

IM124940

Better Model, Better Mesh, Better Results: CAD Modeling for Analysis

Mark Huntoon, P.E.

Learning Objectives

- Understand what constitutes a good mesh, and what are common issues.
- Learn how to model your part or assembly so that you can consistently get a good mesh.
- Learn additional tricks to get a good mesh quickly.
- Learn how to apply the knowledge from this class to your workflows for upfront simulation.

Description

To obtain better simulation results, a systematic and powerful mesh is the key. While some finite element analysis (FEA) software products have advanced meshing tools to help obtain a superior analytical model, applying these tools can be complicated. Using the guidelines and tricks taught in this session, during your CAD modeling for simulation with Autodesk software, you can develop a better mesh more efficiently and obtain accurate and precise results. This class will explain how to recognize mesh qualities and how to use CAD modeling techniques to obtain a refined mesh more efficiently which will save time and money in your analysis efforts.

Speaker(s)

Mark Huntoon is a licensed engineering consultant from Wisconsin, where his primary focus is on mechanical analysis using simulation, with an emphasis on natural hazard loading on equipment. He has a Master of Science degree in Mechanical Engineering and Bachelor of Science degree in Civil Engineering, Structural emphasis, both from Marquette University. Additionally, he has over 15 years of experience, including being the Chief Engineer for two companies. He has served as the Simulation Solutions Engineer for MasterGraphics, an Autodesk Platinum Partner. While he left to do engineering consulting, he continues to teach and mentor on Autodesk Simulation Products for MasterGraphics and other resellers. He has given presentations on a variety of engineering and software topics at Autodesk University 2015, BuiltWorlds, and The Future of Making Things at Lambeau Field.

Better Model, Better Mesh, Better Results: CAD Modeling for Analysis

The Finite Element Method is a powerful tool that can help solve complicated problems. But when performing this type of analysis, there are assumptions that are made by the user and are inherent in the method. These assumptions can lead to errors in the analysis; some can be minor and insignificant while others can be catastrophic. The skill of the user can reduce these errors with experience and knowledge; allowing this method to be an invaluable asset to your design process.

From the Textbook Concepts and Applications of Finite Element Analysis by Cook, Malkus, and Plesha:

“Computer graphics has achieved such a level of polish and versatility as to inspire great trust in the analysis,... (One can now make mistakes with more confidence than ever before.)”

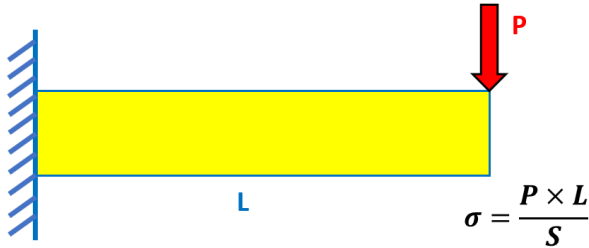
“Although the finite element method can make a good engineer better, it can make a poor engineer more dangerous.”

So to assist you in effectively using this method, this course will cover how to start with a good CAD model, which leads to a better meshed analytical model, and ultimately better results.

First a quick review of the meshing theory used in the method will be covered with simple explanatory examples. Then we will discuss how to build a model in your CAD application that will provide a better mesh for your analytical model. Next, we will talk about other tools to setup your analysis and improve the mesh. Finally, we will discuss how to use these topics in your workflow to obtain and recognize better results.

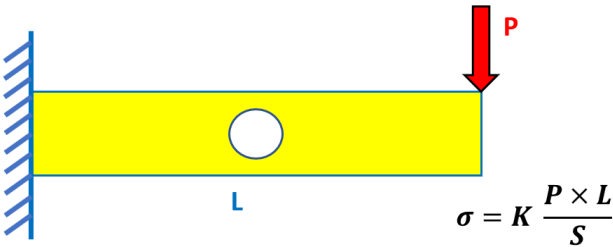
From this class you can become more powerful in your engineering and use your CAD modeling skills to create accurate analytical models to answer your design questions.

But before we cover how to create a better analytical model, it should be said that FEA may not be required. There could be quicker and more industry accepted methods to answer your design questions. Take the following cantilever bar example.

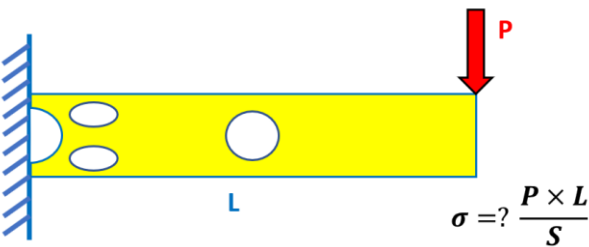


This type of problem can be easily solved using the known equation shown.

If a hole is added in the center of the span, as shown below, it is easy to look in your reference materials or with a search to determine the appropriate stress concentration factor to get an accurate value for the stress.



Now, if you have more irregularities in the section or closer to the area of higher stress the ability to determine an accurate stress becomes more complicated.



This type of problem requires a more flexible and powerful analytical model, this is where the Finite Element Method becomes an indispensable tool. But with the great power comes the responsibility of confirming the results.

- **So, the first tip for better analysis is to use the right tool. If you can solve the problem with an industry accepted method easily, use it!**

Understand what constitutes a good mesh and what are common issues

To get started on how to get better utilization of the Finite Element Method, let's review some of the fundamentals on meshing with this method.

Mesh Element Types

The application of Finite Element Analysis will primarily utilize one, up to all, of the three elements described below.

Beam Elements

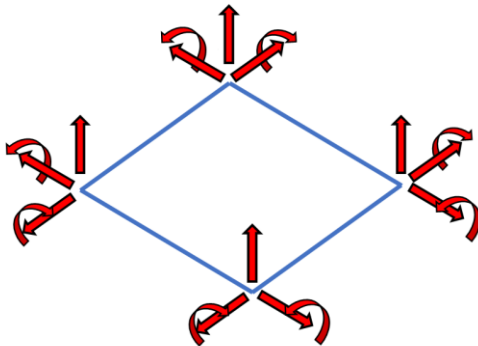
The first element is a beam element; this type of element has three translational and three rotational degrees of freedom at each node.



This element provides for the most displacement functionality, but it has the greatest limitation in terms of applicability. It assumes a structural profile with constant cross-section so there is a possibility of error if you do not have this type of part. But with engineering judgment, this type of element can be applied to a variety of applications. This element can solve very quickly and provide accurate results.

Plate Elements

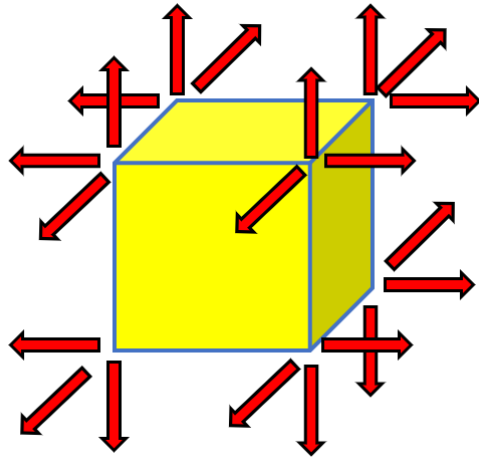
Next are plate elements. These elements are infinitesimally thin and are best used when the thickness of the part is much less than the other dimensions, such as sheet metal. The plate element has three translational degrees of freedom and two rotational degrees of freedom, both out of plane, at each node.



These elements provide for a fast way of analyzing thin plates and typically provide more accurate results with fewer elements than using solid elements.

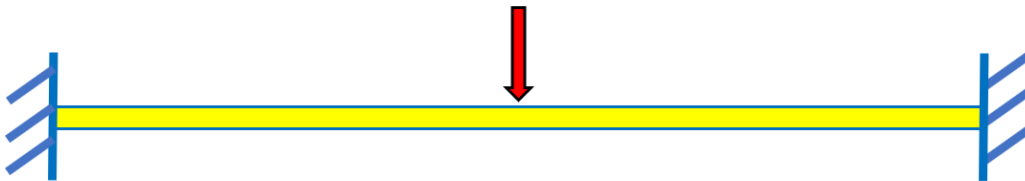
Solid Elements

The last element type is the most adaptable to complex geometries. At each node there are three translational degrees of freedom, but no rotational degrees of freedom.

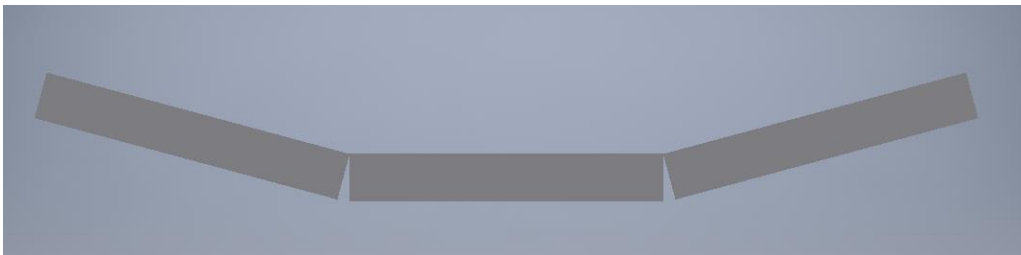


While these elements can be used for complex geometries, they do not exhibit rotation and therefore you need enough elements to model the displacement properly to get good results.

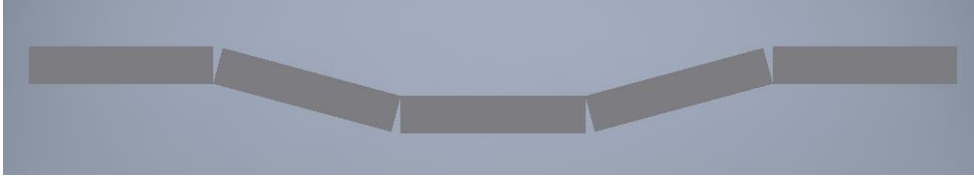
For example, if you have a beam that is fixed at both ends with a point load in the center.



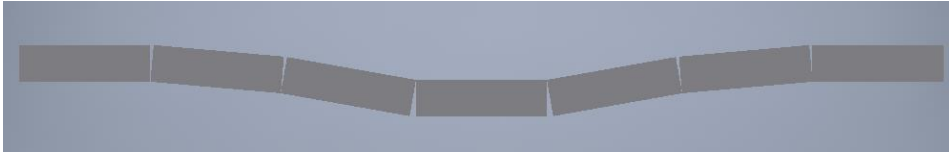
If we use three solid-like elements we would get the following displaced representation.



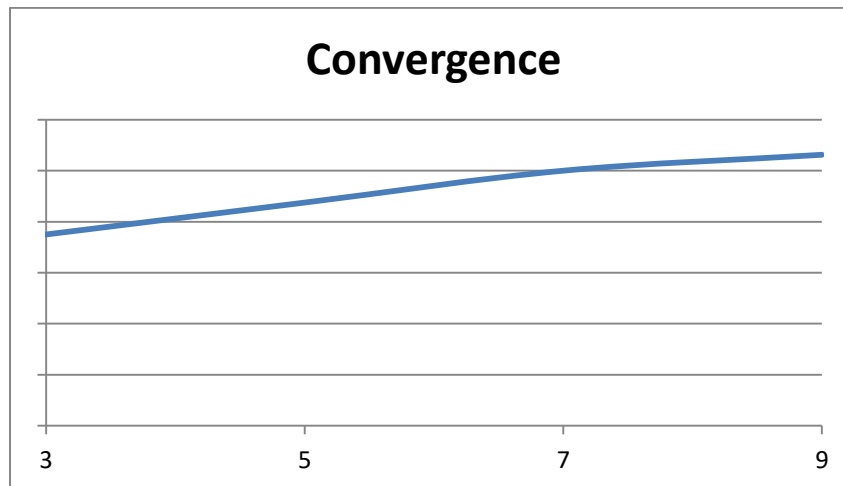
This would not be adequate to model the displaced shape of the loaded beam, so now we will try using five elements.



The five elements would match the displaced beam better, but would still not be accurate enough. Now we will try seven elements.



This matches the shape even better, and would give us an accurate model. We could increase the number of elements, but they would not result in dramatic improvement in the results, this is referred to as convergence. Convergence is when that an increase in elements does not result in more accurate results for the additional computation time.



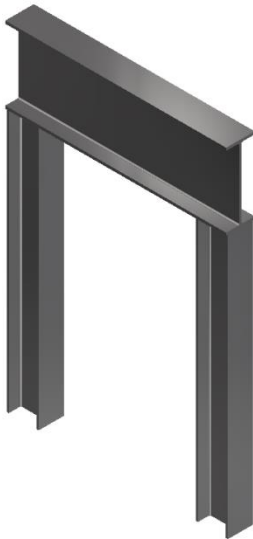
- **Making sure that you have an adequate number of elements is critical to a good analysis.**

Though this is not necessarily something that we can improve with our modeling techniques directly, how we setup our CAD models can have an impact on the number of elements required in the analysis.

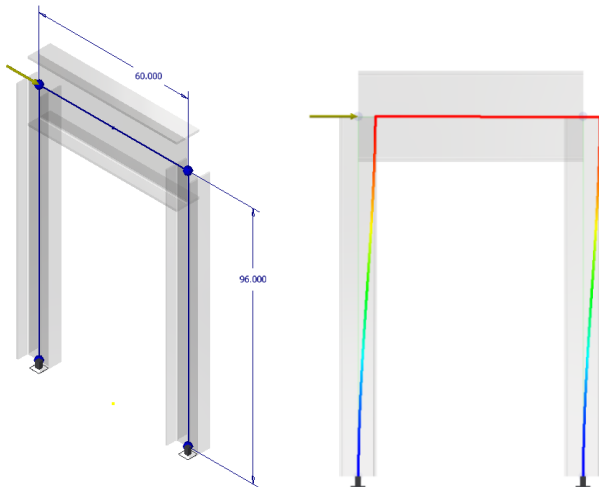
Element Relative Size

But the number of elements is not the only key to a good mesh, also important is the size distribution of the elements. It is important to have elements that are close to the same size.

Let's say you have a portal frame with two columns of equal length and a beam that is infinitely stiffer than the columns.

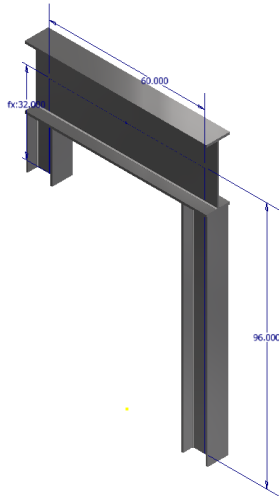


If a horizontal load is applied the portal frame would deflect and since it is much stiffer the beam would not deform and the columns would take the deformation.

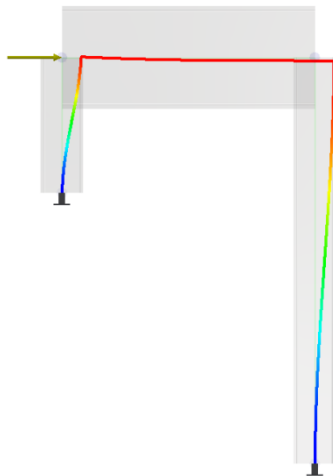


We can see that the deflection is the same for both columns, so the stress in each column would be the same.

If we look at a similar portal frame, with the only difference being that one column is shorter than the other.



When the load is applied, the portal frame deflects in much the same way.



But since the deformation is the same in each of the columns, the strain would be higher in the shorter column.

$$\epsilon = \frac{\Delta}{L}$$

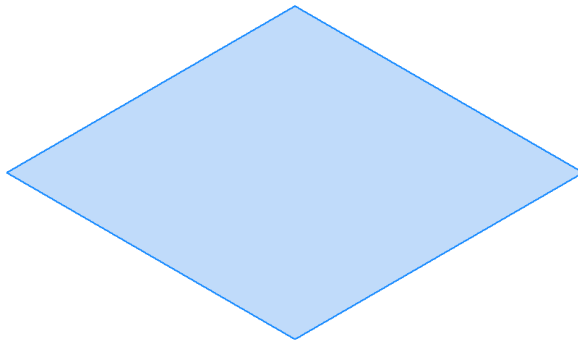
Since the strain would be larger the stress would then be higher. To apply this to a Finite Element model, when you have two elements next to each other their shared nodes would deform the same, but the smaller element would have a much larger stress. One type of stress singularity is caused by having a very small element next to large elements, where the stress in the small element would be much higher and not be accurate.

This is the first and most important tip for CAD modeling for better analysis. If you have small regions or small geometries in your model, this could cause you to have proportionally small elements in that area and cause higher stresses than what is actual. So, if something in your model is small and not necessary to the question you are asking, get rid of it for the analysis.

- **Remove small features in your CAD model to keep your mesh element sizes close so their relative stiffness is compatible. Otherwise you will have to increase the number of elements to get a more reasonable stress distribution.**

Element Shape

We have established for a good mesh, elements should be close to the same size, but they also need to be as regular shape as possible. The equations for stiffness are based on each element having equal edge lengths. To illustrate the effect of elongated elements, consider the following flat plate.



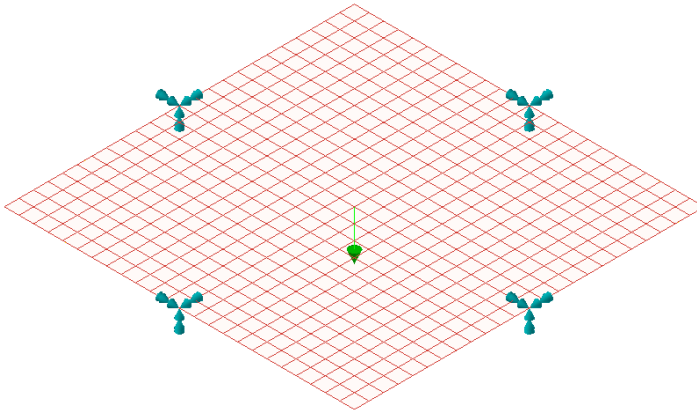
First, we will calculate the maximum stress using the method from Roark's Stress and Strain. If you do not have a copy of Roark, you can search online for a plate calculator or use the one in Inventor for the case of thin plate parts.

- **It is always a good idea to use a known and accepted reference to compare our results too, even if we are just looking at just a portion of our model. This will still provide a baseline for what kind of element distribution is required for our analysis.**

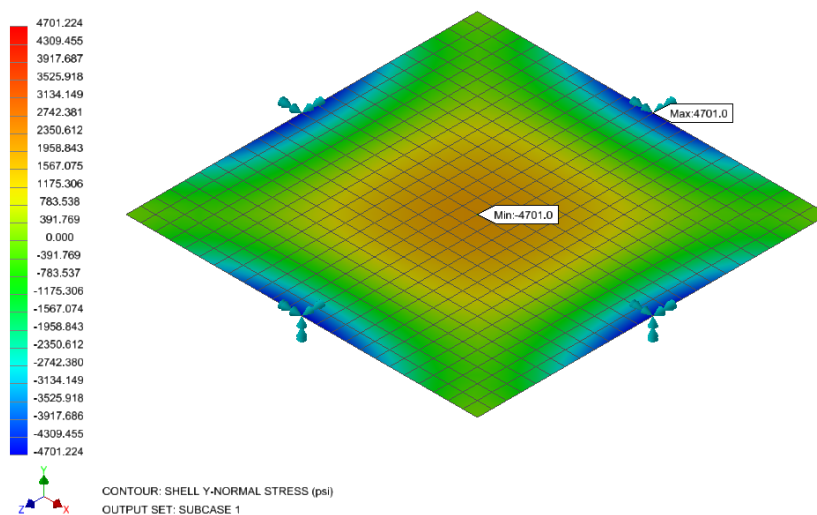
From Roark' Stress and Strain Table 11.4 Case 8

$$\sigma_{max} = \frac{-0.3078 * 0.434psi * (48in)^2}{(0.25in)^2} = 4924.8 psi$$

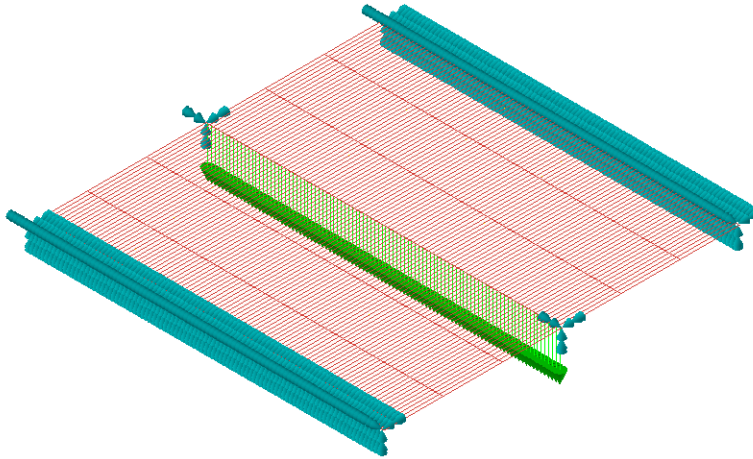
Now we will use FEA to calculate the maximum stress. Here we are allowing the meshing algorithm to determine the element configurations, and it does a very good job.



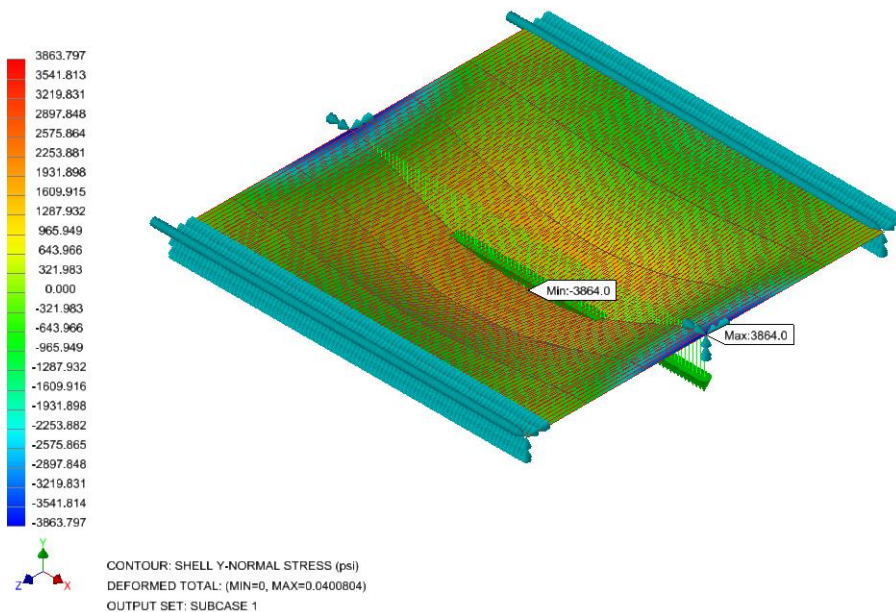
Running the analysis, we get a very close value to our hand calculation.



Now, we will go into the model and we will create splits to force the meshing algorithm to create elongated elements.



After running the analysis, we can see that the resulting stress is less accurate than when we used the regular shaped elements, even though the same number of elements was used.



- From this example we now know that having elements with sides that are close to the same will result in better results.

Section Summary

To help summarize what we have covered so far, let us circle back to the plate element definition and the need for plate elements.

First, we want to make sure that we are using enough elements to model the deformation. If you have a thin part such as sheet of plywood, you would need multiple layers of element or plies through that part to adequately match the deformed shape.

Second, we know that our elements to have side lengths that are close. So, if you had the sheet of plywood that is $\frac{1}{2}$ " thick, and you used two solid elements thick, keeping the sides of the elements close would result in over 8000 elements and a long computation time.

The plate used in the FEA example above was 48" x 48" and a reasonable result was obtained with only 256 elements. This helps us circle back to the first tool for better analysis, use the right tool for your problem.

- **If you have a sheet of plywood, or another thin part, use plate elements for better results and create them in your CAD system as shells. The analysis software will allow you to change the thickness to evaluate options without having to change the CAD model.**
- **If you are analyzing long structural members, use beam elements, and you can create just a stick model in your CAD system and vary the member sizes in your analysis tool. This will help keep your model simple and flexible for early design phases.**

How to model your part or assembly to consistently get a good mesh.

Now we will discuss some effective CAD modeling practices that will result in you getting a better mesh and thus better results. The first practice is probably the most effective; make your analytical model as simple and small as possible.

When I was in college, my professor used the example of a car driving from Milwaukee to Chicago. If the question you are asking is how long the trip will take, you do not need to analyze a fully detailed car. If the car during the trip is involved in an accident and you want to know the force that is generated by the impact, the cars involved can be modeled as a generic mass and an analysis can be done. Now if you want to know if the stress caused by the accident could make the gas tank rupture, and create a larger problem, this is a more complicated problem but it can be solved efficiently.

The entire car still does not need to be modeled; you can just focus in on the gas tank itself and the way it is connected to the car. Using constraints and loads you can get determine if the impact from the accident would cause an issue from the collision.

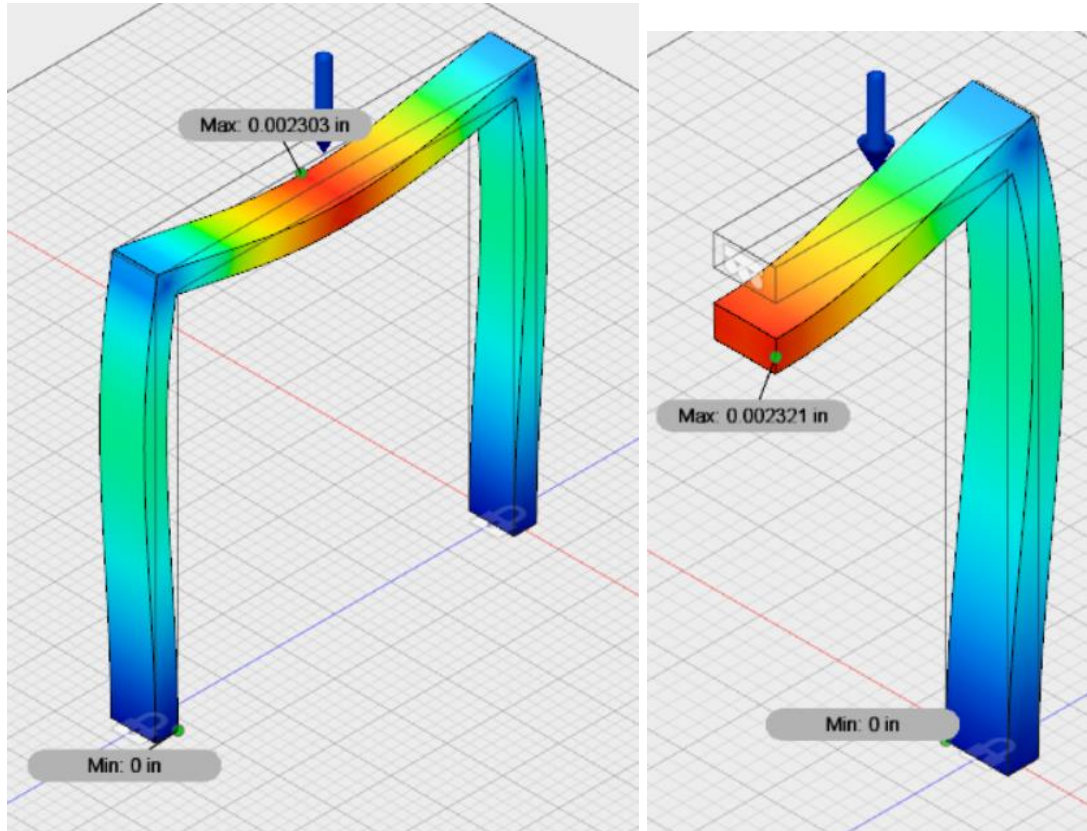
In my consulting world, when I am presented with a project I always try to limit the size and scope of my analytical model. One example is a hinge bracket on a septic truck that failed, I only modeled the bracket and using the constraints where the bracket connected to the tank and the truck frame, I was able to analyze and solve my client's problem. This is also similar to the number one issue that I have found when doing mentoring and training: people try to analyze the entire assembly and not just the parts of interest. Their models get large and complex, when they start to mesh and analyze them; they get errors and get frustrated. So to avoid frustration, keep your models simple.

When Vince Adams of Autodesk would go around and do a presentation on Mastering FEA he would tell people "Make sure you are asking the right questions!"

- **When you are taking on a problem and are going to be using FEA, ask yourself what is the question I am asking and how to most efficiently get the answer. Then use an analytical model that encompasses what it will take to answer those questions, and nothing more, or else you are just using up unnecessary computation power and time.**

Symmetric and Repetitive Models

If you have a part or assembly that has symmetry and your loads are symmetric then use only half the model in your analysis. With the symmetry constraints provided in Nastran In-CAD you can analyze half the model, and still get a valid result. In Inventor Professional and Fusion 360 you can utilize the Frictionless Constraint.



- When you enable symmetry, you can cut your number of elements required roughly in half, thereby reducing your computation time significantly.

Taking this a step further, if you have a repetitive assembly, then create an analytical model of one instance and then use this same base model and then run multiple analyses for the multiple instances. One type of project where I have used this methodology is air handling equipment for commercial applications. The customer has asked me to certify their large assemblies for hurricane loads in Southern Florida and these assemblies look similar to the picture below.



Used with permission from Carrier Commercial Systems.

Here we have multiple condenser units that are nearly identical, so rather than try and create one model for each complete assembly and analyze it, I took a different approach. I took one of the condenser units and constrained it where it meets the support frame and where it connects to the adjacent units. I then applied my loading to it and got the results for the center unit. I then removed the constraints on one side and replaced them with loads based on the reactions of the first analysis. I then continued this for all the internal units. This was a huge time saver, as I have multiple large assemblies to analyze and I could maximize the usage of the individual condenser analysis. This did require some thought and engineering judgment, but it allowed me to be very efficient in my analysis.

- **So, if you have an assembly with repetitive parts or assemblies, creating one analytical model and using it in multiple analyses is a good strategy.**

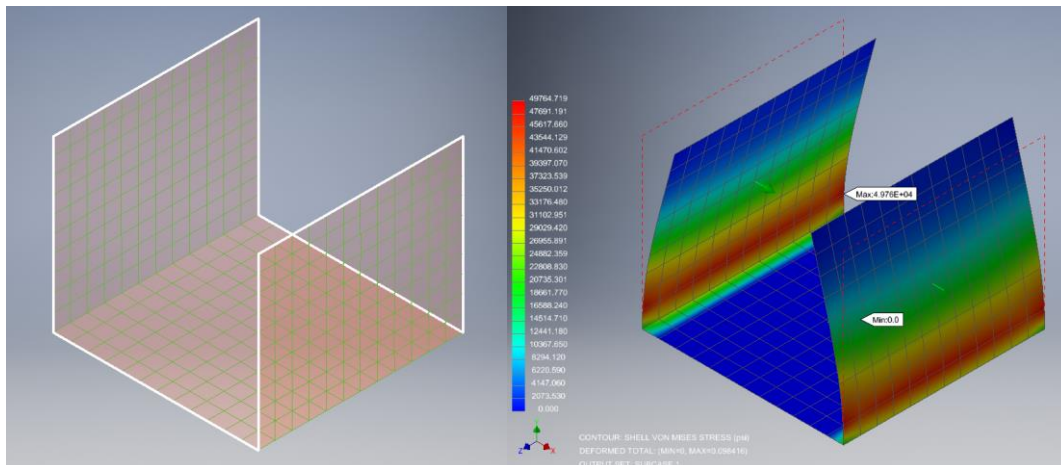
Model based on the Elements you intend to use.

These projects, and others, have taught me to use a CAD model that is based on the analysis elements that I intend to use. This has led to much more accurate and faster solving models, as well as much less frustration.

Plate Element Models

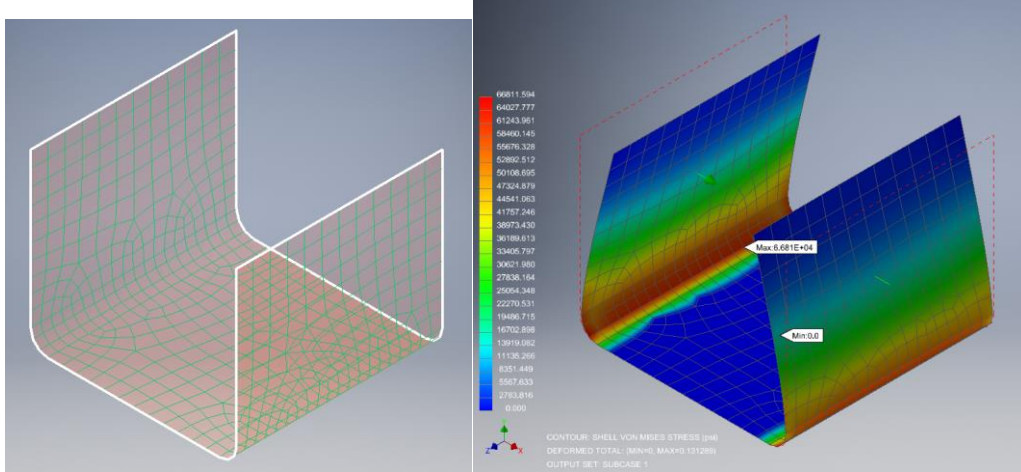
Earlier in this class we discussed the benefit of using plate elements for thin parts. If you are going to be using plate elements, then use surfaces in your model rather than the sheet metal feature or solid bodies. This trick works best when using Nastran In-CAD, and is beneficial in many ways. It can make your model much simpler and faster to create, but it also allows you to determine the necessary thickness within the simulation tool by just changing the thickness in your element setup and doing multiple runs. This method is valuable in upfront design, because you know the area required but the tool will help you determine the thickness before you make the effort of modeling the sheet metal parts.

If you do use shells, the other trick is when you get to corners. If you use a sharp corner like what is shown below, this can create very high stress that are not representative nor in the area that would be expected in the actual application.

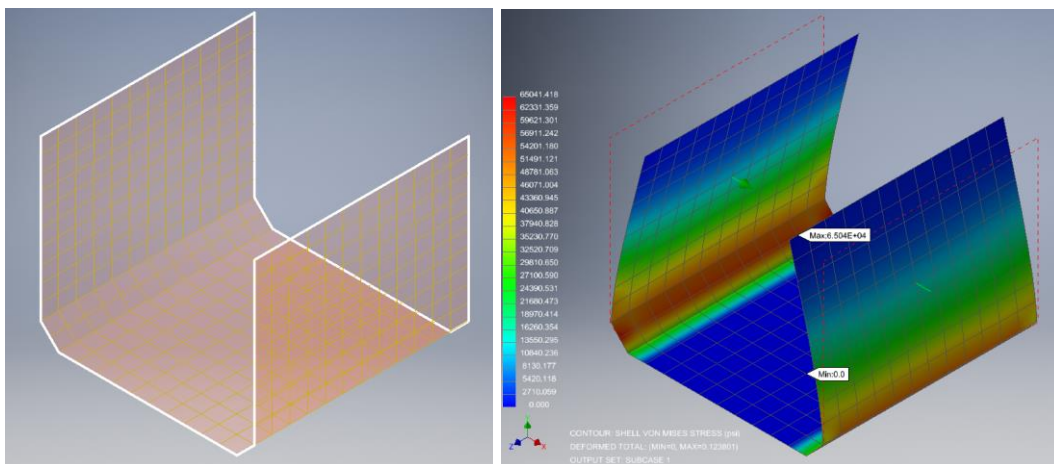


The high stress in this bent bracket is in the lower part of the side walls since the base is being constrained, this is not what would be expected. The high stress would be in the bend of the plate bracket, not above it.

Since sheet metal is bent and there is a radius, you can model a transition at these corners. But do not use a rounded corner like a fillet. These cause problems when meshing with plate elements, because it is more difficult for these elements to follow a curve. You will either get a very small mesh in this area, and as discussed before this can cause issues, or you will get a mesh that does not match the CAD model surface very well.



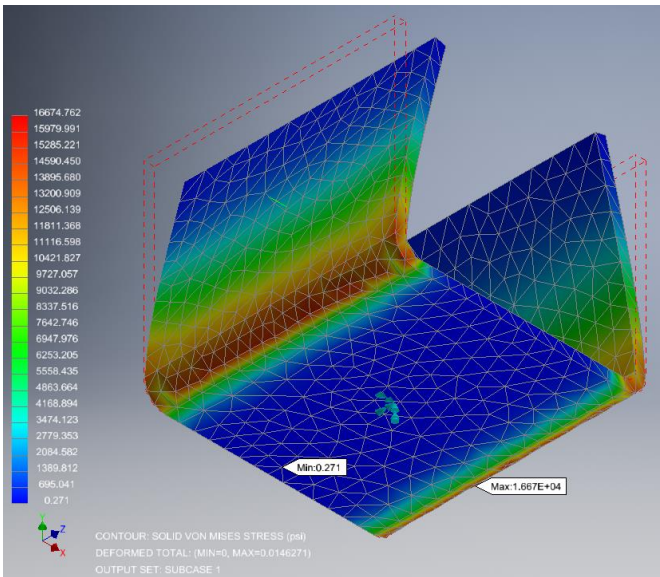
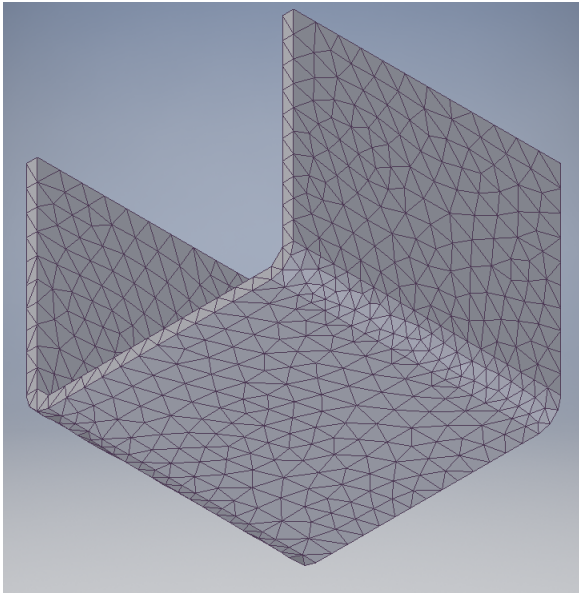
This also does not create a very clean delineation of stress distribution since the elements vary in size and shape. Instead use more of a chamfer in your model. Since plate elements can handle out of plane bending, this can provide you a decent result.



This creates a much better stress distribution, similar to what would be expected in the actual application. These chamfers do not have to be very large either; just make sure that the elements used in the chamfer are not significantly smaller than the surrounding elements. When using this modeling technique, from my experience, it is best to keep your included angle to be 135 degrees or more. Otherwise you may get stress concentrations at these turns.

Solid Element Models

If you are going to be using solid elements, then instead of using chamfers with plates, you should use rounded corners at transitions. Because, as described above, solid elements do not model rotation they will not transfer displacements well if using a chamfer. Having a multiple element transition at a corner is your best technique for solid elements.



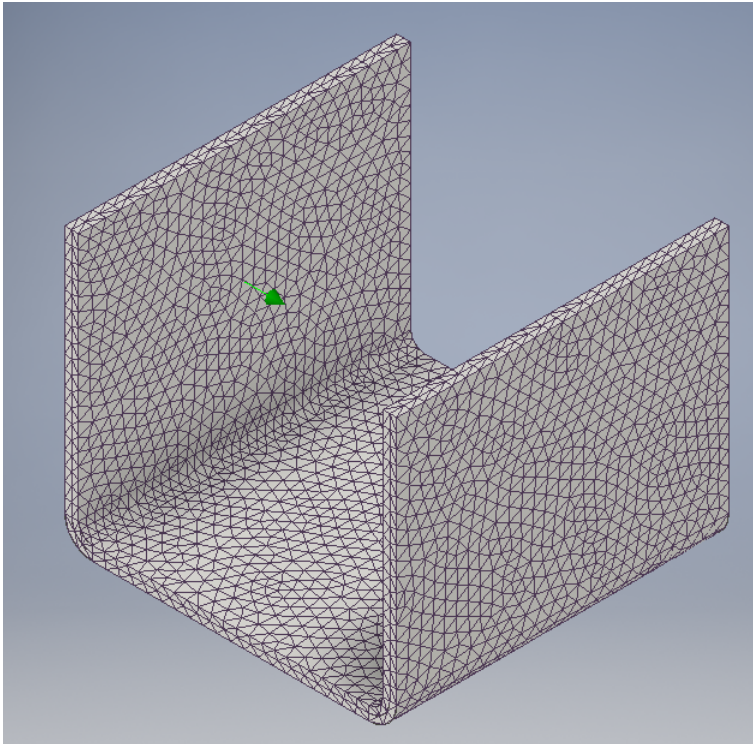
The results show a nice distribution of stresses at the corners, but not as good as we obtained from using plate elements. This is because the variations in element size and shape, reinforcing that using a shell model with plate elements are a better method to analyze this bracket.

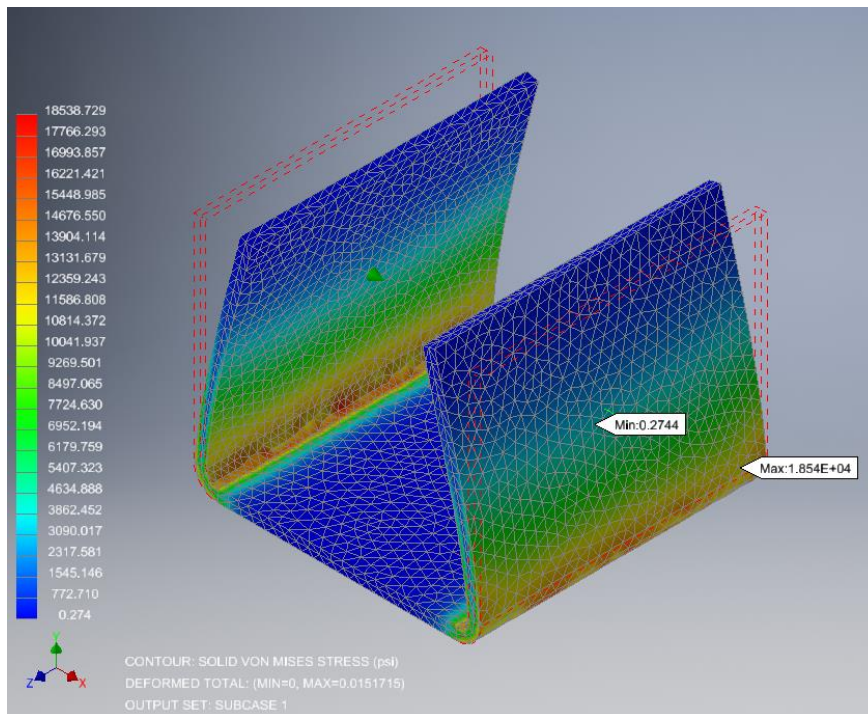
Using rounded corners with solid elements may require either using smaller elements, in which case the nearby elements must not be significantly larger, or increasing the size of

your transition. While a larger transition may affect the calculated stresses, if the main area of concern is away from the transition then the stress in the transition area may not be relevant for the question you are asking. If your area of concern is the transition, then perhaps you could do a more focused analysis on that area by itself.

There are times when a solid bodies, and thereby solid elements, are required in the analysis. I have come across this several times in my consulting projects for different reasons. If the solid modeled thin part is important to your analysis, then you need to make sure that there are multiple elements through the thickness of the part; otherwise the elements may not behave in a way to get you adequate results.

When I have come across these situations, I have relied on the split command in Inventor. By creating a work plane or a sketch through the thin part, at the center of the thickness, then splitting the faces you can force the meshing algorithm to create two layers through the thickness. Now, this does take some practice and you need to adjust your element growth factors so that the mesh algorithm does not deviate on larger pieces and you end up with only a single element through the thickness.





It can be seen that the stress varies through the thickness, and like a cantilever beam, the stress would not be the same through the part when loaded in this manner.

Splitting the thickness will require smaller elements, but to ensure that you are not using more elements than are required, use a representative basic plate model of about the same size and compare your results at different mesh sizes to what can be calculated using Roark or another reference. This will help you set your mesh at an adequate size.

Beam Element Models

If you can answer your question using beam elements, then just a stick model is required. This type of model can be used with Frame Generator to create an analytical model that can be analyzed in Inventor Frame Analysis or Nastran In-CAD. Custom profiles can be created and used in the Frame Generator, so you are not limited in your available profiles.

If you intend to use plate elements, then model a surface model with chamfers and analyze with Nastran In-CAD. You can also model with solid elements for solid bodies, but remember to use rounded corners and watch for thin elements. Nastran In-CAD also allows you to use all three types of elements, and additional special elements, so you can model different parts in the manner that you intend to mesh them.

- **Therefore, when creating a better model for a better mesh, model based on what type of elements you intend to use.**

Section Summary

CAD modeling for analysis is the best strategy. First determine what will be required to answer the question you are asking. This is also true when setting up your analysis type, a static linear analysis will not produce post yielding or buckling behavior.

If you have a symmetric or repetitive model to analyze, then simplify the model. For symmetry, you can use the symmetry constraint in Nastran In-CAD or use the Frictionless Constraint in Inventor and Fusion 360. For a repetitive assembly setup a single unit for the analysis and use that single model form multiple analyses. Simple is better and both of these strategies will help you keep from getting frustrated.

Once you have determined what your analysis model will include, next model based on the types of elements that will be used in the analysis.

- **Beam elements should start as a line model**
- **Plate elements should be modeled as CAD surfaces**
- **Solid elements are created from solid CAD bodies.**

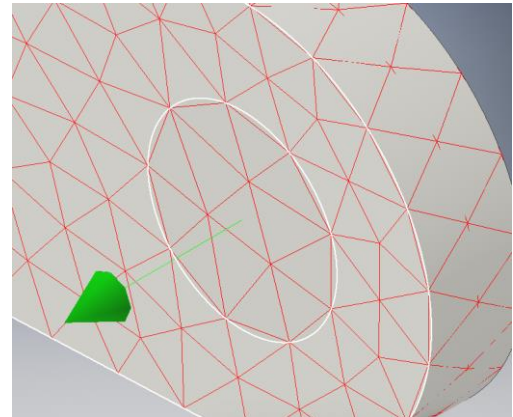
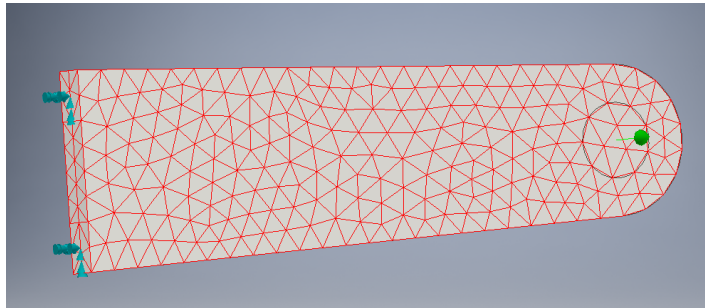
When modeling an assembly for analysis, make sure all the parts are touching. Nastran does have the ability to create offset bonded contacts and Inventor can close gaps when creating contacts, but it is just good practice to have adjacent parts touching in the model. This reduces the chance for error if the meshing algorithm does not behave the way you expect or want it to. Once you have proved a design concept works, and possibly optimized it, you can go back and remove those touches to create the final manufacturing assembly model.

Additional tricks to get a good mesh quickly

Now that a simple CAD model has been created for the analysis, let us discuss some additional tricks and tools that can improve our mesh in the simulation tool.

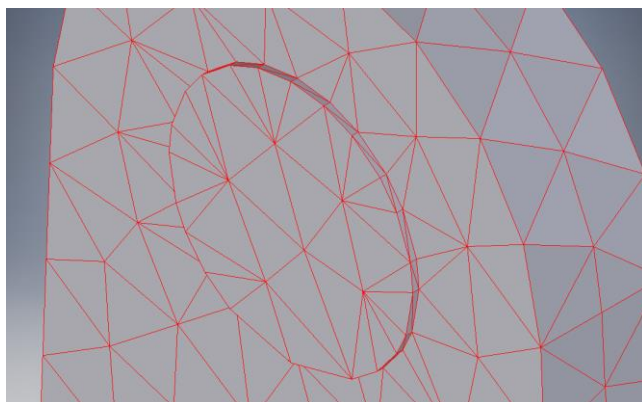
Splitting Surfaces

As was discussed earlier, the split command can be used to ensure there are multiple elements through a thickness. But the split feature can be used for other uses as well. If you have a load or constraint applied over only a portion of a part, you can use this feature to separate this area and then apply it to the smaller area.



You can also use the split to isolate an area for meshing, as was done for splitting a thickness, the meshing algorithms respect the edges of a split so you can have a different mesh size or pull results from your analysis in that specific area. The split feature is a powerful trick to expand the capabilities of your analysis tool and get you better results.

An alternative way to focus a load or constraint on an area is to create a raised surface as can be seen below.



The raised surface creates small elements that will have disproportionate stiffness, which can lead to less than accurate results. So the split function is a better method.

Meshing Tools in the Simulation Application

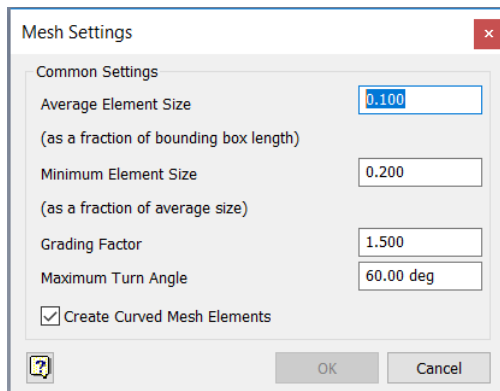
Once in the analysis, tool there are different options for controlling the mesh size and shape. To get an adequate number of elements to model the behavior the average size of the element can be changed. Meshing techniques is a large topic on its own, so it will be only briefly covered in this class. Below we will limit the discussion to solid elements, but there are similar options for plate and beam elements.

Mesh Tools in Inventor Professional

Controlling the mesh in Inventor is done with setting an average size element based on the bounding box. While this does not initially appear to have a much to offer for meshing control, there are other controls that can help you.

The first is the choice to mesh based on the part or assembly bounding box, controlled by the “Use part based measure for Assembly mesh”. You can also use local mesh sizing, where you specify the absolute size of the element and apply it to surfaces that need the finer mesh.

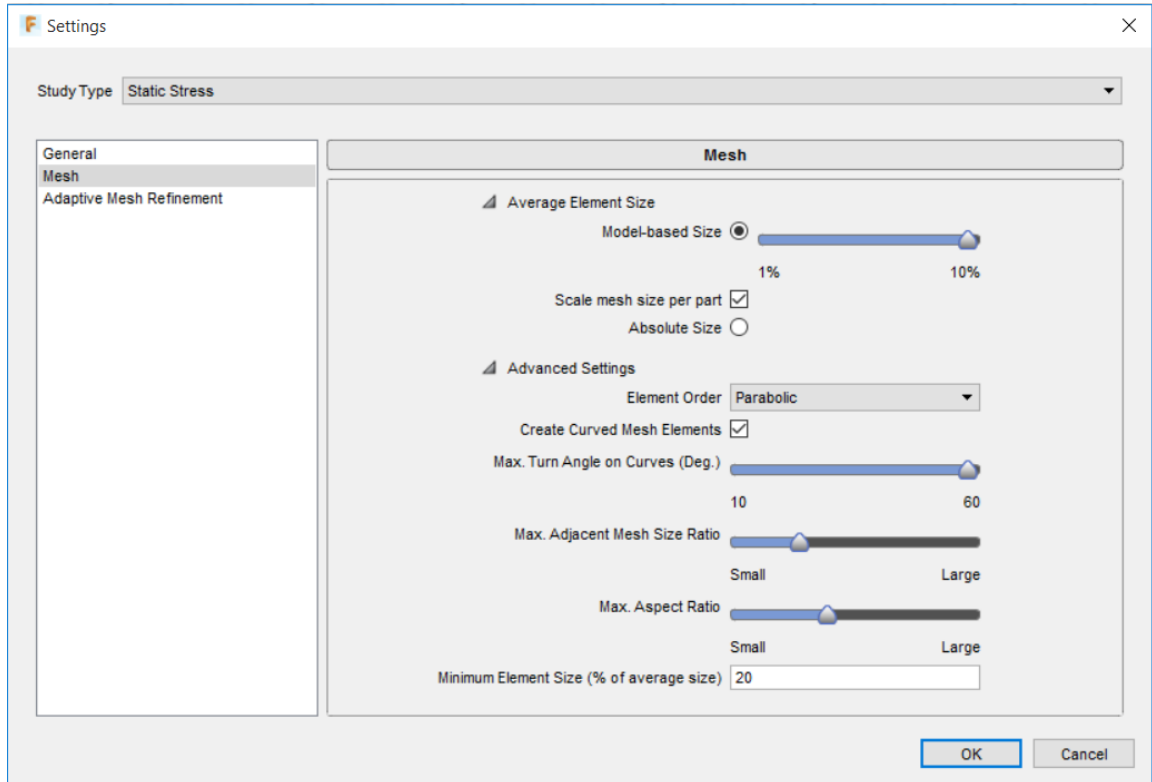
Local mesh can be helpful, but it is recommended to be used after the mesh refinement has converged. Otherwise you may get divergent results since the mesh refinement does not alter the local mesh settings. The best tool to control these is the growth rate variables in the mesh settings dialog box. Here you can control how fast the mesh grows or shrinks among adjacent elements.



I prefer to use a higher value for the “Minimum Element Size” and a lower value for the “Grading Factor” as I feel this provides a more uniform mesh. We discussed the benefit of this in the first section.

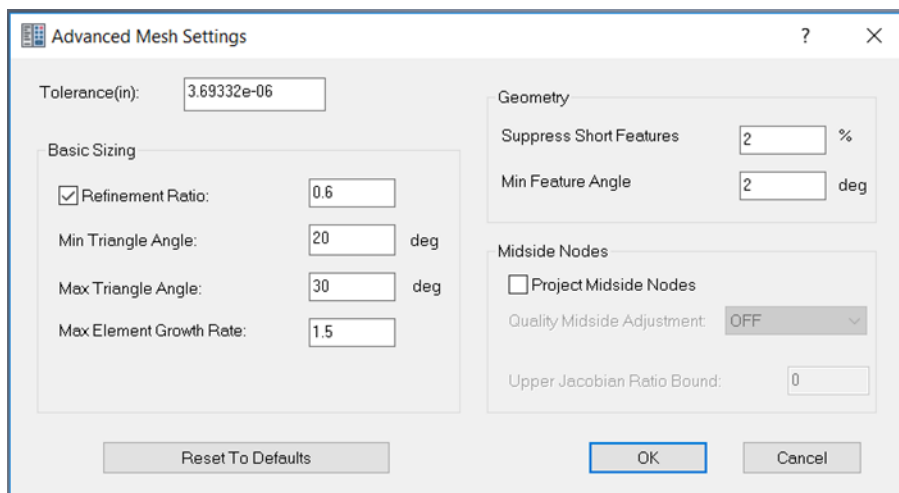
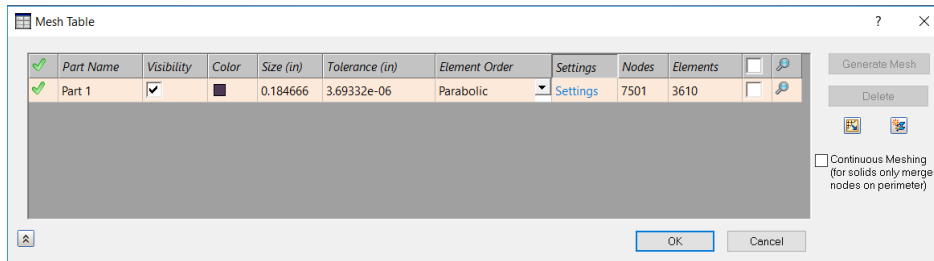
Mesh Tools in Fusion 360

Fusion 360 has many of the same controls as Inventor, but also allows you to set your average element size as an absolute value which can be a huge time saver.



Mesh Tools in Nastran In-CAD

Nastran In-CAD provides you the most control with meshing. Since you setup each part independently as an idealization, you can set the mesh size for each part independently. You can also specify the minimum and maximum angles used in your triangular shaped elements.



Here again, my preference would be to keep my growth rate low. Also, if two parts are going to have a bonded contact, it is best to try and keep the element sizes close so stiffness and displacement can be transferred effectively.

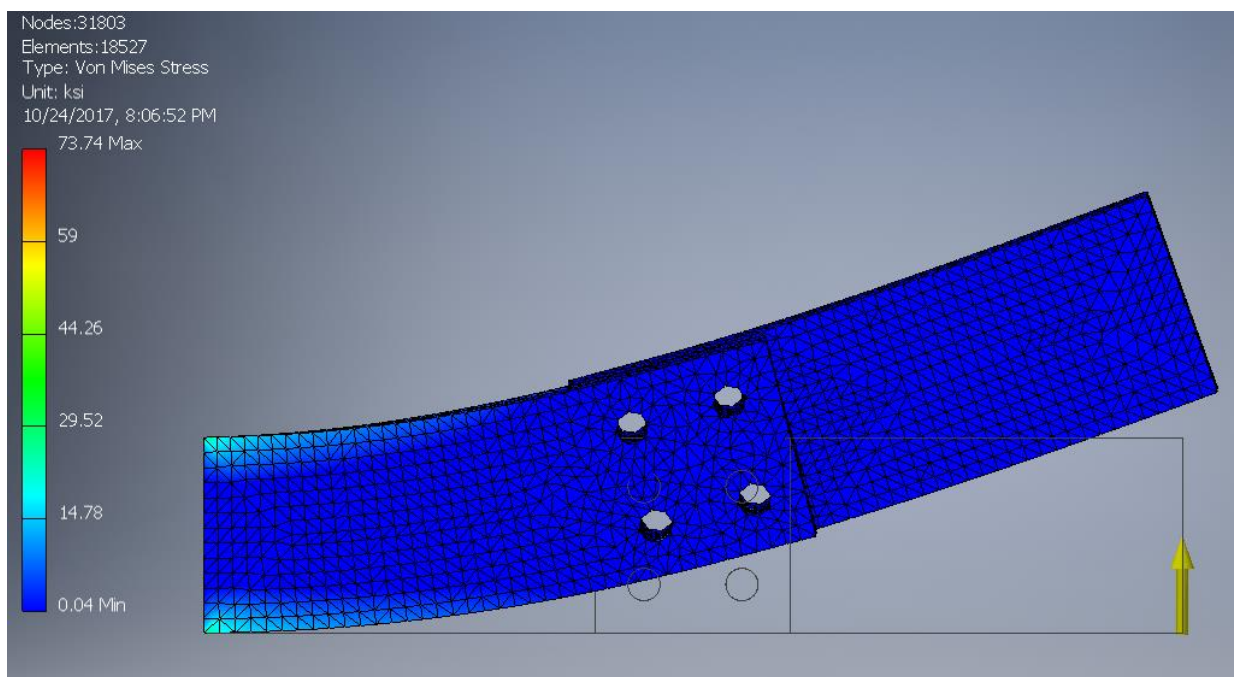
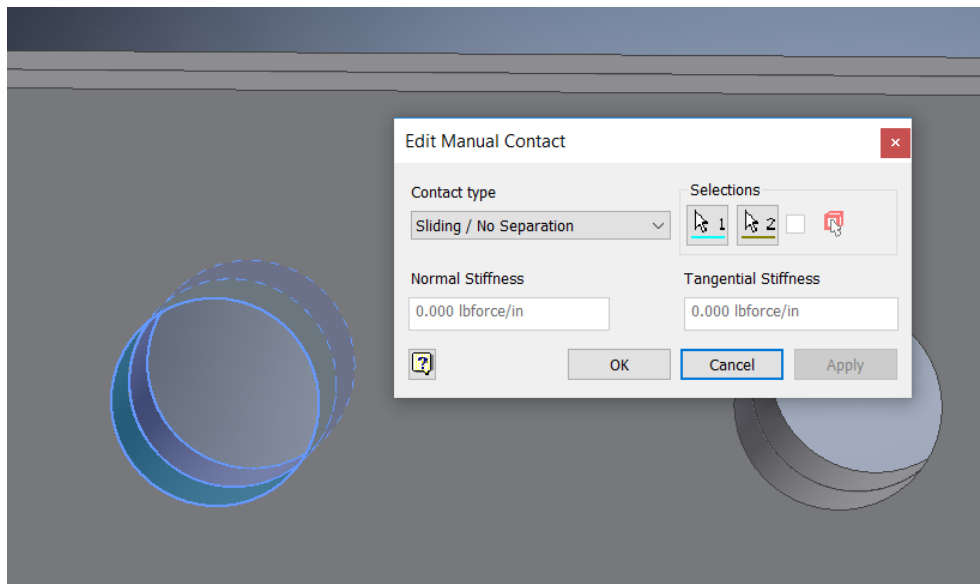
Accounting for Bolts and Welds

Bolts are placed in models to show how you are going to connect two parts together; the same can be said for welds. But these two design elements can cause meshing issues; depending on the level of detail. They can have very small features which will create small mesh elements and the opportunity for stress singularities.

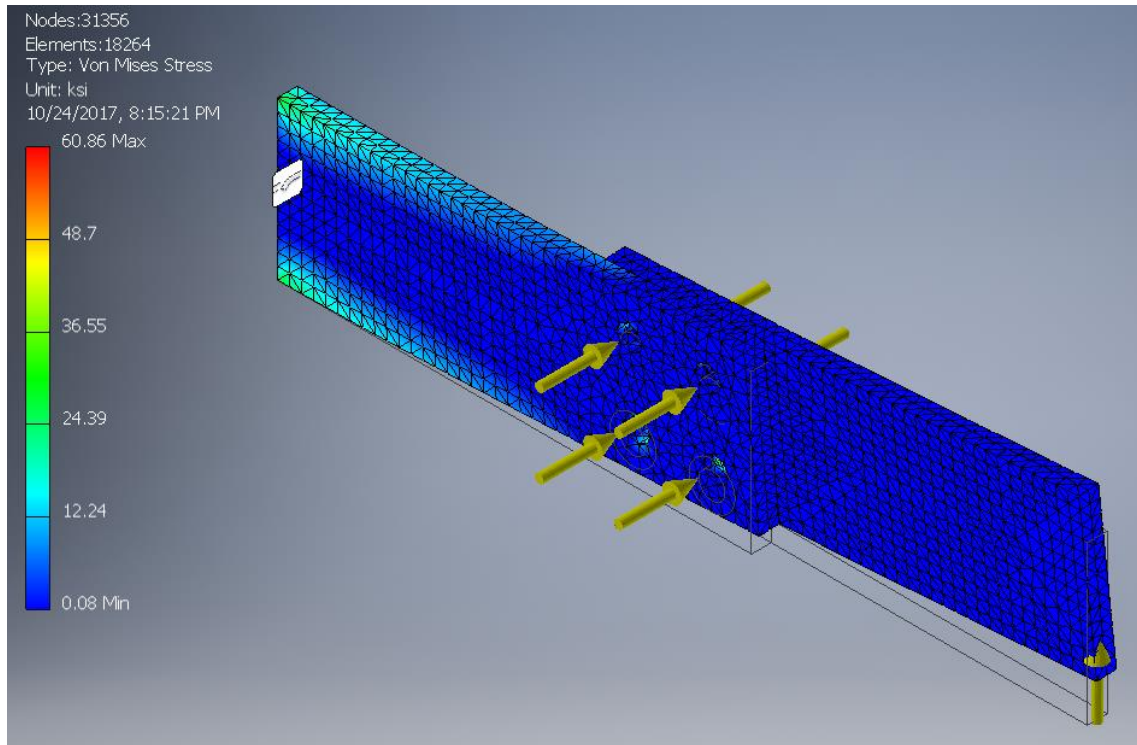
Nastran In-CAD and Fusion 360 Ultimate have special mesh features that can replace bolts in your model with beam elements, which is an efficient method to analyze loads in each bolt, it also allows for the application of pre-tensioning of the bolt. Andrew Sartorelli had a presentation last year at AU 2016 called [“Let's Get Together: Using Connectors in FEA”](#) that discussed how to use these tools.

If you are using Inventor Professional or Fusion 360, then you must be more clever. I have found that by removing the bolts and nuts, I can use the sliding no separation contact on the

inside of the bolt holes and on the contact surface to replicate the behavior of a bolted connection.



If you want to approximate a pre-tensioning of the bolt, you can split the surface where the bolt head and nut would be and apply a force to replace the torque.



Now this method will not tell you what the forces are in the bolts, like the beam elements do, but it will help replicate the behavior of the connection.

FEA should not be used to analyze welds, Vince Adams did an entire presentation on this last year at AU 2015 called [“A Job Weld Done”](#) so I will not go into great depth on this topic. But where a weld is to be placed, make sure you have a good contact between those two parts, and you can model in the weld with proper contacts. From the results of your analysis, it is recommended that you use standards and codes, like AWS, to determine the adequacy of the welds in the design. There is also a very good weld calculator inside of Inventor that can help you analyze and size welds.

Section Summary

There are several tools that can be used to help improve your mesh, and therefore your results.

Split off areas on your model to apply focused constraints, loads, and mesh detail.

Use the mesh setup tools within the simulation environment to create enough elements to model behavior. Also, use the minimum element size and growth factors to promote a uniform mesh. Limit the use of local meshes created with the mesh tools until after convergence, since the element size is held static with a local mesh.

Use the built in tools to replace bolts in the analysis model. If these features are not available in your tool, utilize a primitive shape model or contacts to replicate the behavior of these connectors.

Apply the knowledge from this class to your workflows for upfront simulation

The topics covered thus far provide techniques on how to more effectively in the use of FEA. Now we will discuss how to take these and apply them to your workflow.

As has been discussed, a simple analytical model is the easiest model to work with. While your application may not allow for a simple model, efforts to focus your analysis and simplify your model will be rewarded with accuracy and time. Focus on what you are asking the analysis to answer for you and build your CAD model based on this, using lines and surfaces for beam and plate elements, respectively, in your analytical model.

- **An early design stage and simple analysis model will help test theories and provide valuable insight as the design progresses.**

Result Validation

To make sure that you have a better model, a better mesh, and you are getting better results let's discuss model validation. Checking convergence of your results is important and should always be performed. The use of hand calculations based on accepted engineering methods, to check your analysis results is also an important step in the validation of your analytical model. But there are two validation checks that I perform, one is qualitative and the other quantitative. The qualitative method is based on the fundamentals of Finite Element that it solves for displacements at the nodes. (The Appendix has a simple example of the theoretical development of the FEA equations for a basic rod element.)

Before you run the analysis, make an assumption about the approximate displaced shape, and use this assumed deformed shape to compare to your analysis deformation results.

- **Make sure the displaced result correlates with what you expected. If the displaced shape is not what you expected, ask yourself why.**
- Are you using an element type that can support the expected deformed shape?
- Are the number of elements adequate to generate a proper deformation?
- Are the elements properly sized and shaped to not create elements that are too stiff and generate inaccurate large displacement and stresses?
- Was your deformed shape not correct, is there something occurring you did not expect?
- Is the model not constrained properly?

The displaced shape is the first qualitative verification of your model, as you can quickly make a judgment on the accuracy of the analysis by just comparing the assumed and calculated deformed shapes.

The first quantitative check of the analysis is to check your reactions. Going back to statics, do they satisfy static equilibrium and are they what you expect them to be. I have had analytical

models where reactions were being generated in locations I did not intend them to be and this caused me to go back and refine my model so I could get results that I could work with.

- **The Autodesk FEA products all have ways to get the reaction values, use these values to check for static equilibrium and verify your assumptions.**

Also of great importance is that the reaction forces work for how the part or assembly under analysis is attached to the rest of your product or application.

- **If the reactions from the analysis are larger than what the connection can handle, the model will not behave the way you assumed and your results will be invalid.**

It may be that you need to strengthen your connection, for instance that you use larger or more bolts. But keep in mind that changing the connection, including within the analytical model, will have an effect on the stiffness. And as we know, an increase in the stiffness will affect the displacement and thereby the stresses.

Application to more complex analysis

We briefly discussed that you should also use the proper analysis type for the question you are asking, there is one caveat to this rule. If you are going to perform an advanced analysis, start with a linear static analysis and use the methods described above to make sure the analytical model is behaving the way you expect. In my consulting, I take this even further, even though it may take more time in the beginning, it boosts my confidence in the model and saves me time overall.

- **Start with a unit load analysis, without gravity, to make sure that the model is behaving properly and you have a mesh convergence, calling this the datum analysis. Then proceed with your analysis and what if scenarios with more confidence in your results.**

Once you have your datum model verified, and you have run what if cases, you can add more detail to the model. If changes are to be made to the design, you can rely on your analysis results to provide insight on how to make these changes.

As the design progresses, you can always do subsequent analyses trying to focus on areas of concern to validate your design. But as the design model develops a more complicated analysis could be required.

- **Even with large and complex models, your early analysis results and these tips can help validate your analysis and improve your efficiency.**

Appendix – Finite Element Theory Example

To better understand how the Finite Element Method works within the software, the following basic theory example is provided. Consider the following simple bar that is constrained on each end.



This bar is assumed to be weightless and positive is considered to be to the right.

At each end, Point 1 and 2, there is a possible Load, P_1 and P_2 , respectively. There is a chance for Displacement at each one of the ends, Δ_1 and Δ_2 , also respectively. If we assume a Cross-Sectional Area denoted as A , length, L , and the Modulus of Elasticity, E , we can use the following equation for engineering mechanics for the bar.

$$\Delta = \frac{PL}{AE}$$

If we take the Load, P , out of the fraction this equation becomes:

$$\Delta = P \frac{L}{AE}$$

Dividing both sides by L/AE :

$$P = \frac{AE}{L} \Delta$$

The expression of AE/L for the case of the bar is known as the stiffness. The stiffness is critical in the Finite Element Method, as it is used to relate the finite elements to one another.

Close relative stiffness between finite elements is one of the ways you can get more accurate results, and we will cover that throughout this class.

Taking this stiffness focused equation, we can write specific equations for each end.

$$-P_1 = \left(\frac{AE}{L} \right) (\Delta_1 - \Delta_2)$$

$$P_2 = \left(\frac{AE}{L} \right) (\Delta_2 - \Delta_1)$$

Notice that there is a negative sign in front of P_1 , this is because that Load is assumed to be acting in the opposite direction of the displacements and the other Load, P_2 .

The Displacements are subtracted from each other because their value is dependent on the value of the other. The value of Displacement at Point 2 is affected by the Displacement at the other end, so we need to subtract the Displacement at Point 1.

Now we can multiply these two equations out:

$$\begin{aligned} -P_1 &= \frac{AE}{L} \Delta_1 - \frac{AE}{L} \Delta_2 \\ P_2 &= -\frac{AE}{L} \Delta_1 + \frac{AE}{L} \Delta_2 \end{aligned}$$

Writing these in a matrix format:

$$\begin{Bmatrix} -P_1 \\ P_2 \end{Bmatrix} = \begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix}$$

To simplify this linear algebra expression, we can assign the following identifiers.

Force vector:

$$\{p\} = \begin{Bmatrix} -P_1 \\ P_2 \end{Bmatrix}$$

Stiffness matrix:

$$[k] = \begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}$$

Displacement vector:

$$\{d\} = \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix}$$

Using these identifiers we can rewrite the linear algebra expression above in a more simplified manner.

$$\{p\} = [k] \{d\}$$

This is the fundamental equation of the Finite Element Method on an elemental level for stress analysis. What makes FEA so powerful is that it can take this expression for each element and combine it to create the global stiffness expression.

$$\{P\} = [K] \{D\}$$

Simply put, the Finite Element Method then takes the inverse of the global stiffness matrix and multiplies it by each side.

$$\{D\} = [K]^{-1} \{P\}$$

Which determines the displacement at each node, from the displacement we can determine the strain in the element.

$$\varepsilon = \frac{\Delta}{L}$$

Using Hooke's Law, the strain can then be used to determine the stress.

$$\sigma = E\varepsilon$$

This is on a simple level, how the Finite Element Method determines the stress in the elements. What is most important is that the stresses are calculated from the displacements determined. If your displacements are not accurate, then your stress results will not be correct. We will come back to this point again.