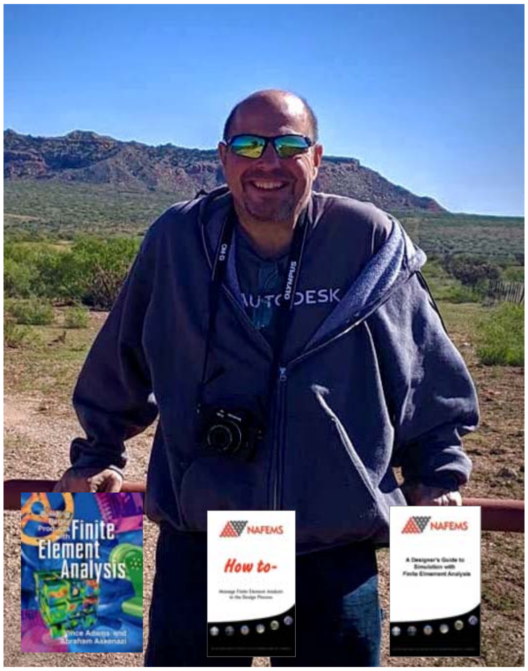
Using CAD Skills for Effective Design Simulation

MFG322193

Vince Adams
Account Specialist

Johnny Molica
Technical Sales Manager

AUTODESK.
UNIVERSITY



About the speaker

Vince Adams

- Career started in product design & engineering management
- Focused on Use, Training, and Support of Finite Element Analysis for 30 Years
- Active in Leadership of FEA User's Groups
- Inaugural Chairman; NAFEMS North America
- Co-Authored "Building Better Products with Finite Element Analysis" & authored 2 NAFEMS Books on FEA in Design
- International Speaker/Lecturer on FE Concepts
- Regular contributor on FEA related topics to Desktop Engineering Magazine, Design News & Ansys Soln's
- Recently selected as a Founding Member to the NAFEMS PSE (Professional Simulation Engineer) Certification Program

ENERGY | POSITIVITY | CREATIVITY

Johnny Molica
Technical Sales Manager
Cleveland, OH
440-867-1332
John.molica@autodesk.com

About the speaker

Johnny Molica

Education

- BFA, Cleveland Institute of Art

Industry Experience

- Product design with Hunter Design Group, product development for room conditioners and lighting

Partner Experience

- Application engineer & technical pre & post sales manager with SolidWorks Reseller in northeast Ohio
- Dassault Systems PLM/PDM consulting and implementation with Razorleaf Corporation

Autodesk Experience

- Territory Technical Specialist
- Technical Sales Manager

The Product Development Process…

Ask MANY questions

Ask the RIGHT questions

Get answers QUICKLY

Know what the answers MEAN

ACT on this information

SIMULATION CAN ONLY SUPPORT THIS IF YOU CAN GET YOUR ANSWERS MORE QUICKLY THAN OTHER MEANS

What is Finite Element Analysis?

- Basic Engineering Equations Describe Stress

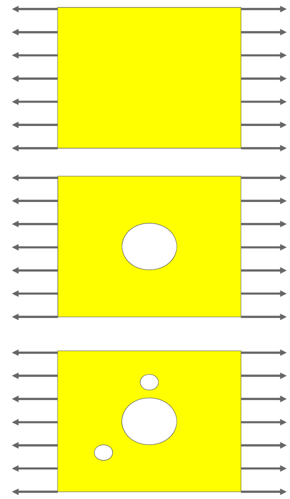
  $\sigma = F / A$

- Supplement Basic Engr Equations with Stress Concentration Factors from Structures Text (Roark) to Determine Peak Stress

  $\sigma = k_t * F / A$

- Engineering Equations No Longer Sufficient to Determine Peak Stress – FEA Best Choice

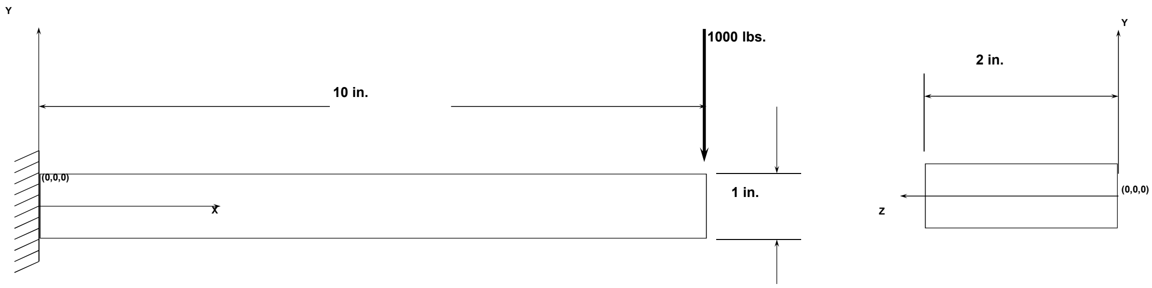
  $\sigma = ??$



Using CAD geometry to drive simulation has allowed engineers in all industries to ask more complex questions and get answers faster than they ever could

It isn't difficult to see how quickly even simple geometries can exceed our ability to use empirical equations.

What is Finite Element Analysis?



To calculate response by hand, use this information:

- Thickness
- Length
- Width
- Force
- End Condition
- Material Modulus

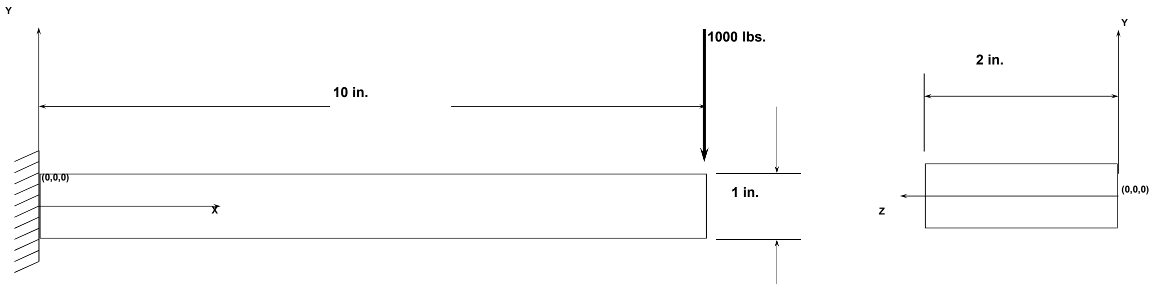
What else?

Even when the geometry seems simple, factors such as tolerances on both parts and assembly relationships can add ever-expanding levels of complexity.

Consider all the permutations of tolerance combinations a single part has. To be sure you've captured all the relevant variations, you must ask many, many questions.

Does your CAD geometry support studies of all the permutations?

What is Finite Element Analysis?



To calculate response by hand, use this information:

- Thickness
- Length
- Width
- Force
- End Condition
- Material Modulus

What else?

- Displacement Small
- Stress-Strain Linear
- Force Doesn't Change Magnitude, Orientation, Distribution
- Gravity Negligible
- Guy who Wrote Equation Got It Right!

Because there are more variables, even for a single use-case, beyond geometry that need to be explored.

What does this all mean?

All FEA Solutions are wrong…

Usually the Answers are right, but the Questions are wrong

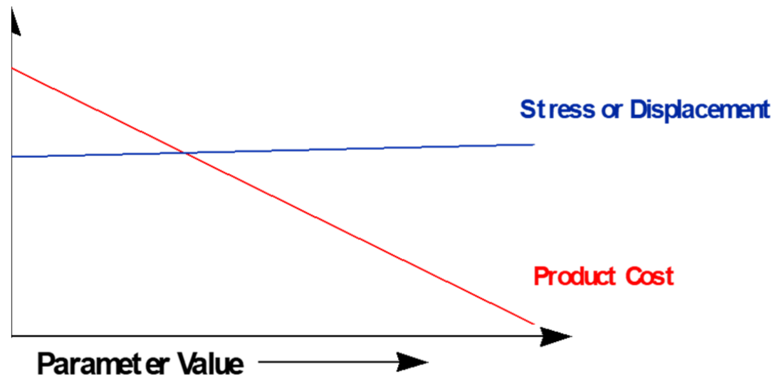
Since any model can only represent a single combination of variables, it shouldn't be considered the end-all for a given problem. The concept of "accurate" is over-rated. No matter how much time you put into the Inventor model in an attempt to represent your design's vision, it still only captures one combination of tolerances and manufacturing variations.

So what are we getting at? Painstaking CAD work on a design before you've proven it is up to the task doesn't make the analysis any more accurate. It only makes the time to solve a given iteration and the effort to improve or optimize longer.

Starting (or in many cases stopping) the Inventor modeling with the most appropriate and minimal CAD to answer the questions, "Will it work?" and "Is it best?" will yield the most effective collaboration between design and simulation.

Absolute Data vs. Trend Analysis

- Actual Values aren’t needed to see the benefit in making changes...
- Assumptions must be reasonable



For these reasons, the most effective use of simulation in the general design process is to focus on trends. If I change a parameter or dimension, does an operating response go up or down? By a lot or a little? Can I measurably reduce product cost without giving away anything in strength or stiffness?

If the answer is Yes, the precision or accuracy of the model isn’t as important as the consistency of the compared studies… within reason. Your inputs need to be representative so that the changes reflect real world changes. Beyond that, you have a lot of lee way to simplify in pursuit of insight.

What is Finite Element Analysis?

IN THE SIMPLEST TERMS, FEA ANSWERS COMPLEX QUESTIONS ABOUT STRUCTURAL BEHAVIOR...

Based on a Given Set of Assumptions!

Key Assumptions…

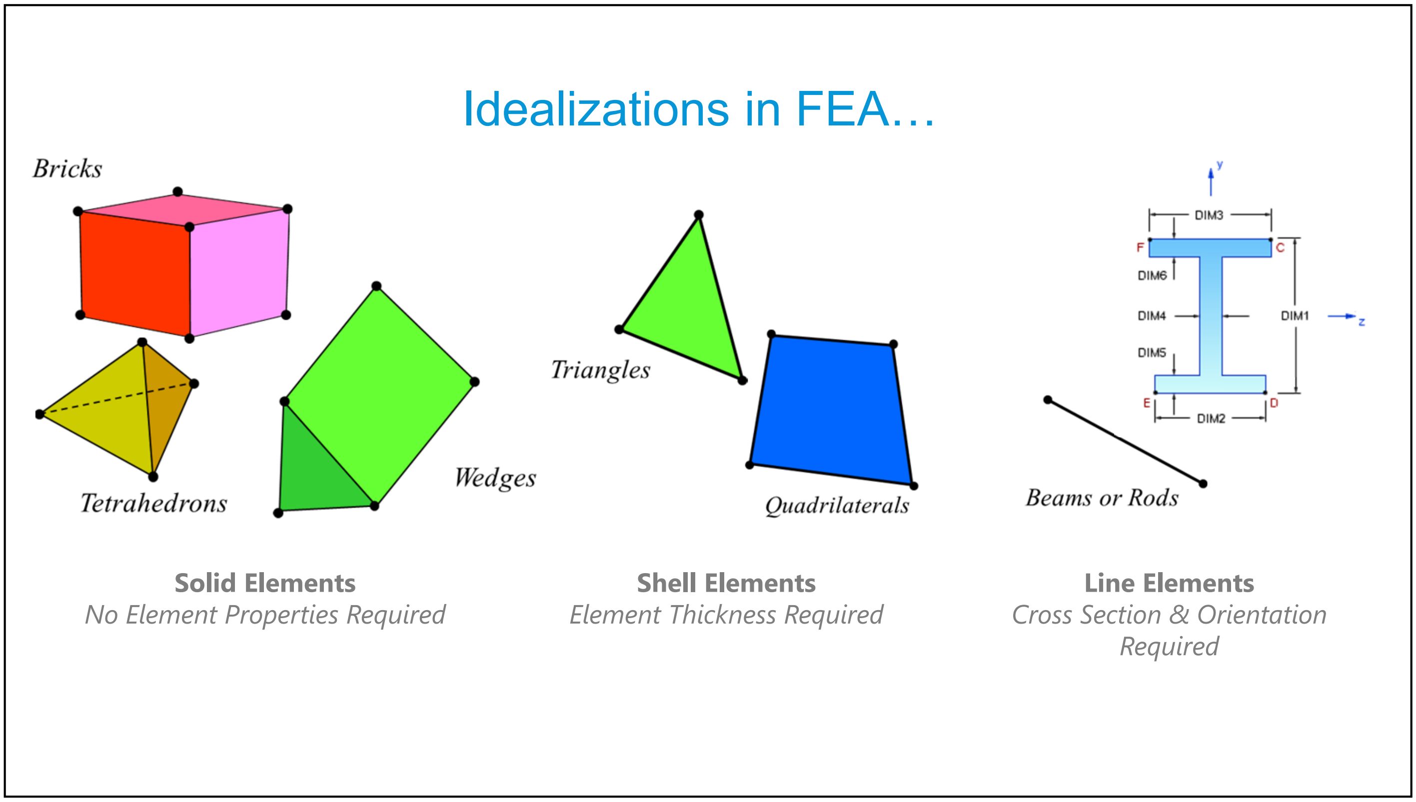
- **Idealizations**
- Properties
  - Element
  - Material
- Geometry
- Mathematical Representation (Mesh)
- Interactions (Boundary Conditions)
- Appropriate Physics

Idealizations in FEA…

- After 30+ years of FEA usage, techniques for reducing the physical, or dimensional, complexity of a structure have been developed… **Idealizations**
- Choosing the Right Idealization Will Help Improve…
  - Accuracy
  - Speed
  - Opportunity to Optimize

How Does Geometry Support Idealizations?

- FE solvers have no concept of geometry – Only nodes & elements
  - CAD Geometry provides a template for the mesh
- Building clean and appropriate geometry helps ALL downstream applications
- Dictates ease of optimizing designs early in the design stage
- If geometry hinders analysis speed, fewer iterations (=questions) will be completed
- Start with the geometry that enables best idealization



There are two types of properties you need to consider in a Finite Element Model:

1) Material Properties
2) Element Properties

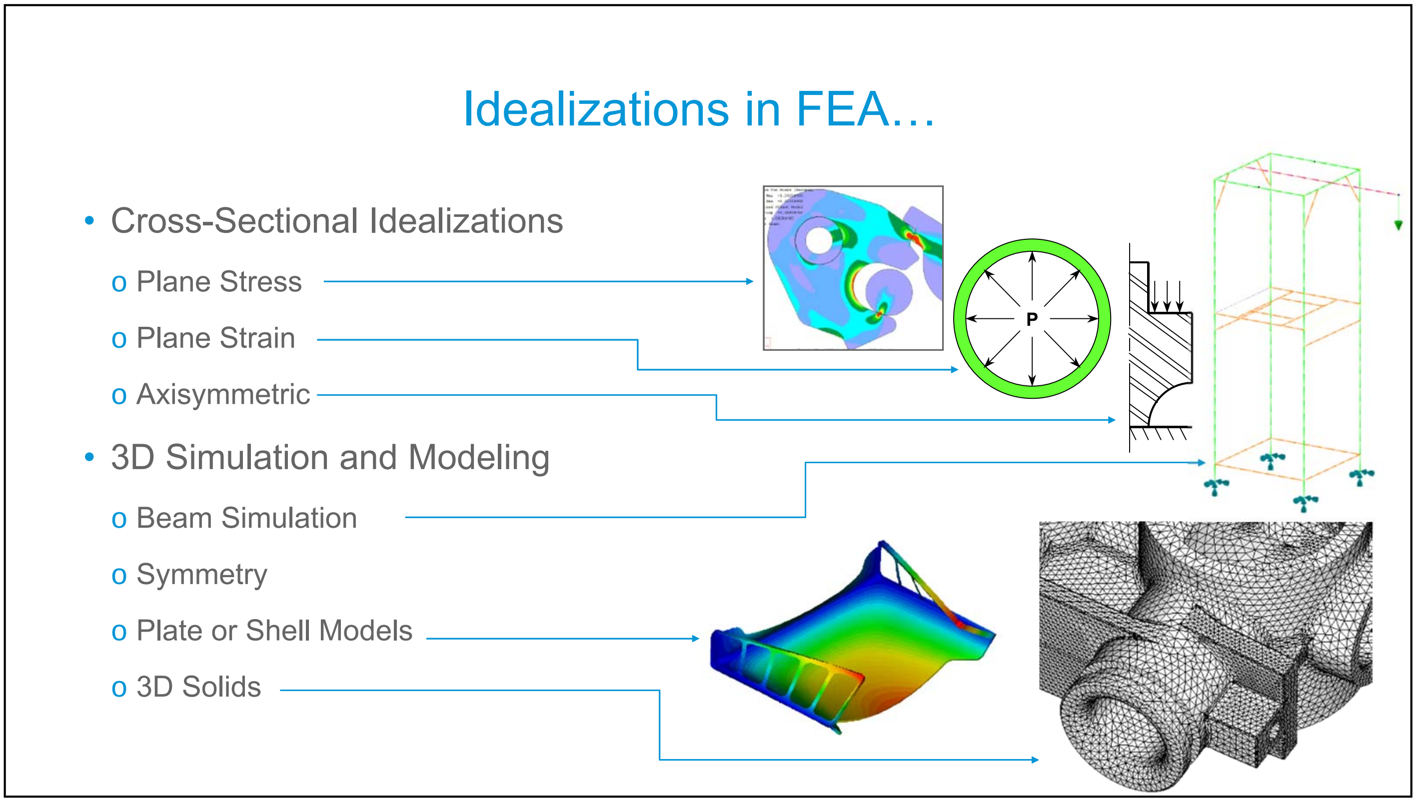
Element properties are the mathematical representation of a geometric simplification.

Solid elements fill a volume in 3D space. In general practice, they don't require element properties since the geometry is fully defined.

Shell elements are surface representations of thin-walled solids. The thickness dimension is replaced with a numeric property. A single element can replace dozens of solid elements for improved speed. They capture bending better than most solid meshes and changes to a thickness don't require CAD re-work. You simply type in a new thickness and Run.

Beam elements are essentially wireframe representations of long, slender bodies. Two dimensions are removed so the cross-section is called out with properties.

Shells and beams have MANY more properties to consider if you find them useful in your modeling. We will only touch on a few in this class but you are encouraged to speak to your Autodesk or Partner specialists to learn how they may improve your understanding of your products.



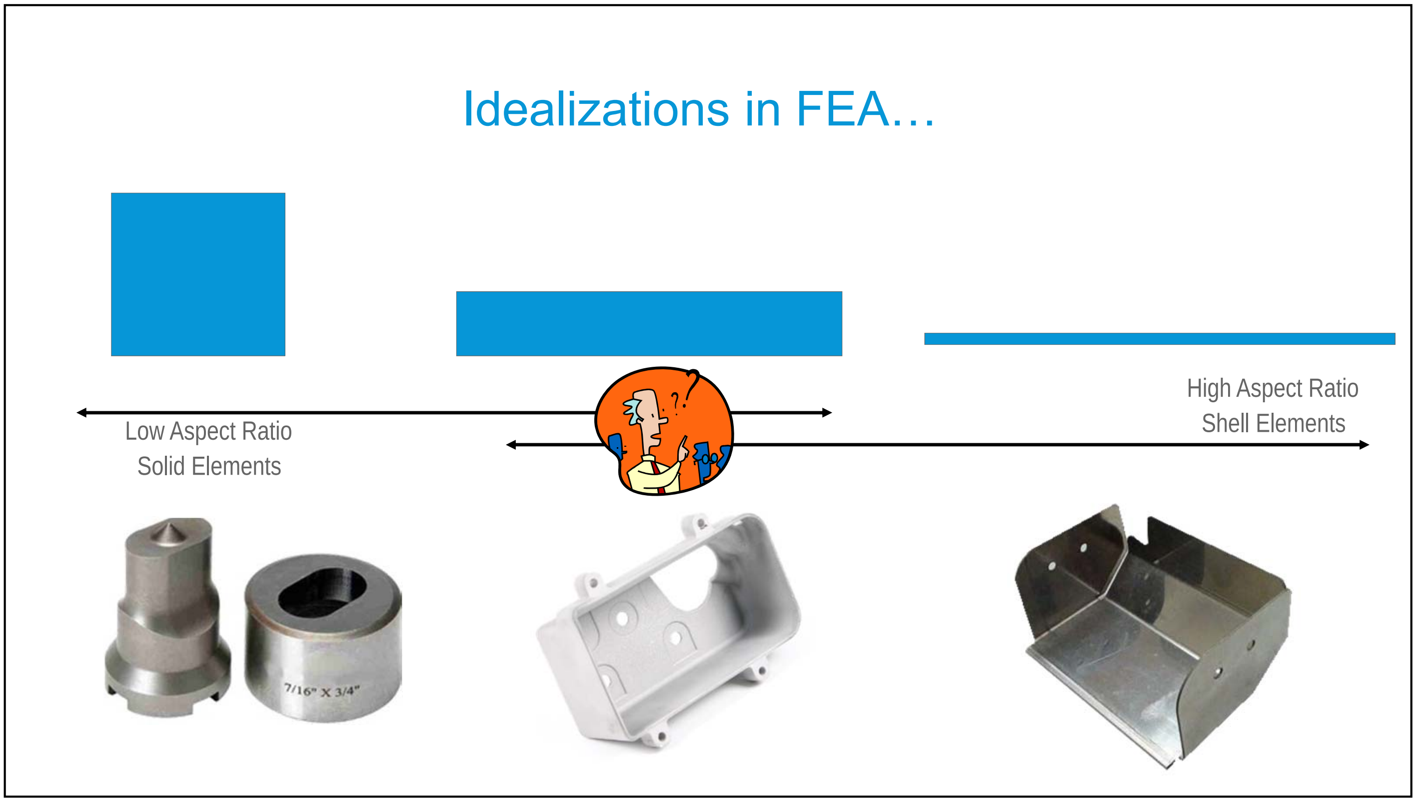
In addition to solids, beams, and shells, you can consider cross-sectional idealizations that let you examine starting shapes or even simpler representations in pursuit of fast answers for better design decisions.

Symmetry is another form of idealization that can be used in conjunction with the others. Remember that both the boundary conditions AND the geometry need to be symmetric, ore reflected about a plane, to use symmetry.

Idealizations in FEA…

RULE OF THUMB…

- Fit CAD model to screen
  - Do long slender members look like lines?
  - Do thin walled features look like surfaces?
- The closer the idealization resembles the CAD model, the more valid the idealization is

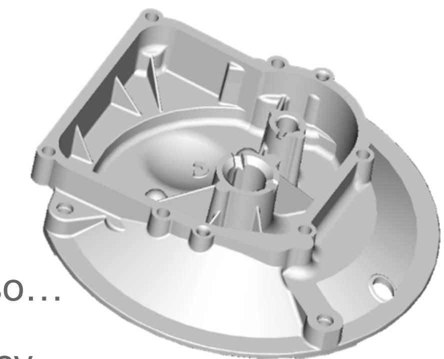


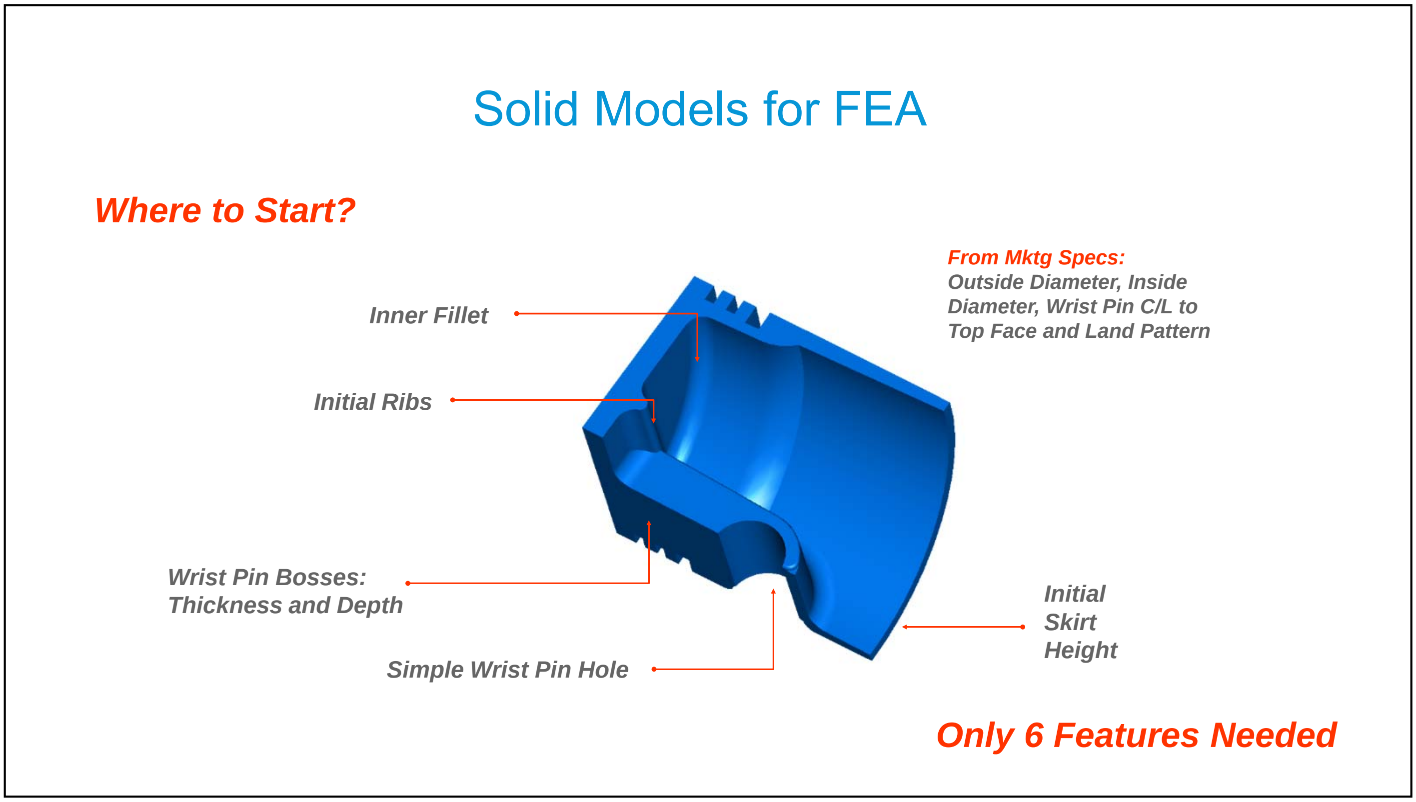
The challenge, art… if you will, of successful idealization use lies in the gray area between the obvious end cases. You need to use test models and solid engineering judgement if the idealization isn't obvious, as in the middle example. When in doubt, use solids but check convergence. You may only know you can use shells or beams AFTER you've modeled with solids first and examined the parts behavior.

Solid Modeling

Solid Models for FEA

- Again…start with the geometry needed for FEA
- Avoid dirty geometry
- Avoid fragile parent-child relationships
- Build insignificant features last
- Remember… FEA has no concept of geometry so…
- Elaborate CAD models do not guarantee accuracy



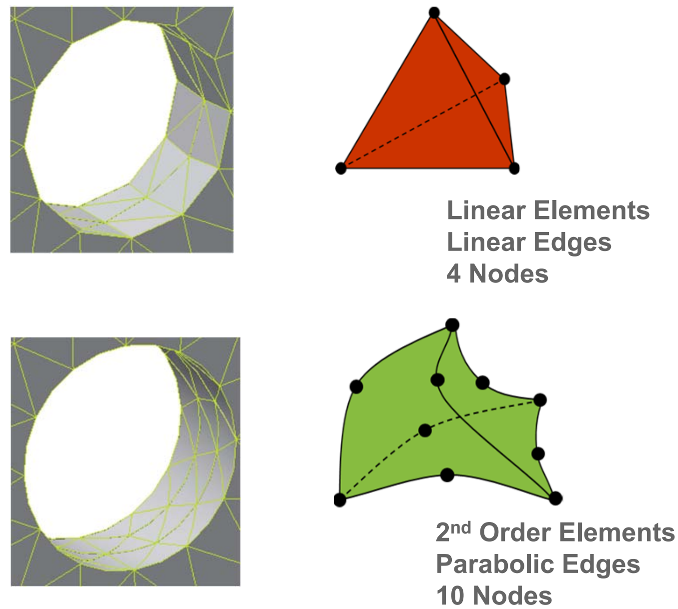


In this real-world example, a piston designer realized that he could validate the acceptability of a piston with 6 simple features that captured the key responses he tested for. There were several dozen fillets, chamfers and other features need for the manufactured version but they could take the better part of a day to add and about ½ that time to remove if the piston wasn't where it needed to be. STOPPING and checking before adding features that add no value is hard to do but as a habit, can save you hours to days in a project.

This engineer needed to take a step back and ask, "Is there a better way?" to find it but I've encountered plenty of scenarios in many industries where this question should be asked. Challenge the status quo.

What About Small Features?

- Small features (compared to typical feature size) may:
  - Overload model unnecessarily
  - Represent the most critical parts of the model
- DON'T indiscriminately suppress or delete them
- If you leave them in, mesh them adequately
- Since deformation drives stress; consider curved element edges



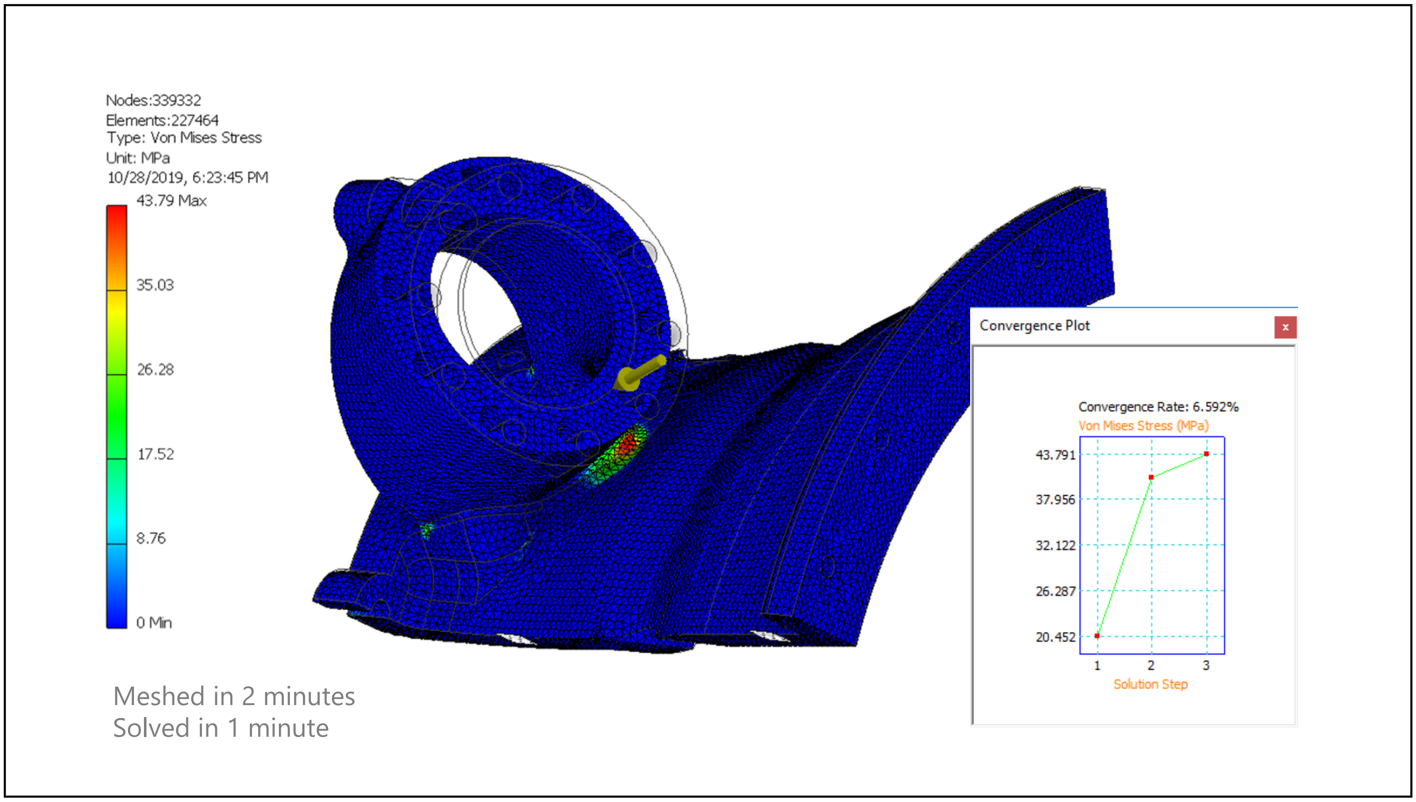
I often hear FEA coaches advocating the blanket removal of small features. If you were predisposed to hearing it, you might have thought that's what I was saying earlier. There's an important distinction between small and irrelevant. I advocate leaving out (don't put them in so you feel the need to suppress them) irrelevant features. If they add model complexity without any value to the end results, they likely don't need to be there. However, many small features NEED to be there to accurately capture the response of interest. Think fillets in an inside corner. Without the fillet, the stress could go singular and you'll never know what you should have expected. Think about value, not size.

Remember that FEA doesn't know what a hole or a fillet is. It only knows where elements are and how their connected. Consequently, for all the geometry you leave in, make sure the mesh accurately captures it. This means small enough elements to capture curvature and, possibly the use of curved edged/faces in a parabolic element so the initial math brings the starting point closer to the geometry you are exploring.

Solid model example

001201.ipt

Symmetry

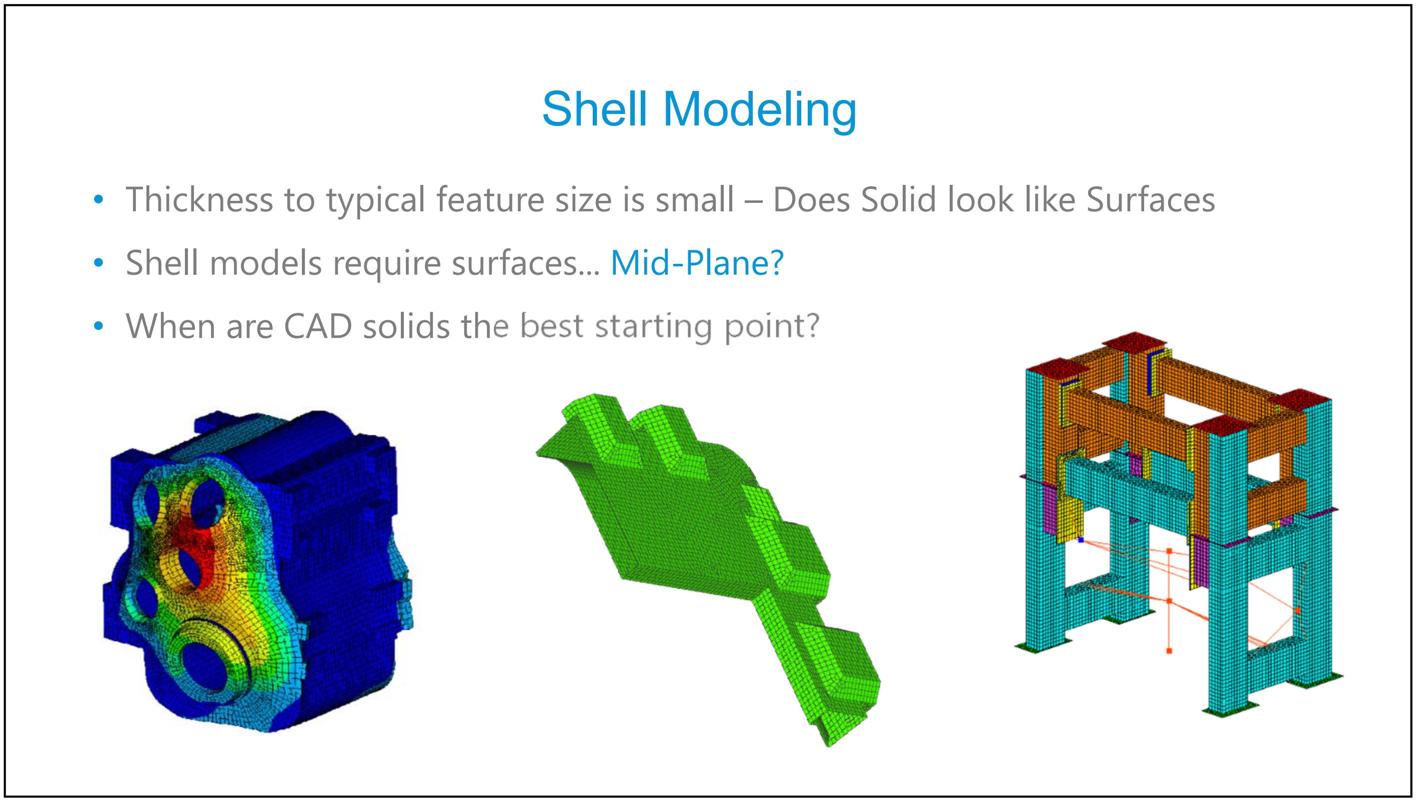


Inventor Stress Analysis is a fantastic tool for fast meshing and solving of solid models. Full transparency, we had intended to use this example to illustrate why you should be judicious with irrelevant features but it meshed and solved so fast, in a linear static solution, that our objections would have felt forced.

Some important points we'll still use this model to illustrate include:

- Physics: This solved fast as Linear Static. If you needed nonlinear material properties or sliding contact, you wouldn't be so lucky. A MUCH simpler model would allow you to explore multiple design iterations.
- Correction or optimization: If the initial structural concept wasn't sufficient, making changes to this model would likely take longer than the simulation. Keep in mind that the duration of an iteration will encompass both the remodeling and the solution. You want the whole process to go quickly so you don't short cut the optimization process.
- If this was part of an assembly analysis, the time savings of a simpler, more prismatic model would pay off as well.

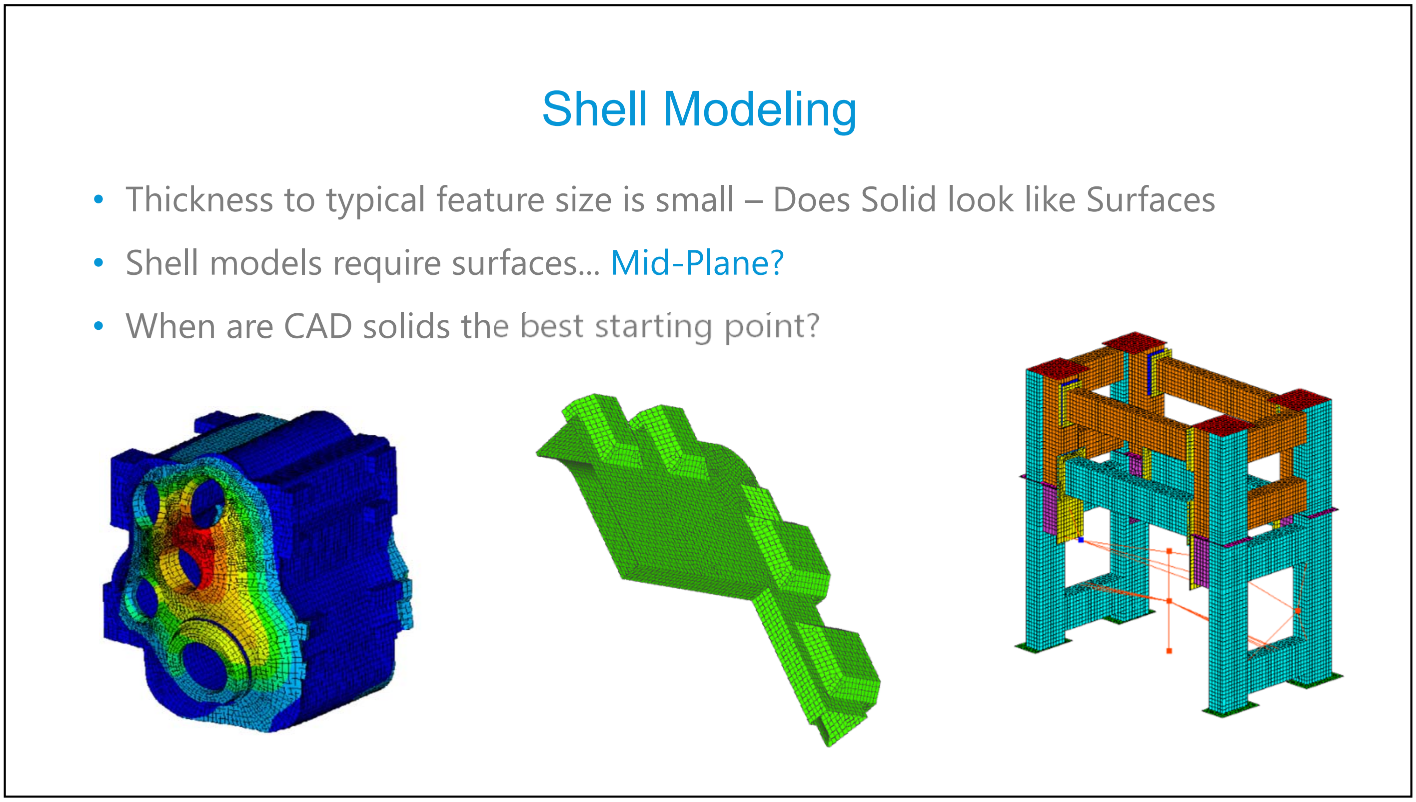
Shell Modeling



Shell elements geometrically surface elements. To use them, you need to have a face or surface placed to represent the thin-walled part. Inventor Nastran has made it incredibly easy to idealize sheet metal parts and assemblies as shell models. However, there are may other geometries that don't fit that recipe.

In the first case, a cast aluminum transmission housing is essentially thin-walled and a good candidate for shells. The faces on the solid weren't easily mapped to the appropriate shell face so this model was built with a combination of solid faces and for-purpose surfaces. This model would reside in parallel to the design model or as a pre-cursor. Solids might be used for a single linear static study but this assembly had 12 load cases and dynamic requirements.

The second example represents a roto-molded lid to a chemical tank. The design model was a shelled solid but mid-surface extraction wasn't 100% reliable with all the features needed. The ideal solution was to shell it to ½ the intended wall thickness which placed the inner CAD surface at the mid-surface of the intended part. With this technique, the design model could be retained for the simulation and all changes for optimization were immediately reflected in both use cases.



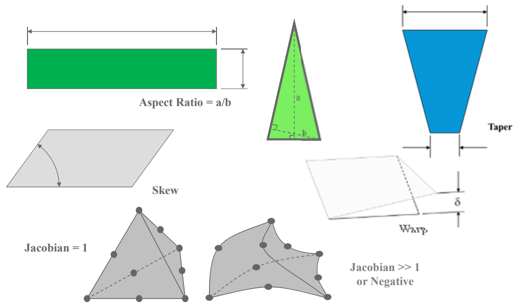
The third example appears to be a structural frame members but their depth exceeded standards. The beams were essentially big shells. The design model didn’t lend itself for midsurface extraction so the best way to explore functionality and options was a for-purpose analysis model, designed so all the edges and faces lined up.

These are extremes in the greater scheme of shell modeling needs. However, if examples like this are your business, it’s important to know the best place to start.

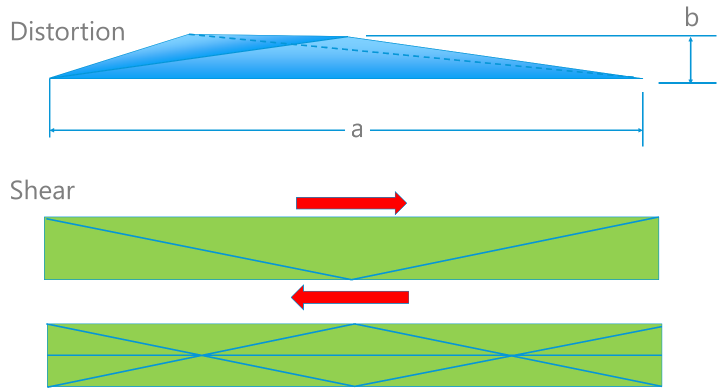
Going forward, we’ll focus on the most common cause for shell modeling, sheet metal assemblies.

Shell Modeling

- Primary benefits of Shells where appropriate:
  - *Meshing and Solving Speed*
  - *Hard Drive Space*
  - *Solution Accuracy*



Element Distortion Decreases Accuracy



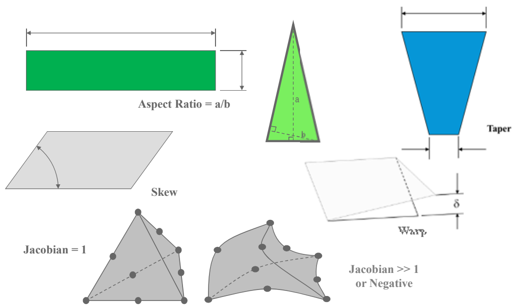
Why are shell elements preferred over solid elements?

The easy answer is speed. Many models can be solved with an order of magnitude fewer shells than solids and faster models means more iterations, more questions asked and more answers to build your insight.

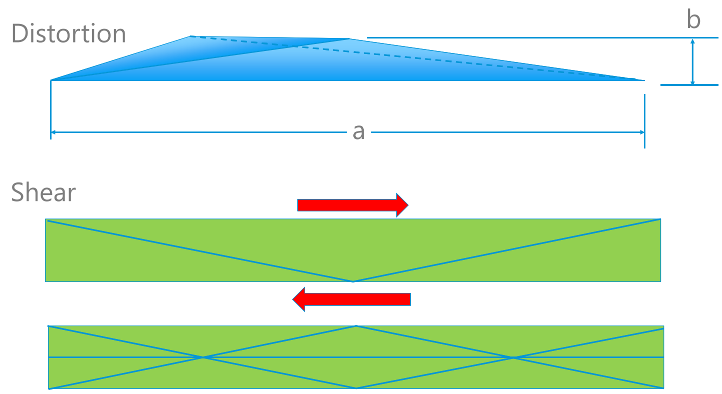
However, solution validity plays a big part too. Since this class is focused on geometry, not on meshing per se, it will suffice to say that the math behind an element's response is most accurate when triangular faces are equilateral triangles and rectangular faces are squares. As a face deviates from ideal, error gets introduced. Now consider a what a tetrahedral element squished into a flat wall looks like. The technical term for these elements is "pancake tets." None of the faces into the wall are close to equilateral so the element is unnaturally stiff, computationally. You'll see the impact of this in the example. One could use elements with edges on the order of the wall thickness but the model size for most real-world problems would quickly outpace your computer's ability to solve or your patience to wait.

Shell Modeling

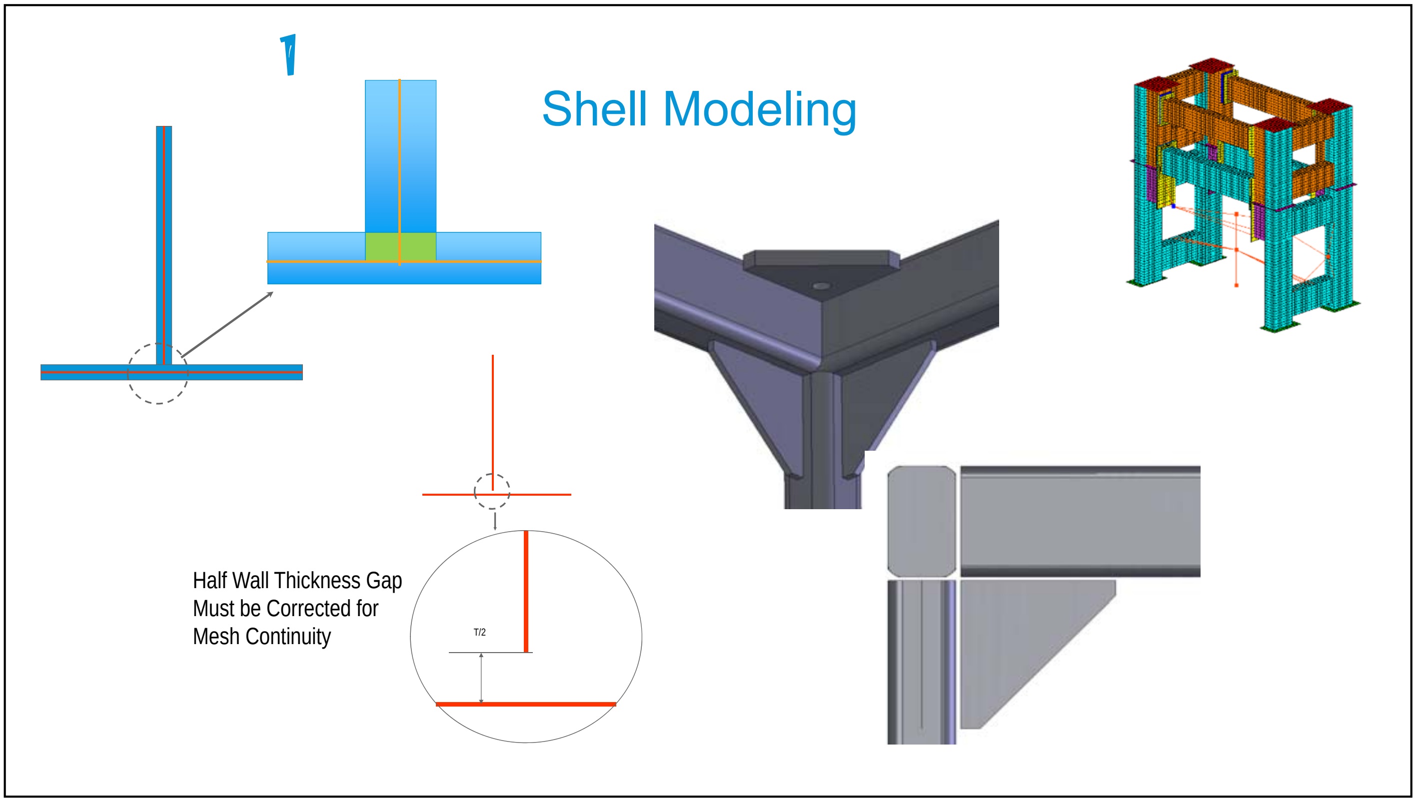
- Primary benefits of Shells where appropriate:
  - *Meshing and Solving Speed*
  - *Hard Drive Space*
  - *Solution Accuracy*



Element Distortion Decreases Accuracy



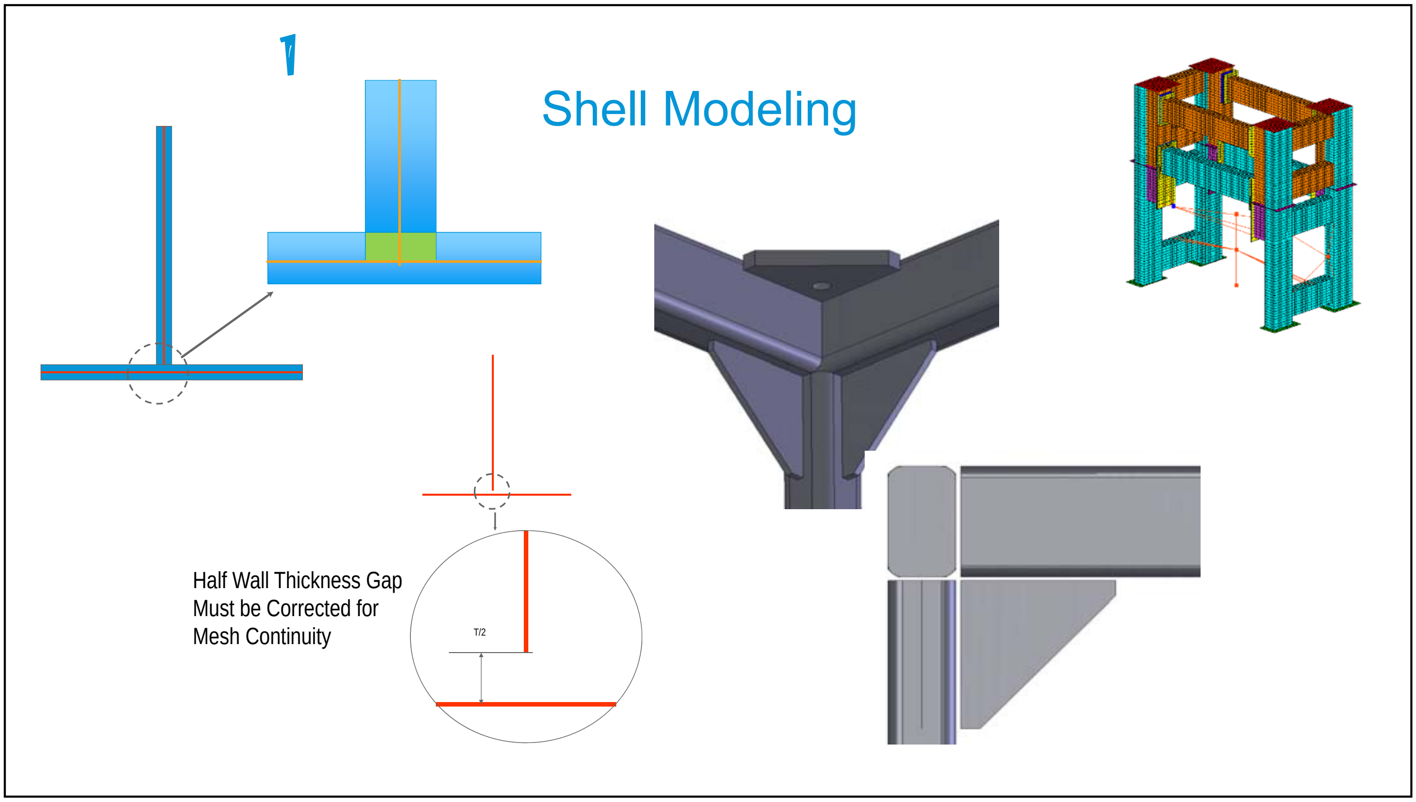
Even if you were able to get a model with decently shaped elements to solve, you'd still likely only have one element through the thickness of the wall. This can cause a problem computing shear in bending. I've spoken to many engineers who have seen good results come from models with one tet through a wall thickness. If your response is predominantly in-plane, you'd expect to see that. However, the more bending that occurs, the more shear stiffening will impact your results. Unless you are prepared to investigate this difficult to measure difference, use shells where they are appropriate.



One important characteristic about idealizing thin-walled solids to representative surfaces in FEA is that it is most convenient, analytically, to place the surfaces at the mid-plane or mid-surface of the original solid. When placed as such, gaps between parts occur. Compensating for those gaps historically comprises the bulk of shell modeling in practice. As you'll see in the example, Inventor Nastran has the best tools for these types of models. However, it is important to understand what's happening at these joints.

I'm often asked if joints in shell assemblies are accurate. What about welds? Well, the absence of a weld bead in the model when there is one in real life should answer the first question. If the parts are welded, the stress in the joint of the model will be difficult to predict. On top of the earlier admonition that all FEA results are wrong… inaccurate… joints of two shell surfaces have some added error because of what the shell elements represent.

Looking at the figure marked "1", you can see the orange lines represent the shell element placement while the blue areas represent the intended walls. When you bring the shells in contact for load continuity, there is an unintended overlap in volumes, indicated by the green area. The software essentially assumes there's more material than there should be, which obviously will impact the quality of the LOCAL results. How much? Too much? This is case dependent but the better suited your model is for shells (i.e. BIG faces and THIN walls,) the less it will matter.

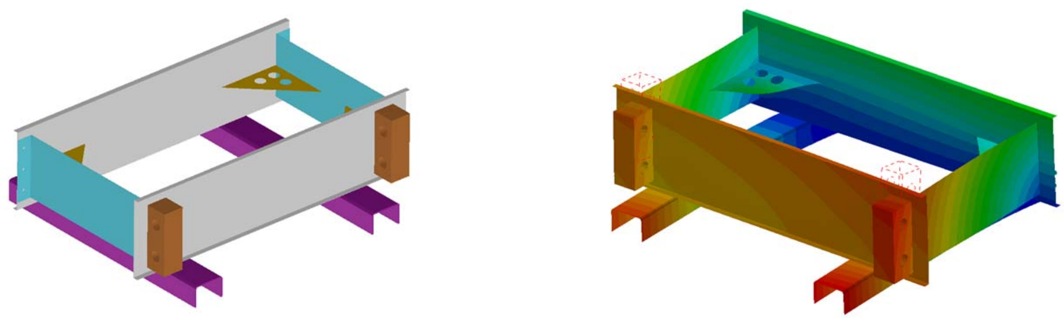


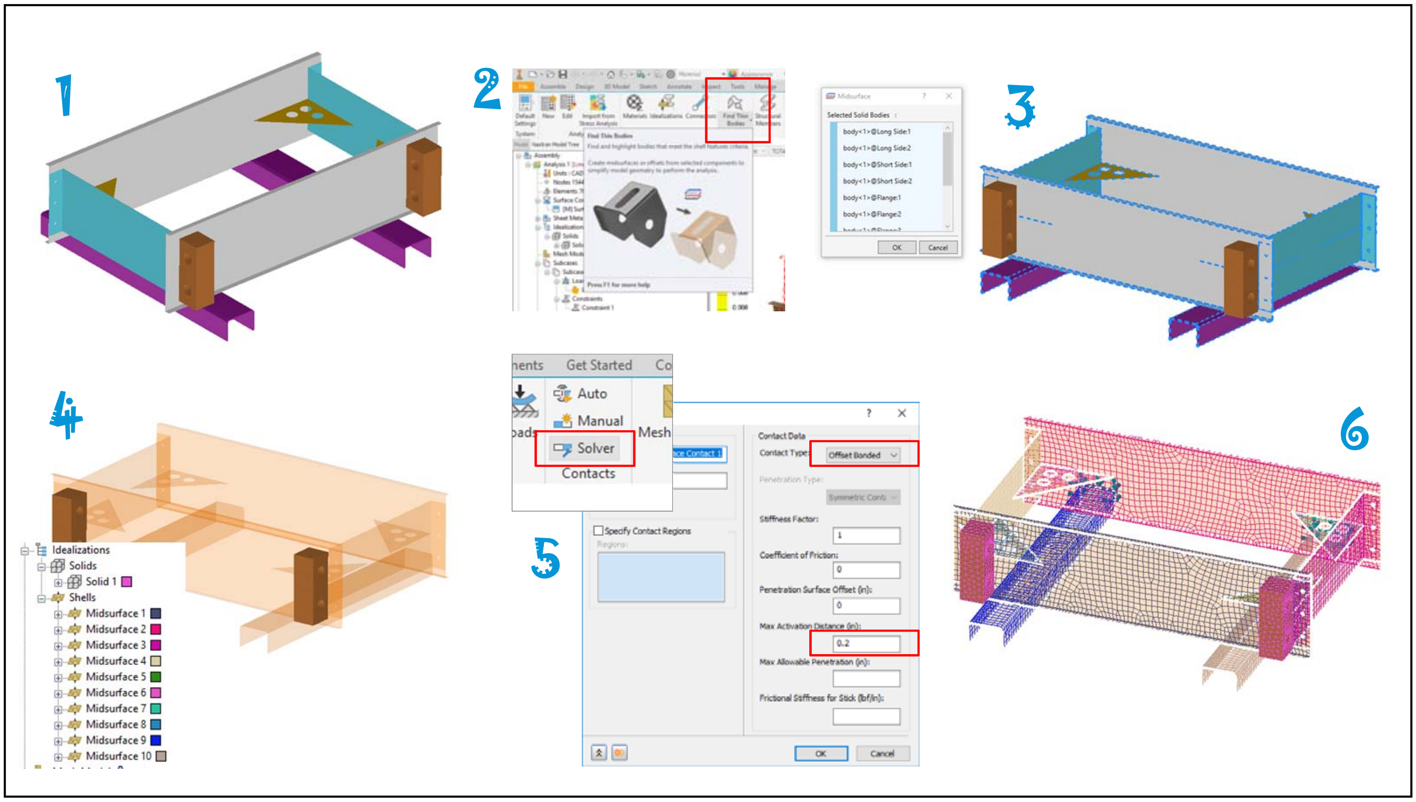
Another method of bridging the gaps from mid-surfaced solids is thru what can best be described as mathematical "glue." More technically speaking, you can use stiff springs or rigid elements to connect the components across the gaps. This is the preferred method in Inventor Nastran. It eliminated the "added material" concern but does introduce it's own error into the local results. The mass computation for dynamics will benefit though.

Long and short of it is you should assume stress at a joint in a shell model is questionable. If the model is otherwise reasonable, the stress should become reliable a few elements past the joint. If you need better stresses in those joints, consider using solids locally. YES, you can model solids in the joint and transition to shells later. This technique is beyond the scope of this class.

If you want to learn more about weld modeling, I did a class at AU2015 called "A Job Weld Done" that goes into more depth.

Shell Model Example





1) The assembly is comprised of sheet metal components and 2 solids. This technique works best in this condition. Don't join your parts but do take out fasteners. You can add them back analytically later

2) In Inventor Nastran, choose "Find Thin Bodies" and accept the list

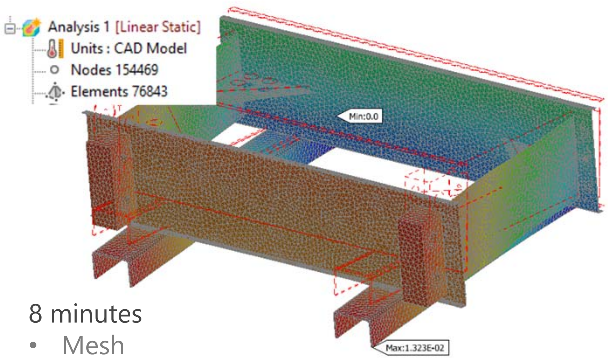
3) You'll see all components that Nastran identified as being good candidates for shells highlighted. If a part you wanted to make into a shell didn't make the cut, you can go back to that part later with more manual techniques.

4) All parts that were compressed to midplane are shown as orange surface bodies. Any left as solids are obvious. They'll mesh as solids so you are pretty much done prepping for the mesh.

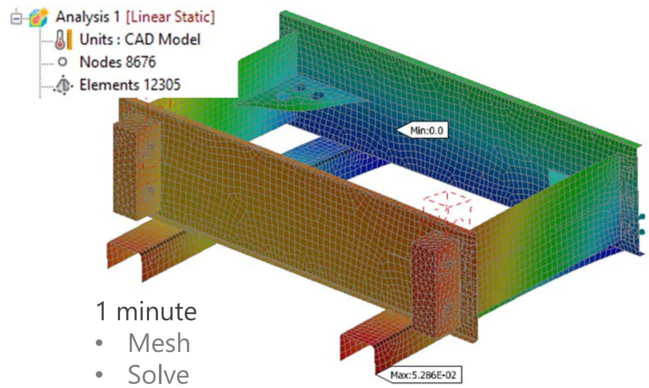
5) It is important that you tell Nastran to join the separate bodies. The unique feature in Inventor Nastran that makes this easy is the Offset Bonded option under Solver Contacts. By leaving the "Specify Contact Regions" field blank, Nastran will examine all the element faces and edges as it compiles the model for gaps to bond across. You specify the gap to consider with the "Max Activation Distance" field. This should be slightly larger than the biggest gap you expect. If all the parts are the same thickness, use that thickness because if there's a face-to-face gap, it will be 2 times ½ the thickness… or the thickness. In this model, the channels are ¼" thick and they mate to a 0.06" flange. The gap will be 0.03" + 0.125" or 0.155" I used 0.2" to cover my bases.

6) Once the model is meshed, the run takes less than a minute.

- **8x** Longer to mesh and run as solids (VERY SMALL MODEL)
- Solid model Displacement is **25%** of shell model – Consistent with element error



8 minutes
- Mesh
- Solve



1 minute
- Mesh
- Solve

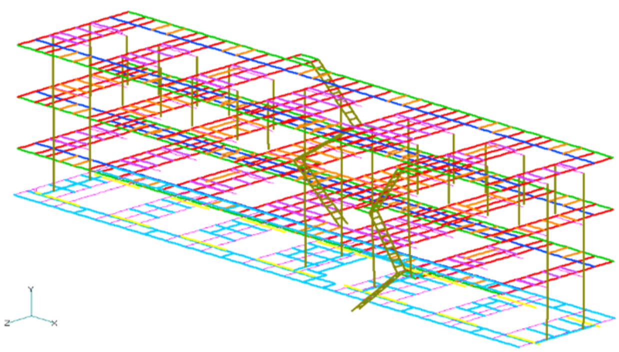
Note that I did run the same assembly as a solid model. It took much longer to run. 8 minutes may not seem like a lot but this was a very small sample model.

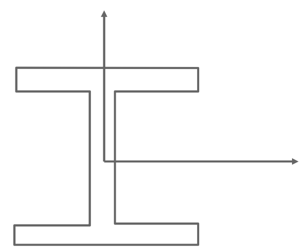
More importantly, the computed displacement is 1/4th the shell model's displacement. This is consistent with the stiffening effects of pancake tets and shear stiffening. The answers from the solid model might be OK for a trend study but not for an investigation on the designs overall performance.

Beam Modeling

Beam Modeling

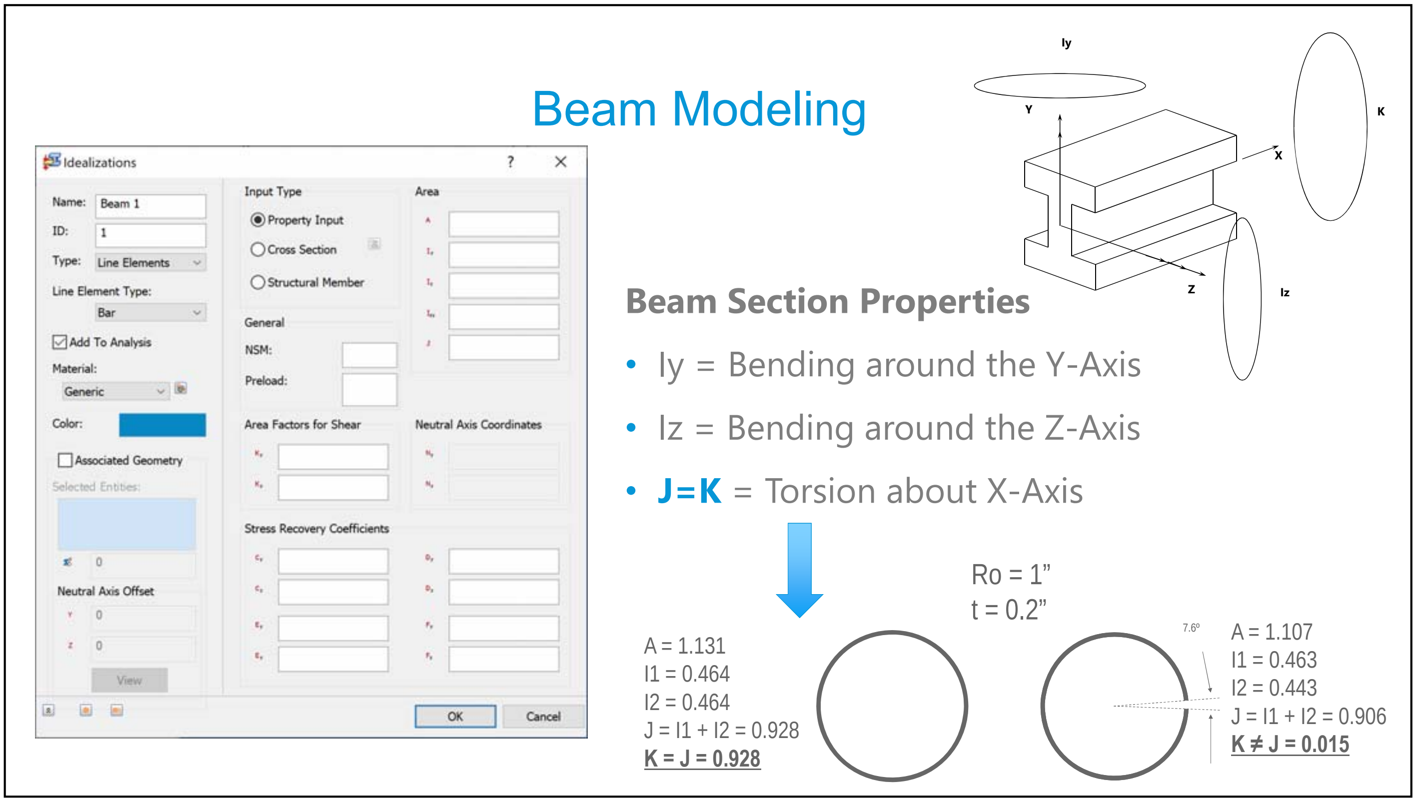
- When cross-sectional dimensions are small compared to length
  – Does Solid look like Wireframe?
- 3D Wireframe at neutral axes best for beam models…
- When are CAD solids the best starting point?





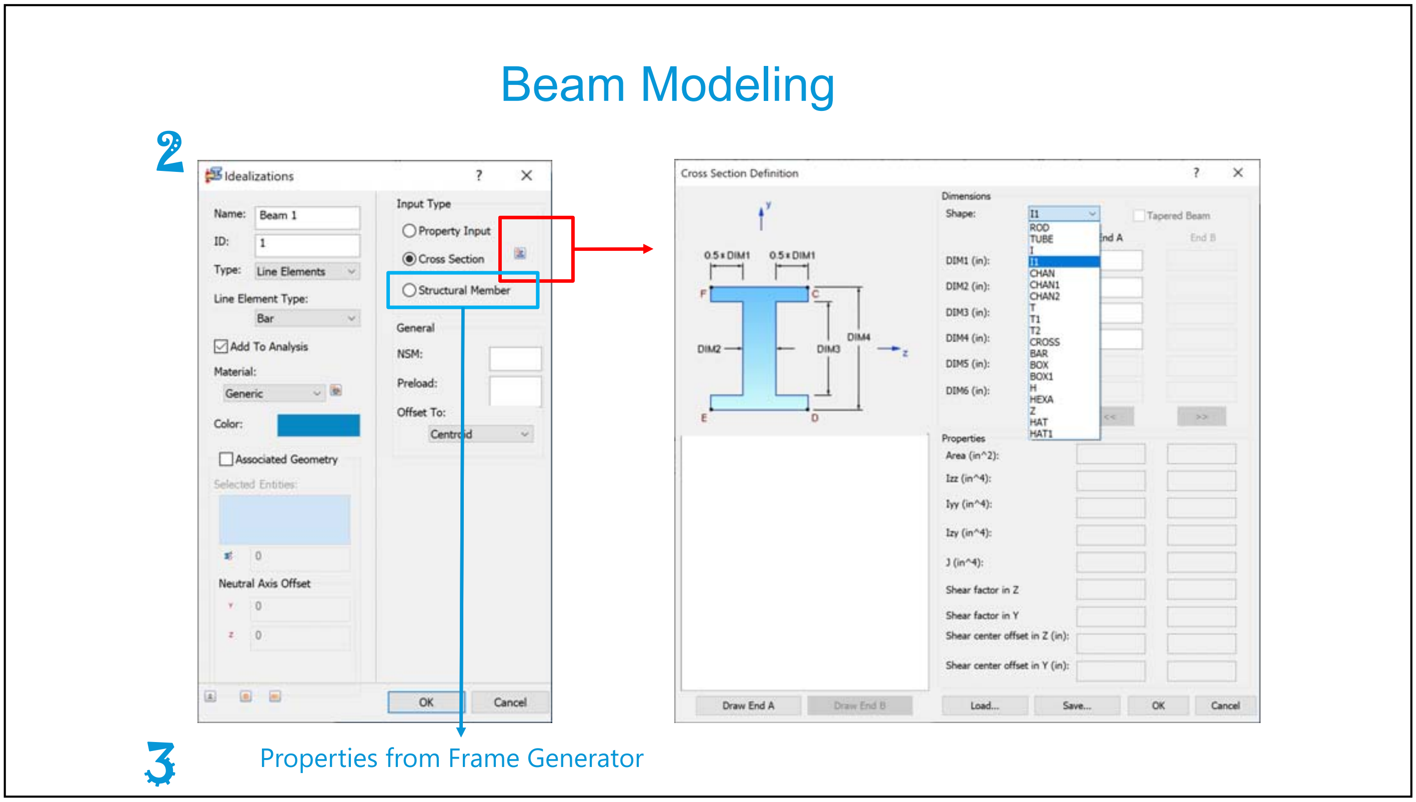
As stated previously, a line, or beam element forfeits it’s cross-sectional geometry for speed. The cross section is added back with properties.

There are very few cases historically where the CAD solid would be the best starting point. However, the use of Frame Generator in Inventor is a great way to begin models that have standard ANSI-type cross-sections. If these types of components are common in your day-to-day, you should be familiar with all the ways to build beam models in Inventor Nastran.



First, things you may have forgotten from college come into play. Moments of Inertia, neutral axes, and Torsional constants are some.

The software takes care of most properties these days but one poorly understood property is torsional stiffness. J is the most common symbol for torsional stiffness. J is defined as the sum of the bending moments. However, for open sections, this doesn't apply and can lead to erroneous torsional calculations. K is the correct property to use and can be found in tables and references on the Internet.

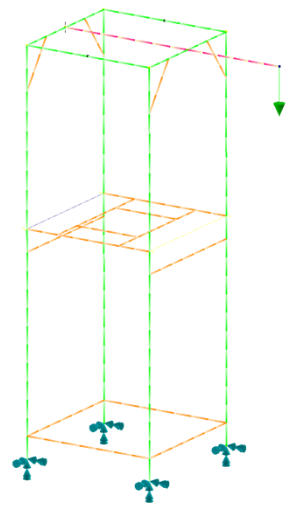


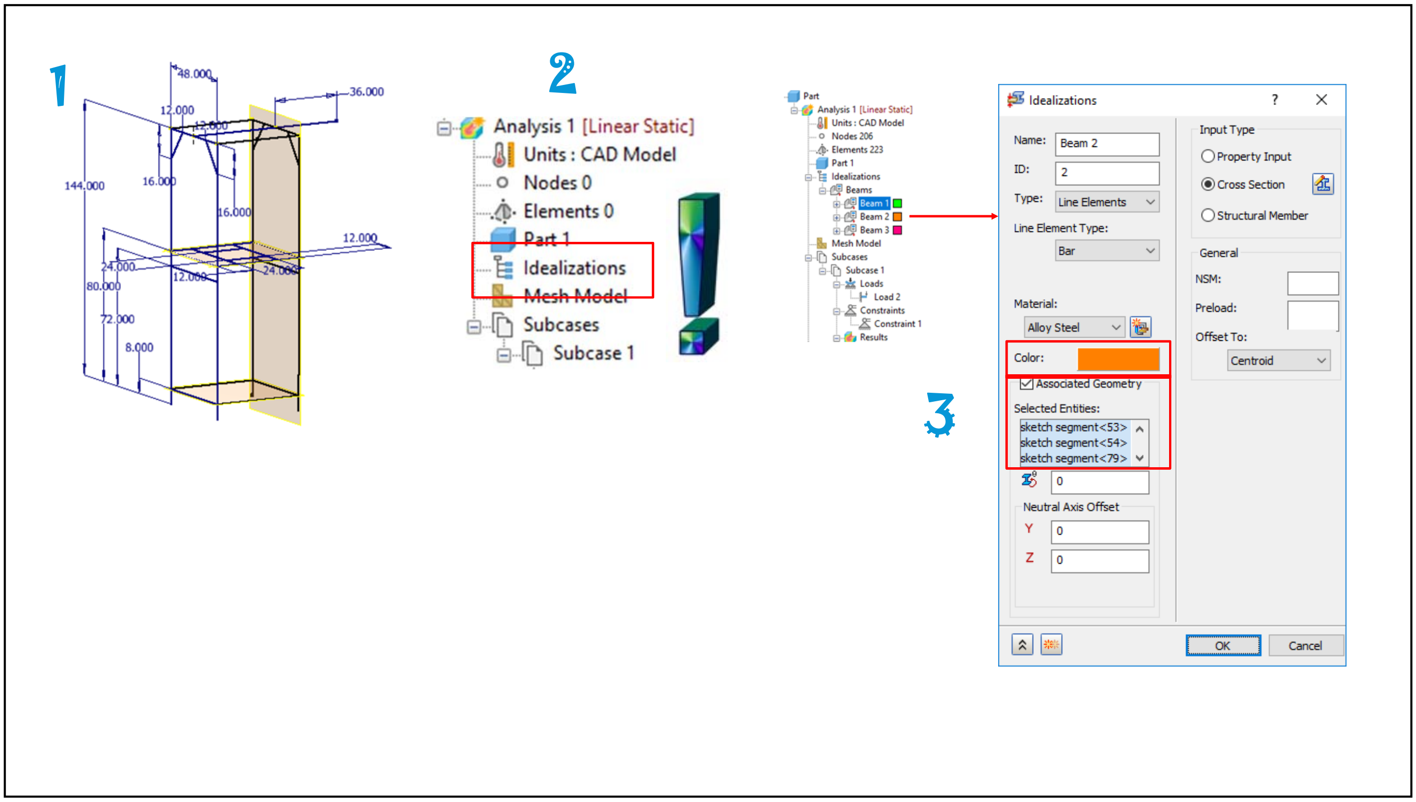
You have 3 ways to define section properties in Inventor Nastran. The first was shown in the last slide where you are responsible for EVERY property. This is had but its nice to have the flexibility if you need it.

The second uses the Cross-Section Calculator. You can start from a template and add detailed dimensions. The properties are calculated for you. We'll show this in an example.

The third relies on properly constructed Frame Generator components. Since the section is baked into those parts, Inventor Nastran can pull it out. We'll also show this.

Beam Model Example #1



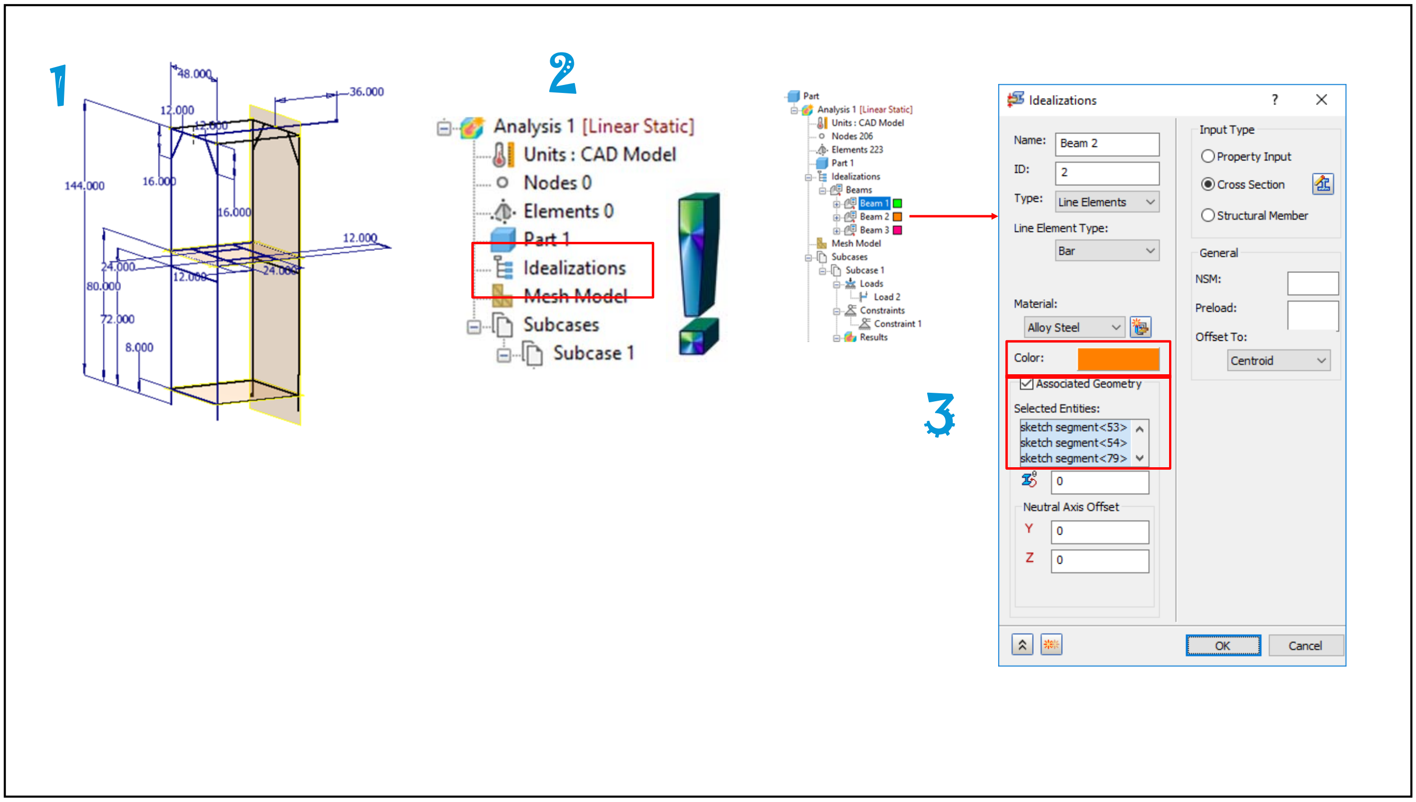


1) In this first example, a 3D sketch (actually a bunch of 2D sketches in a single part) was created to represent the base geometry of the assembly. The method to get lines and curves where they need to be is unimportant. You your judgement on this. It is important though to split each curve at joints or the mesher won't connect the beam elements across it.

2) You'll note that when the Inventor Nastran Environment is opened, there are no idealizations predefined. You'll need to create them manually and assign them to the appropriate curves

3) In this example, there are 3 cross-sections in play. I used the Idealizations button on the Ribbon and the Cross-Section calculator. I recommend choosing a different color for each section and assigning the idealization to the appropriate curves in the Associated Geometry field.

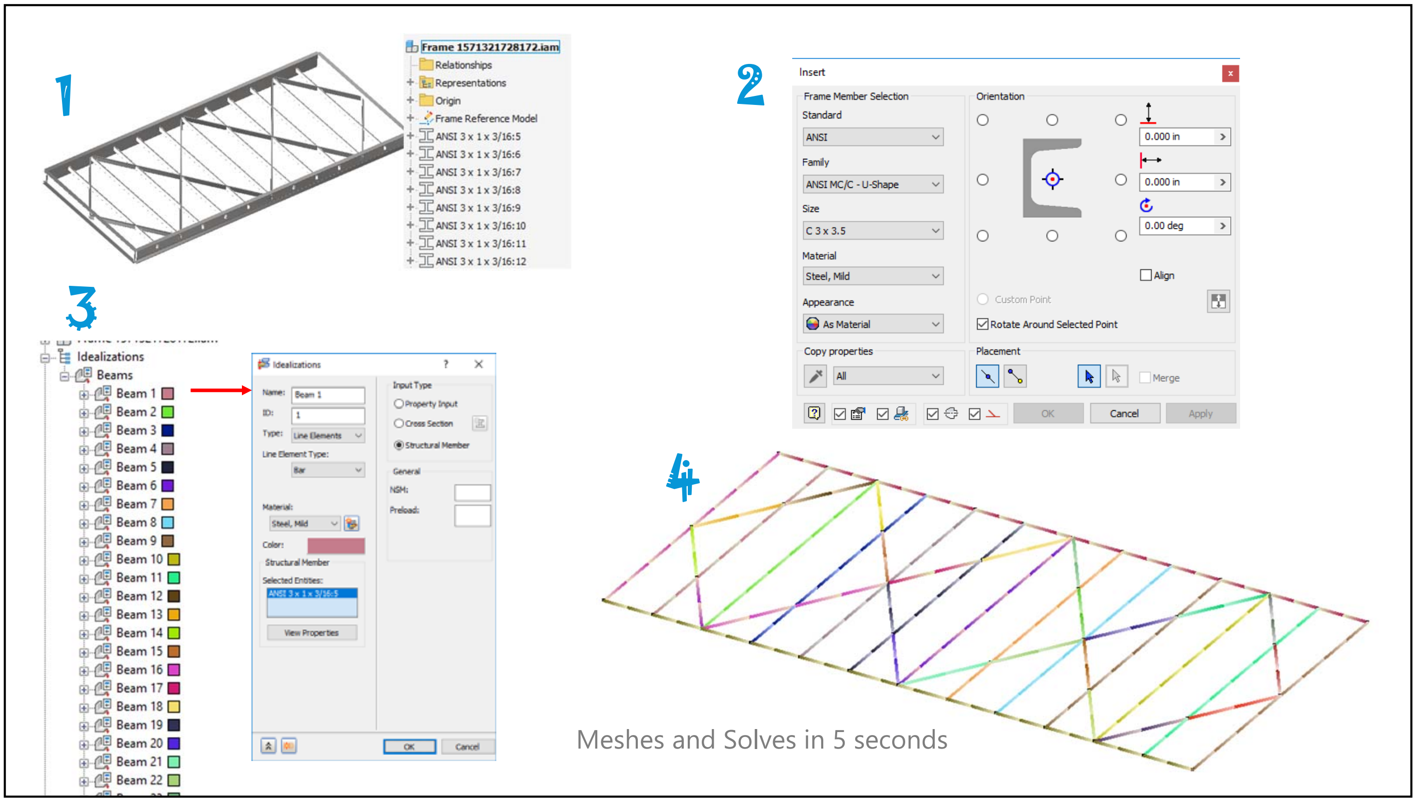
4) Loads and constraints are applied to points or lines and you are ready to solve.



While a bit laborious to get started, changes are really fast. If I want to explore the impact of different I-Beam depths or tube thicknesses, I can simply update a form directly within the Nastran Environment and solve it again. You can go thru dozens of iterations in minutes to ensure your structure is optimal.

One important technique on complex idealized assemblies is to solve a for the first handful (10 maybe?) natural frequencies to identify bodies that didn’t tie in correctly. You may not see the discontinuity in an operational loading scenario but it can impact your results. This technique should be used for shell models too.

Beam Model Example #2

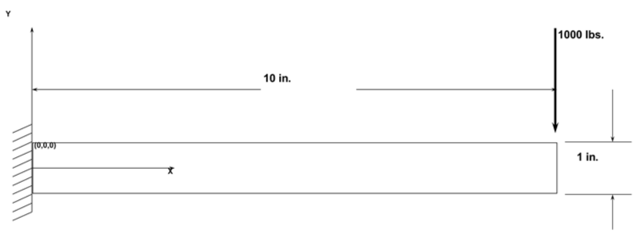


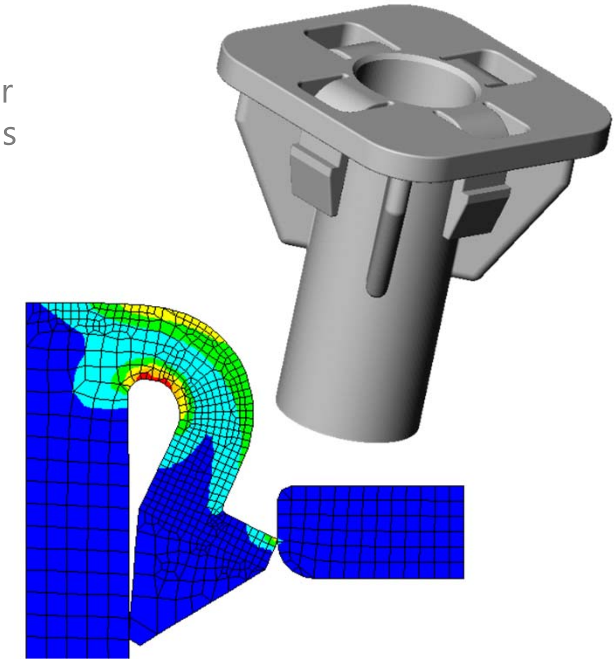
1) This assembly was built with Frame components in Inventor.
2) You'll note that these sections are visually more compelling than the prismatic sections in Nastran. The properties computed are essentially the same
3) When this assembly is pulled into the Inventor Nastran Environment, you'll see a separate Beam Idealization was created automatically for EVERY SEGMENT, splitting single members at their joints. While this is automatic, finding which idealization is which segment can be really hard. You have to trust your CAD was right on complex models.
4) The meshed model solves fast. Geometry changes must be made at the Frame Part level, you can't change properties in the Nastran form

Other Geometry Considerations

Cross-Sectional Models

- Properly positioned solid cross-sections or 2D surfaces are required for planar models
- When are CAD solids the best starting point?



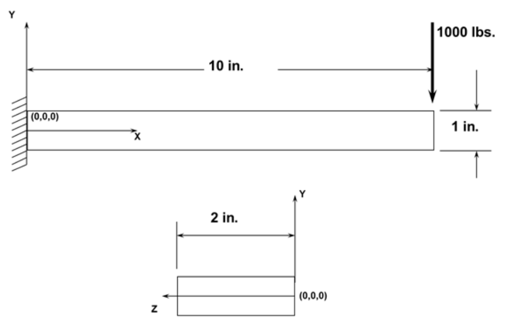


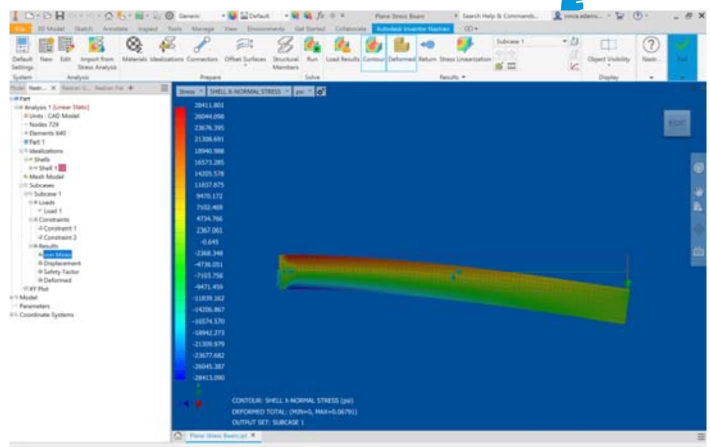
Just as hand calculations on simple geometries are often a more expedient starting point than FEA, a planar, or cross-sectional model may be the next step if complexity exceeds Roark & Young.

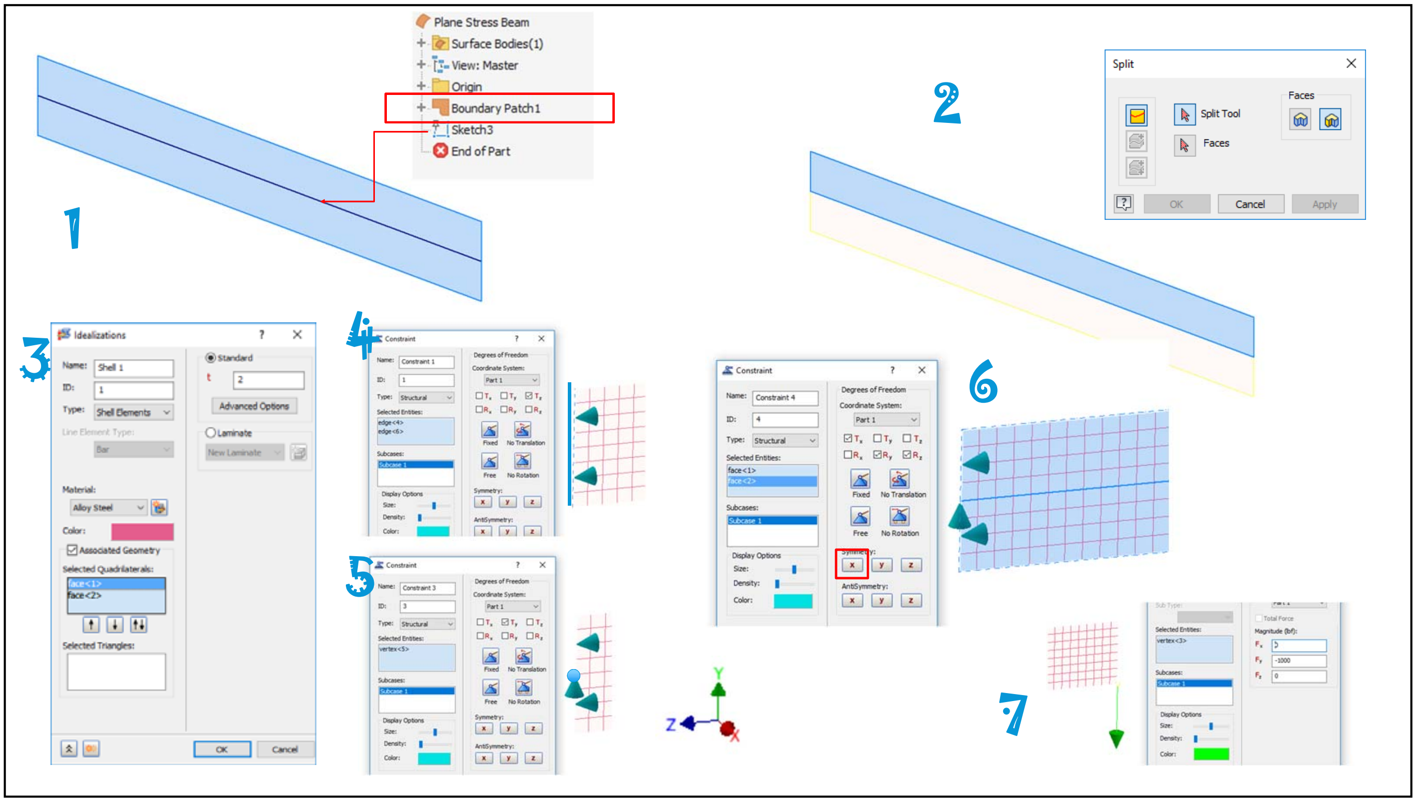
In the case of the snap-fit mount, the only design challenge was the snap feature. There was sliding contact so hand calcs didn’t fit. A plane stress model made short work out of this problem as the ideal section was identified after a few dozen iterations. That section was incorporated into the 3D model and the part was finished for tooling.

Inventor Nastran doesn’t have a plane stress or axisymmetric model space as some other solutions do but they are easy to mimic using the tools available.

Planar Model Example







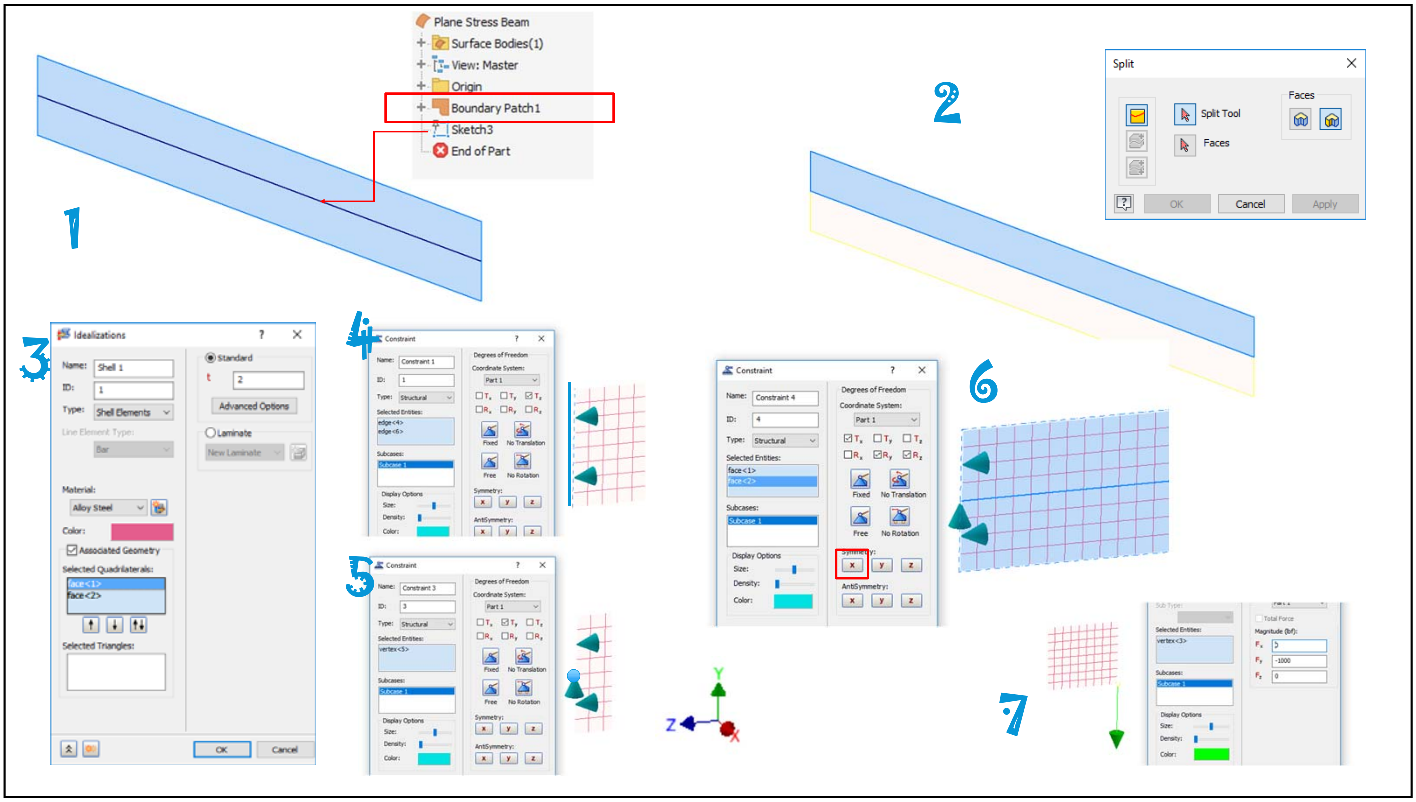
In the case of the cantilevered beam we led the class off with a few tips may help you leverage plane stress modeling.

1) Start with a 2D surface, however you choose to create that. It can be the face of a solid.

2) For this example, I wanted a point in the center of the constrained edge. The Split feature in Inventor is the best way to force that. You can use this technique for load patches, fastener engagement patches, or for any reason you may need to isolate a portion of a face or surface. One use is for contact models. If the contacting areas are very predictable, you can define contact between two limited patches vs. two complete parts. The solver won't have to search so many element faces to determine if contact occurs.

3) For Plane Stress, define your shell to be the depth of the body. This contradicts the 'thin shell' concept but works in this case.

4) For this model, constrain the fixed edge only in length direction of the beam. This essentially attaches it to the fictional rigid face but can slide up or down.

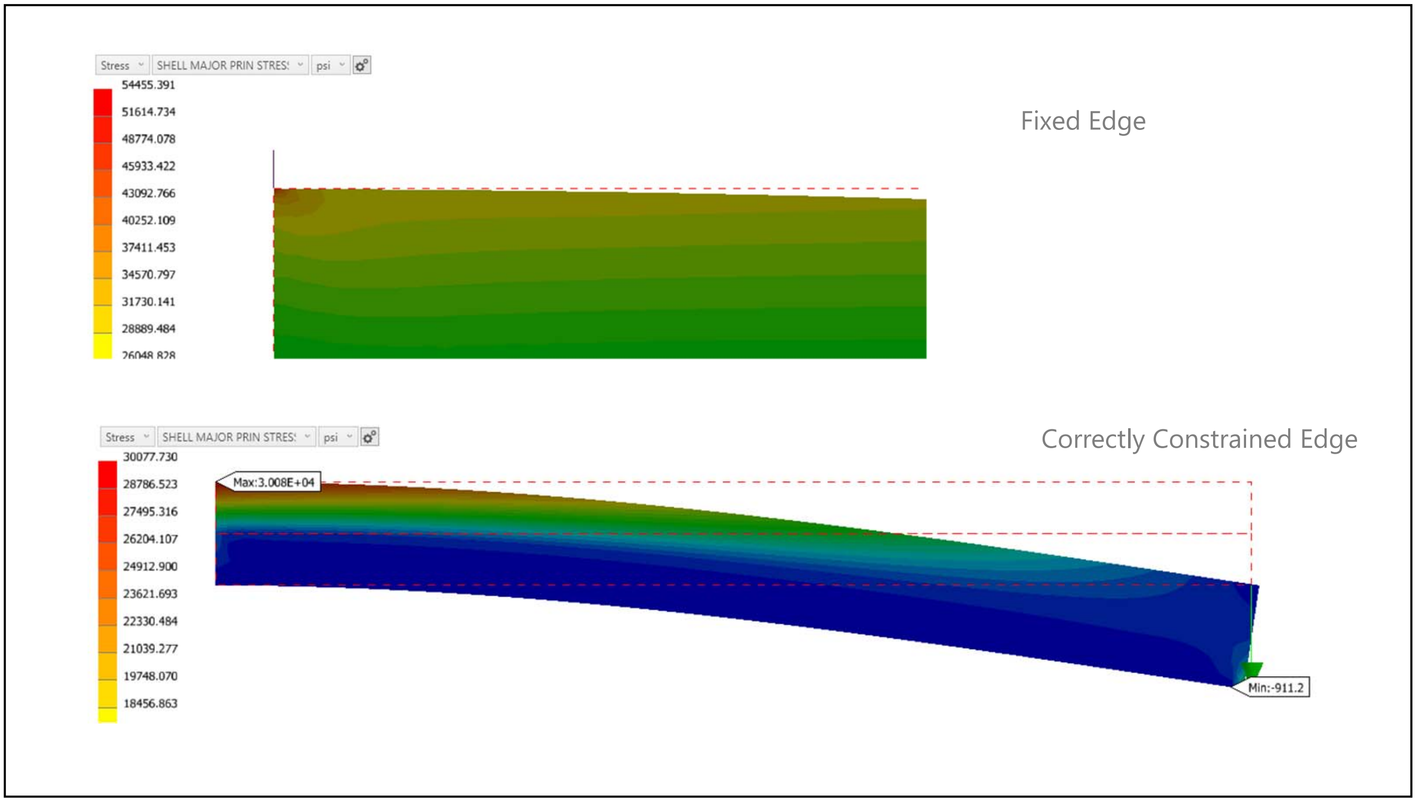


5) Constrain the center vertex created by the split vertically.

6) Constrain both faces to remain in-plane.

7) Apply the point load to the proper vertex.

The reason I chose to show this complex-seeming constraint scheme is to highlight a problem that an occur in 2D and 3D models. Poisson Effect singularities.



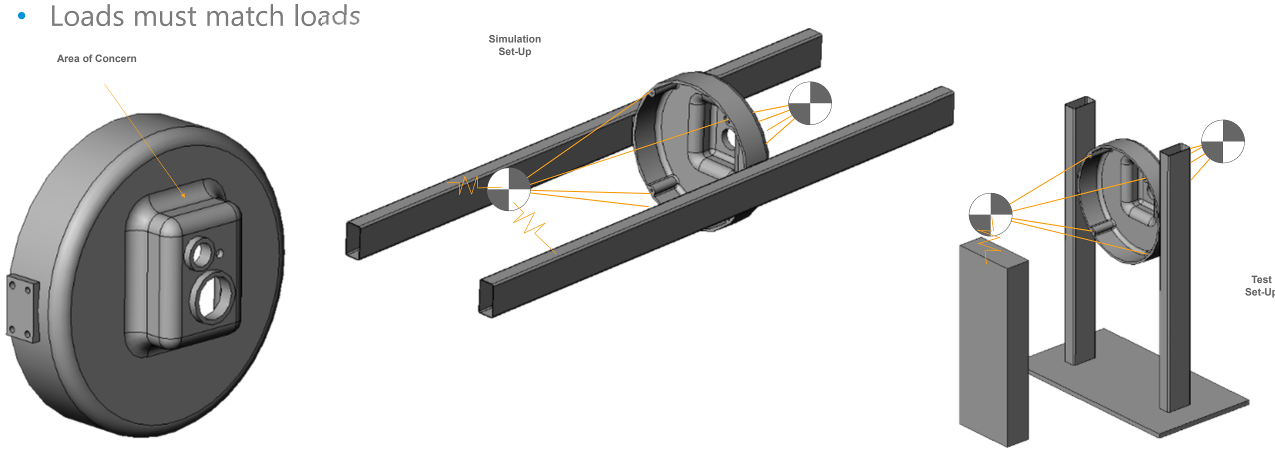
In the top, “Fixed Edge” example, the edge was fully constrained as you might expect to do based on the diagram. Note the stress exceeds 50ksi in the top corner. This is because as the top fibers in bending want to stretch, Poisson’s ratio forces those nodes to contract to the neutral axis. (Google Poisson’s Ratio if this is unclear.) If you didn’t recognize this, you could conclude the effective working stress was nearly twice the 30ksi shown in the lower model, computed with the more natural method shown in the example.

Poisson Effect stress can mimic the hot spots caused by geometric singularities, “red spots,” so understanding why you have those hot spots is key to knowing how to handle them.

Correlating to Test

Test What You Analyze… Analyze What You Test

- Correlation to test requires that the simulation matches it. Obvious?
- Parts must match parts – Does the as-manufactured part match as-designed?
- Assemblies must match assemblies
- Loads must match loads



AUTODESK®

Make anything.™