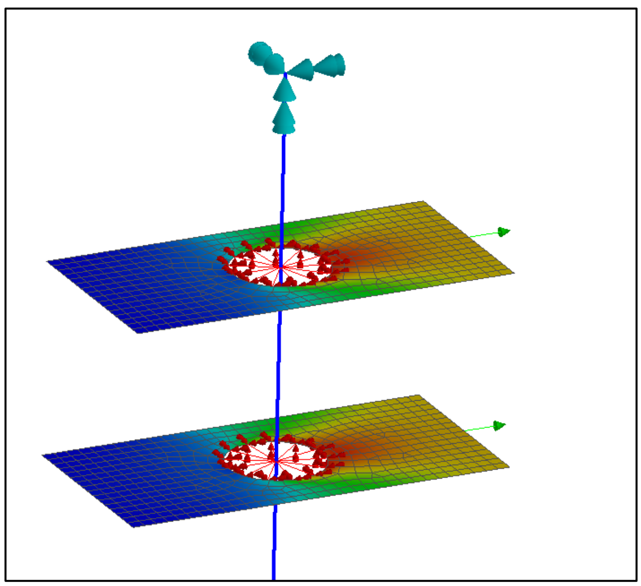


FIGURE 18 – STATIC STRESS RESULTS WITH LINE ELEMENT RUNNING THROUGH SHELL HOLE EDGES

Connecting Spring Connectors to Shells / Solids

By default spring connectors can only create stiffness between two points. Another approach is needed to transfer spring stiffness to a larger surface area (e.g. rubber isolators, spring mounts, etc). This method is basically the same as connecting line elements to shells & solids.

Start by splitting the surface (or solid) body with a silhouette matching the spring diameter. This will typically be the surface of the spring mounting plate or bracket.

After creating the spring connector with Nastran, add a rigid body connector from the end point of the spring to the split face. The "Dependent Entity" will be the split face on the shell or solid. The "Independent Vertex / Point will be the end point of the spring connector (Figure 19).

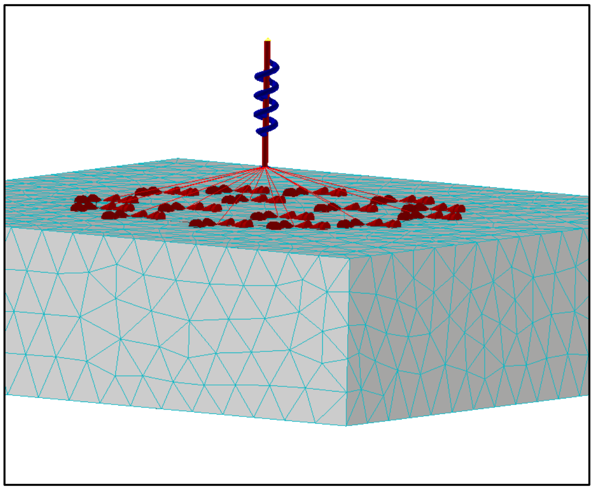


FIGURE 19 – SPRING CONNECTOR JOINED TO SOLID WITH A RIGID BODY CONNECTOR

Connecting Shells to Shells (Continuous Meshing)

Continuous meshing is the best way to connect shell elements to other shell elements in an assembly. In order to achieve a continuous shell mesh, the edges must be co-planar with the faces (Figure 20). I recommend creating the surface bodies using CAD before importing the assembly to Inventor Nastran. This allows for extending, trimming, and simplifying the shapes before generating the mesh.

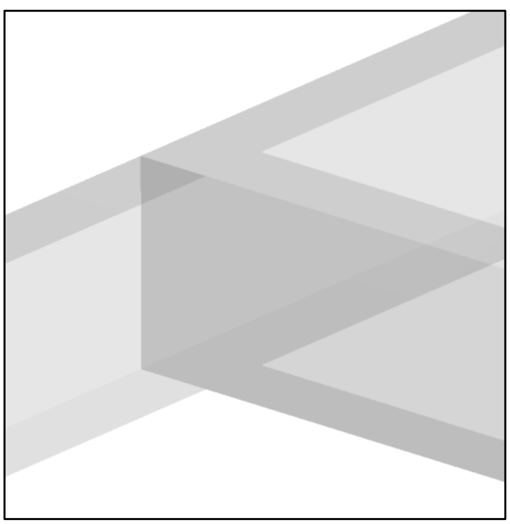


FIGURE 20 – SURFACE PARTS WITH CO-PLANAR EDGES

Once the CAD has been modified to create a continuous surface body, Inventor Nastran can be used to generate the shell element mesh. Make sure all the surfaces have a shell element idealization applied with the correct thickness. Then, in Mesh Settings – check the box next to "Continuous Meshing." Any shell elements that are co-planar will be aligned so they share nodes as seen in Figure 21.

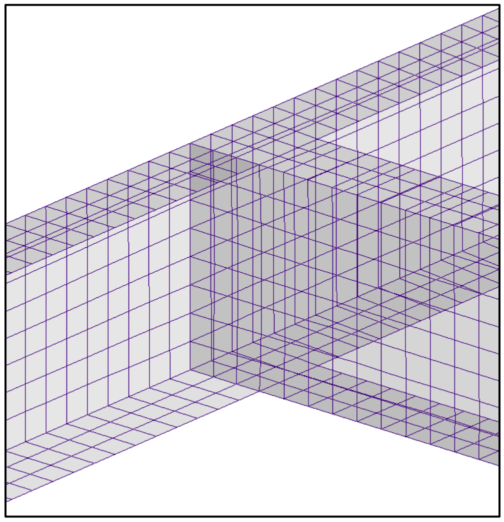


FIGURE 21 – CONTINUOUS MESHING OF SHELL ELEMENTS

A continuous shell mesh acts like a fully welded structure. Forces and Stresses will automatically be transferred from one surface to the next. No contacts are required! Contacts can greatly increase the analysis solve time, so this is a big benefit.

Simplifying CAD Geometry for FEA

Shell and line elements can be created directly from the 3D CAD bodies, but there will always be some level of simplification required. Either Inventor or Fusion 360 can be used to create an FEA-ready model.

Inventor Nastran Surface Tools

The surface tools native to Inventor Nastran (Figure 22) are fast and convenient. However, they struggle with more complicated shapes such as channel and angle.

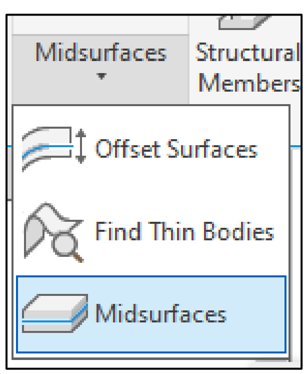


FIGURE 22 – INVENTOR NASTRAN SURFACE TOOLS

Once a surface is created it can't be modified. This can cause issues when trying to create a continuous mesh. Especially if midsurfaces are being used. You also can't split these surfaces for more accurate constraints, loads, and connectors.

For example, the frame shown in Figure 23 has two channel shapes welded to a square tube.

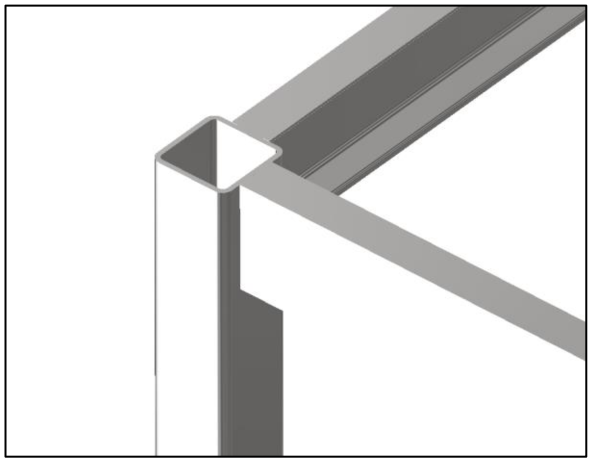


FIGURE 23 – FRAME WITH WELDED CHANNEL AND SQUARE TUBE SHAPES

The Nastran midsurface tool leaves gaps between the web and flange of the channel parts (Figure 24). This happens because the shape has an internal and external corner radius. The same issue happens with I-beams.

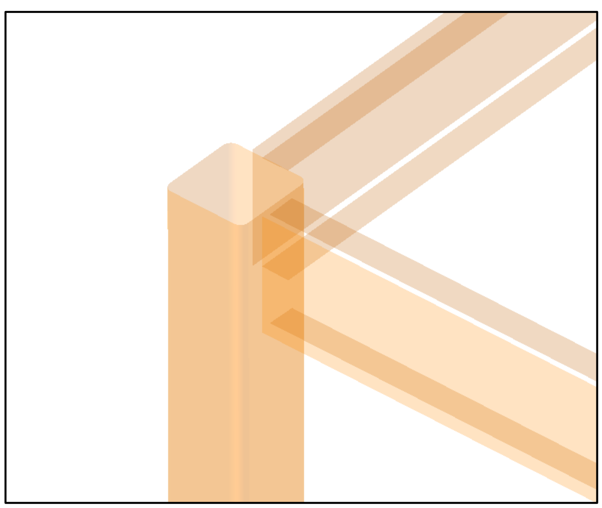


FIGURE 24 – FRAME CONVERTED TO SURFACES USING INVENTOR NASTRAN TOOLS

Inventor Simplification Tools

Surface bodies can be created in the Inventor modeling environment. Creating these surface models prior to FEA will make simplification and continuous meshing easy.

As you may already know – Inventor surface tools can't be accessed from the assembly environment. My recommendation is to derive your assembly into a part file. This can be done using Manage > Derive from a new part file (.ipt). Once the assembly has been derived into a part, the surface tools can be used to create midsurfaces. Extend and Trim can be leveraged to create co-planar surfaces for continuous meshing. Figure 25 shows an assembly that was derived into a part file and the simplified surface bodies.

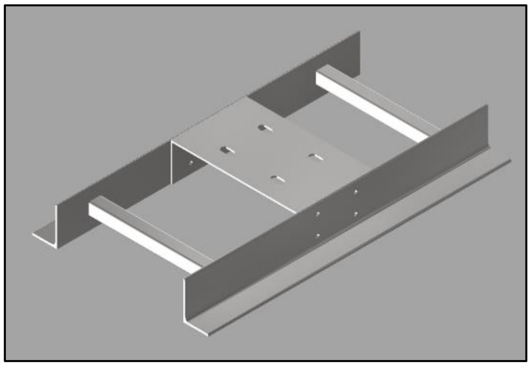
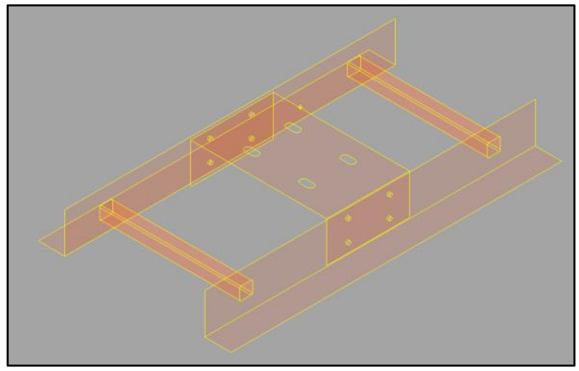



FIGURE 25 – A DERIVED PART FILE (LEFT) AND THE RESULTING SURFACE BODIES (RIGHT)

Fusion 360 Simplify Workspace

Fusion 360 offers another great tool for simplifying CAD assemblies into Nastran-ready geometries. It offers more simplification tools than Inventor. Fusion 360 is a direct-modeler – so assembly tools and surface tools are all available in the same file.

To access the "Simplify" workspace – first switch to the Simulation workspace using the drop-down menu in the top-left corner of the ribbon. When prompted, choose Linear Static Stress. From the Simulation workspace, click on "Simplify" from the top-ribbon. This will bring you to the workspace shown in Figure 26.

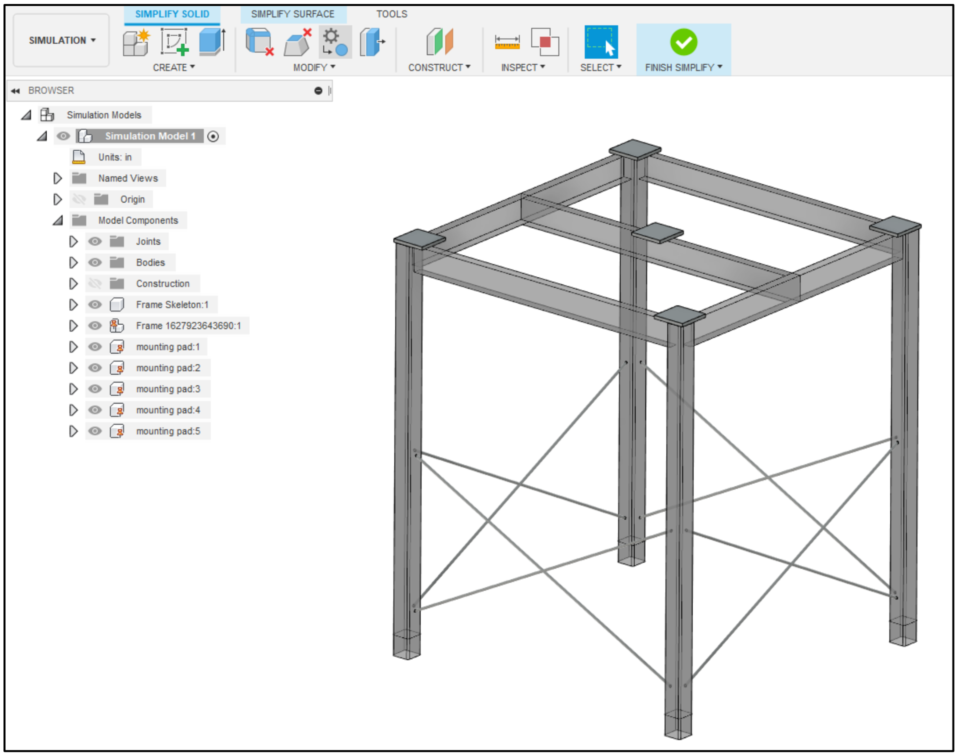


FIGURE 26 – FUSION 360 SIMULATION SIMPLIFY WORKSPACE

Any changes made to the model in the "Simplify" workspace will not be reflected on the original Fusion 360 model. The original will always be available in the "Design" workspace. This allows the user to create multiple simulation models for FEA, CFD, or other analysis. Use the Surface Offset tool to create midsurfaces. The Replace with Primitives tool is great for simplifying things like batteries and motors.

Right-Click on the "Simulation Model" at the top of the model browser and select "Export" to save the file to your computer as a .stp or .iges. Note - the exported model will only include bodies that are currently set to visible.

Importing .step Files to Inventor

If you're importing external CAD files into Inventor from Fusion 360 (or other software) make sure to treat the surfaces as "Individual." While navigating the import wizard – this will show up under "Part Options." As shown in Figure 27 – change this option from Composite to Individual using the drop-down menu.

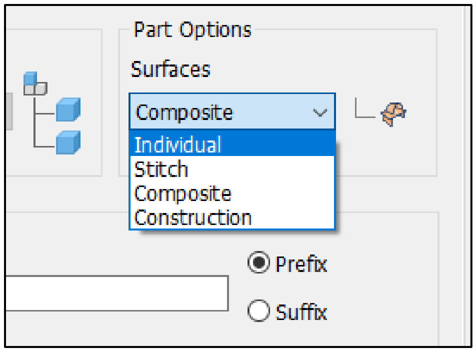


FIGURE 27 – INVENTOR SURFACE IMPORT OPTIONS

Importing surfaces as “Individual” allows for unique shell idealizations to be applied to each surface in Nastran.This is critical when working with I-beams that have a different web and flange thickness.