AUTODESK UNIVERSITY 2014

# Introduction to FEA with Autodesk Nastran In-CAD

Mitch Muncy - Autodesk

**SM6675** This class will be an introduction to the finite element process. Taught through Autodesk Nastran In-CAD, it is intended to provide students with the essentials needed to solve mechanical analysis problems with confidence. Learn tips and tricks for solving efficiently and how to avoid common mistakes.

## Learning Objectives

At the end of this class, you will gain:

- Confidence in your analysis results
- Best practices in Finite Element Analysis modelilng
- Deeper understanding of Autodesk Nastran In-CAD

## About the Speaker

*Mitch is a simulation enthusiast with over 15 years in the CAE industry. Before joining Autodesk as a Product Manager, he was Executive Vice President of NEi Software (NEi Nastran) where he managed day-to-day operations of the sales, marketing, and technical teams. Mitch holds a degree in Mechanical Engineering from the University of California, Irvine.*

*Mitch can be reached at mitch.muncy@autodesk.com*

## Introduction to Finite Element Analysis (FEA)

When used correctly, Finite Element Analysis (FEA) can provide tremendous insight into the behavior of structures and components before they are even assembled in the real world. Class leading companies are taking advantage of FEA today to shorten development cycles, save money on costly prototypes, and get to market before their competition.

**Benefits**

Often, we will receive calls from companies looking for stress analysis software or consulting. Almost always, a failure has already happened in the field. This is the most expensive way to use finite element analysis. The desperation and urgency can be heard in the customer's voice. We will always ask, "When do you need it by?", but we already know the answer. "Yesterday."

A study published by the Aberdeen Group in October 2006 found that class leading companies are saving money by cutting down the number of costly prototypes. These savings ranged anywhere from ten thousand dollars (USD) for simple assemblies of less than 50 parts, all the way up to two million dollars complex assemblies of over a thousand parts.

| Product Complexity | Number of Parts | Length of Development | Cost to Build Prototype | Average Cost Savings |
|---|---|---|---|---|
| Low | Less than 50 | Between a week and a year | $7,600 | $12,160 |
| Moderate | Between 50 and 1,000 | Between a month and 5 years | $58,000 | $92,800 |
| High | Between 50 and 10,000 | Between 1 and 5 years | $130,000 | $208,000 |
| Very High | Between 1,000 and 100,000 | Between 1 and 20 years | $1,200,000 | $1,920,000 |

Reducing the number of prototypes also shortened the development cycle.  For small assemblies, this could be as much as 21 days for a simple project and up to over 5 months for a more complicated project.  This is how most FEA is utilized today.

| Product Complexity | Number of Parts | Length of Development | Time to Build Prototype | Average Time Savings |
|---|---|---|---|---|
| Low | Less than 50 | Between a week and a year | 13 days | 21 days |
| Moderate | Between 50 and 1,000 | Between a month and 5 years | 24 days | 38 days |
| High | Between 50 and 10,000 | Between 1 and 5 years | 46 days | 74 days |
| Very High | Between 1,000 and 100,000 | Between 1 and 20 years | 99 days | 158 days |

Even better, FEA can be used to drive design. As engineers, we're often asked to design within an envelope. We limit the stresses to a certain value, keep displacements in check and try to reduce costs. With computing, we can push beyond that mentality and seek out the optimal design from the start.

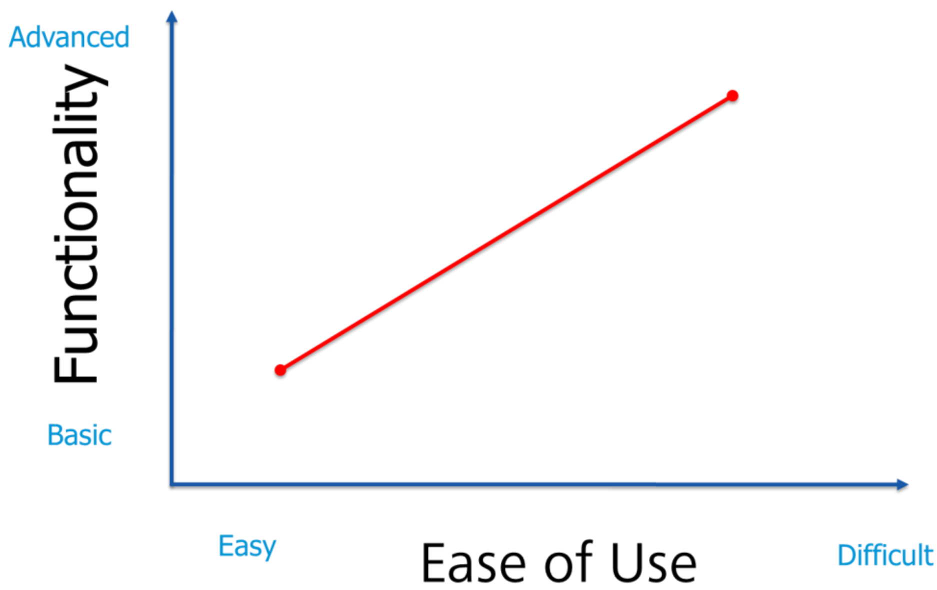
**Fears**

For companies not currently using FEA there are typically a few common barriers to adoption. First is the fear that FEA is difficult to use. Like all tools, there is a ramp up period and typically project schedules are tight enough as it is. Second is the concern that it is expensive. Not only in the upfront purchase cost, but annual maintenance, and the manpower required to run it. Third is a concern for the accuracy of the results. Since it would be a new technology for those who are adopting it, they haven't built the understanding and trust that other companies have.

***Difficult***

When we learn about the finite element analysis process in school, it can be a little intimidating. If you're new, topics like discretizing the structure, assembling the stiffness matrix, and choosing the proper stress, strain and displacement functions are confusing. The thing to remember is that no matter how complex the process the solver goes through to get you the right answer, the FEA process is very straight forward.

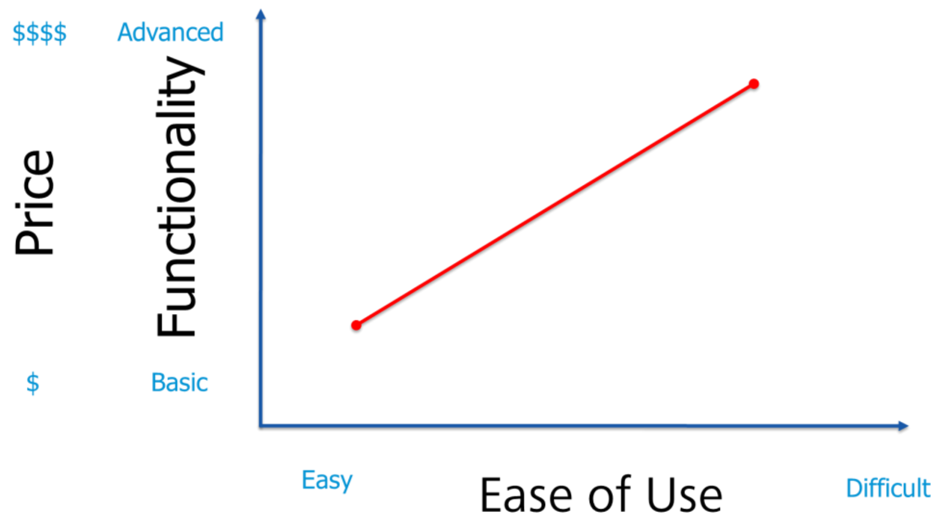

When researching finite element analysis software packages, there are quite a few to choose from. In general, they tend to follow the curve highlighted in the chart below.




If ease of use is a concern, introductory packages are available. The features are typically limited which in turn makes the software easier to use. These typically come with assumptions and methods which are very common. These are a great starting point for an introduction to FEA.

If needs for additional capabilities arise, more comprehensive packages can be found.

***Expensive***
Just like the previous chart above, one can also be used to compare the price for a FEA package:




Price also varies with the functionality. There are a large number of open source finite element analysis programs out there which can be used for free. These may be limited to the specific application for which they were developed. Also, the usability and workflows tend to have one person in mind, the original developer. Also like price, introductory products can be used until more complexity is required. CAD embedded FEA tools are generally on the order of $10,000 and can immediately provide ROI on the very first project.

***Accuracy***

One question that people who are new to FEA will always ask about the accuracy of the software. They are typically concerned that the software will not be able to accurately model the real world performance of their products. Generally speaking, chasing the most accurate FEA tool is like chasing Bigfoot or the Loch Ness Monster. Every FEA package can give the right answers provided they are being used properly.

The most important way to improve accuracy is to understand your goals when using FEA. If you try to model all the real world scenarios that your products will see, analyses can be very complex and prone to errors. Most companies will adopt methods and best practices for analysis of their products. These are common ways to identify whether a part will fail or not before it is tested. They make assumptions about the analysis based on experience in designing, building and testing their products. One such case is using acceleration to represent dynamic loads. The thought is that if the product can survive a 6g load for instance, it should be able to survive the dynamic loading experienced in flight.

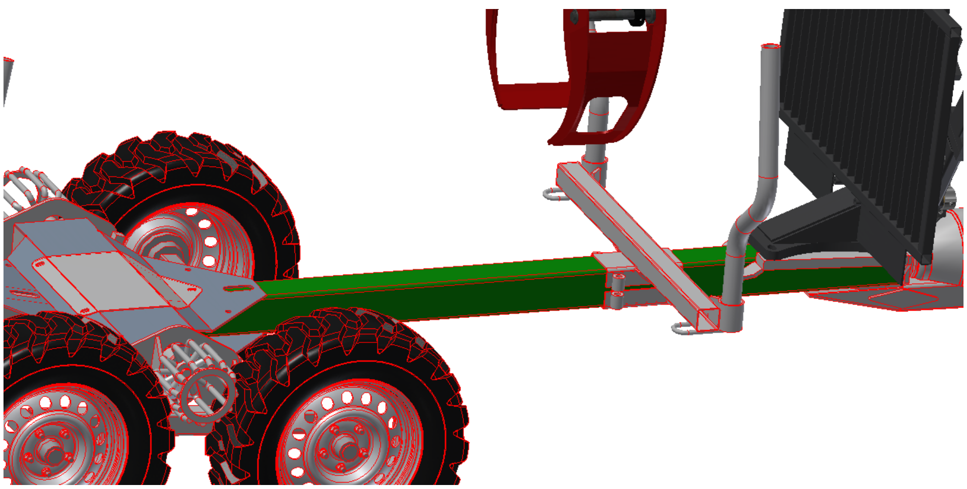
That being said, there are some general tips that can be used to improve the accuracy of FEA. Later we will discuss choosing the right materials and properties, meshing, and element quality.

## Introduction to Autodesk Nastran In-CAD

Autodesk Nastran In-CAD, a general purpose finite element analysis (FEA) embedded in your CAD system, is powered by the Autodesk® Nastran® solver and offers simulation spanning across multiple analysis types such as linear and nonlinear stress, dynamics, and heat transfer. This single product is available as a network license and serves both Autodesk Inventor and SolidWorks – providing a consistent user experience and eliminating the need for multiple single-platform simulation technologies. It delivers high-end simulation technology in a seamless CAD-embedded workflow so you can make great products. More information on In-CAD can be found on the web.

### Sample Model

To help illustrate some of these points, we’ve selected a very simple model. We want to analyze one of the main beams of our logging equipment.



Just looking at a model like this, we can start to imagine some of the different things we need to think about before we begin our analysis.  What type of analysis do we use?  It can be static for understanding service loads and gravity or dynamic/transient if it is navigating difficult terrain or if logs are dropped into it.  Should we use solid elements or shells or beams?

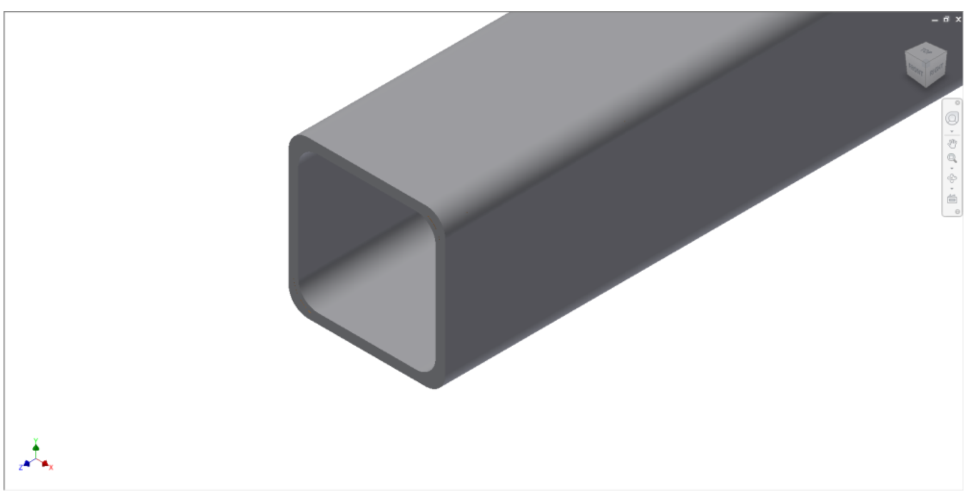
Since our goal of this FEA is to the influence element selection plays, we will keep it very simple. We will assume the only case we're concerned with is the service load of 2 tons. We'll also assume that the loading is static and evenly distributed across the beam. We will compare the results for solid, shell and beam elements.

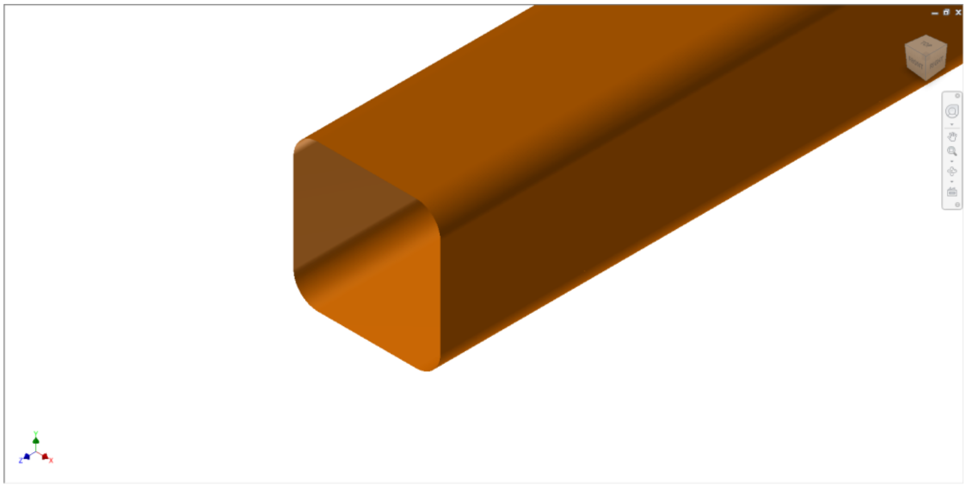
**CAD Geometry**

As we highlighted before, the first step is to build or import the geometry. Geometry is typically generated in the CAD system and transferred to the FEA tool if needed.

***Idealization***

Probably the most important concept of finite element analysis is idealization. It is the process of deciding how to take your CAD model and translate it to the ideal mathematical model. We make decisions about idealization during many points in the analysis. If we decide we want to use shell elements, we'll need the surface geometry available to support it.



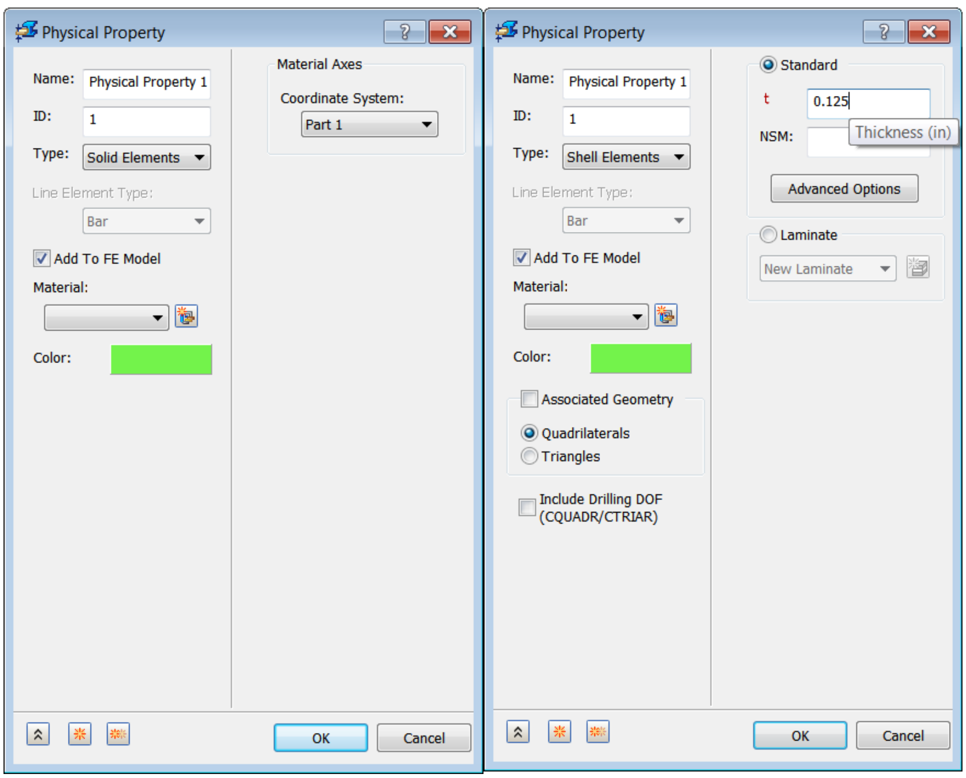


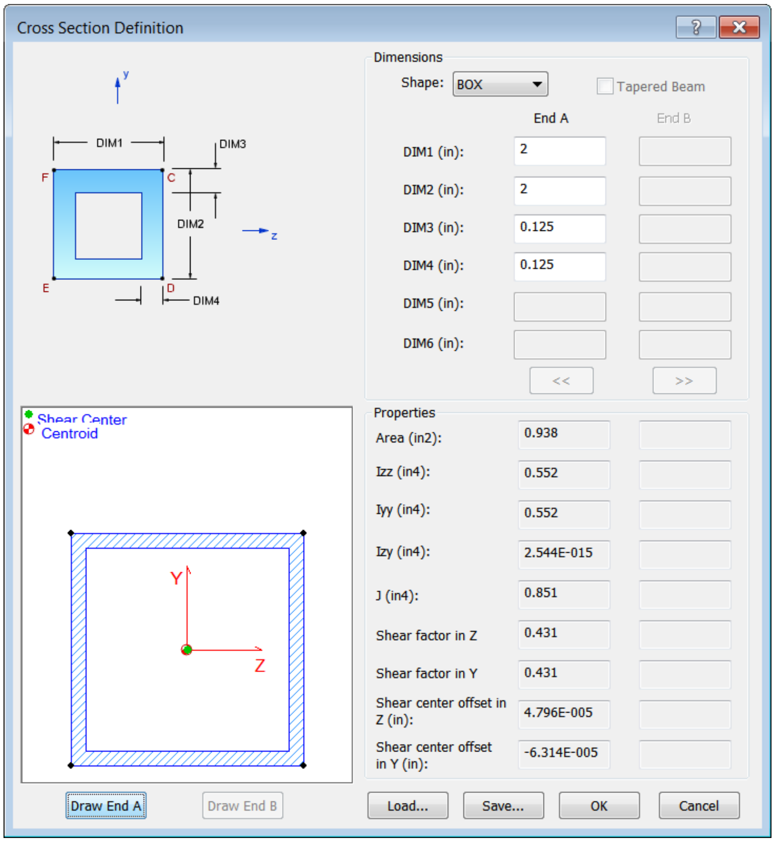
***Simplification***
The goal of simplifying your model is to remove features which will have negative effects on the solution time or mesh. The trick is trying to identify which features can be safely removed without changing the quality of the results. Typical examples are fillets and small holes. We want to accurately capture the mass and stiffness of the structure while keeping mesh count at a minimum. It will also help improve the quality of the mesh.

**Materials and Properties**
As alluded to before, selecting the right materials and properties will heavily influence your results. Material models can range from simple isotropic for most metals all the way up to shape memory alloys with unique hysteresis loops. The trick is to make sure you capture the behavior you are looking for in your analysis. For our model, we will import a steel material from the material library and use an isotropic material definition.

## Idealization

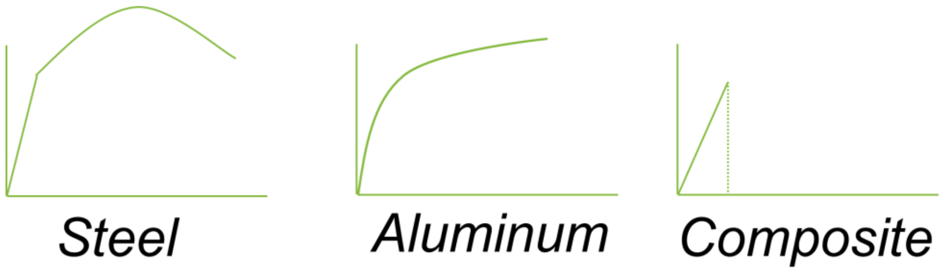
Physical properties determine our mesh types, so it is another location where we are choosing our strategy for idealization.  Our physical properties will allow us to specify features of our model which aren't included in the geometry.  Some examples of this are shell thickness or beam cross sections are highlighted below.  For our example, we will compare solids, shells and beams.



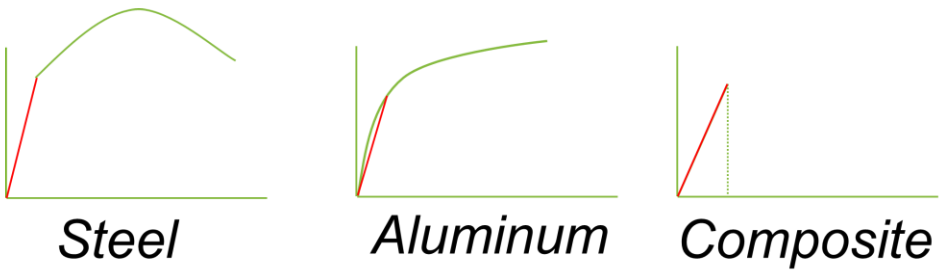



## Material Models

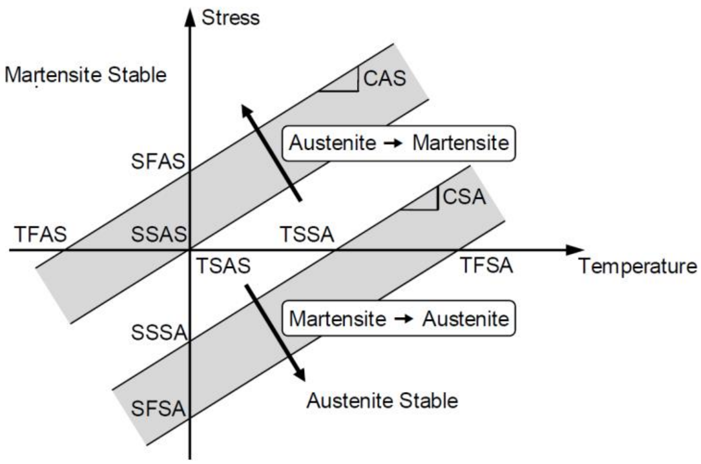
In theory, all materials can be considered nonlinear.

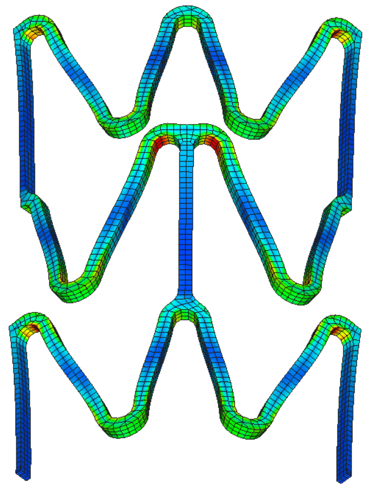



Even composites exhibit failure. However, when we perform a linear static analysis, we assume the material only operates in the linear range, or the stress/strain curve can be assumed to be linear.




Some material models all but require nonlinear analysis. For applications such as biomedical stents and deployable space structures, Nitinol material model may be used to obtain shape memory effects and superplasticity. For architectural applications, concrete material models may be used to simulate the cracking behavior in tension and crushing behavior in compression.



**Constraints**
Constraints allow us to control how part of our model moves. It works by preventing movement in one or more directions. Constraints can be applied to geometric surfaces, edges, or points, but when it is written out to the solver, it is applied at the node or group of nodes.

***Fixed vs. Free***
When a node is fixed, it means that all its degrees of freedom (ways it can move) are constrained. For this, it is important to understand the number of degrees of freedom nodes on different elements have. For instance, nodes on solid elements only have three degrees of freedom (translation in x, y, and z), while beam or shell elements have six degrees of freedom per node (translations and rotations).

***Pinned***

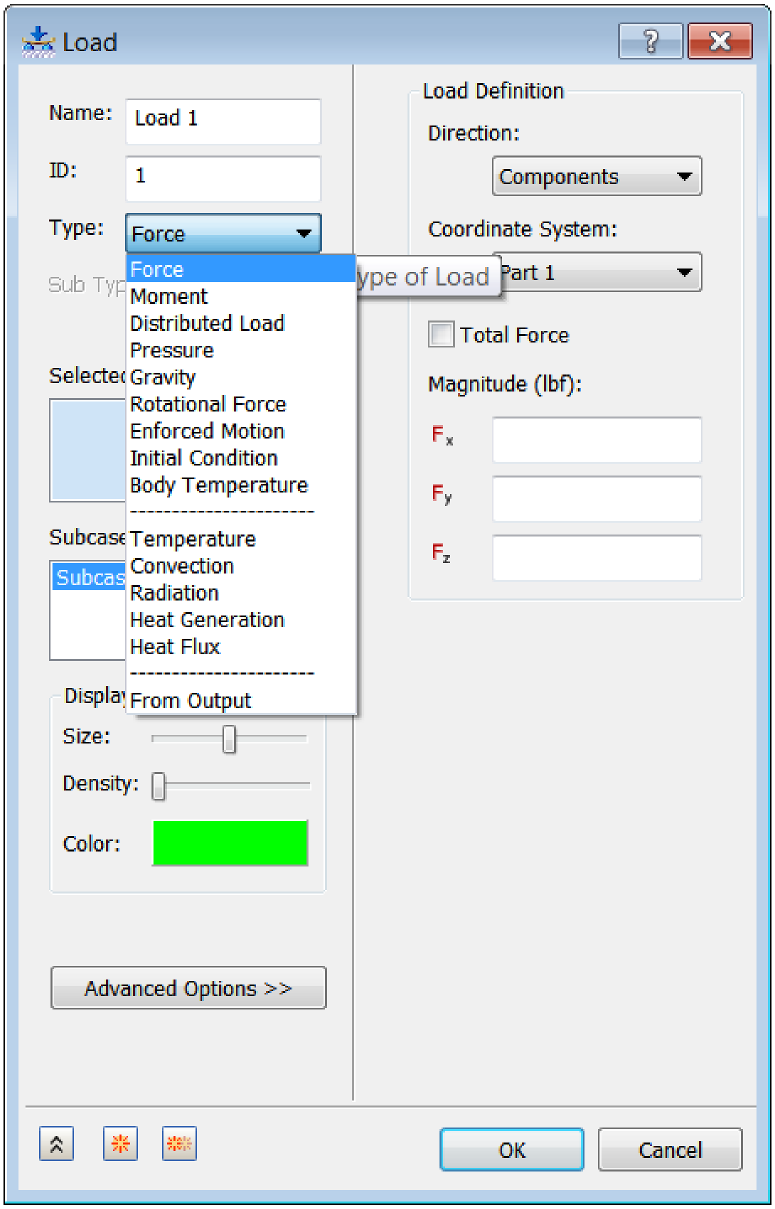
The term “pinned” is used to describe a constraint on only the translational degrees of freedom. The idea is that rotations are allowed. However, when applying constraints to a group of nodes, how they interact also must be considered. For instance, applying a pinned constraint to the (solid) surface of a hole is the exact same thing as creating a fixed constraint.

***SPC Forces***

In Nastran terminology, the longer name for a constraint is a Single Point Constraint. One good check for a model is to investigate the “SPC Forces” which are the reaction forces at the constraints. This will tell you how the load is being reacted and where and when summed can make sure that everything is balanced.

***Loads***

There are a wide variety of loads that can be applied in a model. They can be applied to points, curves or surfaces and eventually at solve are translated to nodes and elements. One common mistake is to apply a force load to a single point. This is the same as applying your force with an infinitely stiff, infinitely thin needle. As expected, there are high stresses at the point of application.


Load
Name:
Load 1
ID:
1
Type:
Force
Force
Moment
Distributed Load
Pressure
Gravity
Rotational Force
Enforced Motion
Initial Condition
Body Temperature
Temperature
Convection
Radiation
Heat Generation
Heat Flux
From Output
ype of Load
Load Definition
Direction:
Components
Coordinate System:
Part 1
Total Force
Magnitude (lbf):
Size:
Density:
Color:
Advanced Options >>
OK
Cancel

### *Total Loads*

One caution is to check the total load that is being applied.  Sometimes there is confusion when applying a force load to multiple surfaces.  For instance, if we apply a 4,000 pounds load over two surfaces, is that 4,000 pounds per surface (8,000 pounds total) or 2,000 pounds per surface?  Most software packages will allow you to specify whether you are applying the load as a “total load” or not.

### *Loading Functions*

Loads can also be applied as a function of time, frequency, temperature, or spatial location. Accompanying analysis types (transient or frequency response) is required for time or frequency dependent loading.  Hydrostatic pressure loads are a good example of a spatially varying load. For the water inside of a tank, the pressure varies linearly and can be interpolated as such. Since loads can vary spatially, this means you can bring in data from other sources such as CFD or Thermal analyses.

## Mesh

### *Idealization*

Selecting the right element types for your mesh is the key to efficient finite element analysis modeling.  Choosing between solids, shells and beams is a balance between pre-processing and the effect that it has on the results.

### *Mesh Size*

Selecting the correct mesh size is also a balance. It balances the ability to accurately model the geometry, ensure high quality elements, and solve the analysis very quickly. Fewer elements are required to get accurate deflections. More elements are required for accurate stresses.

### *Element Quality*

FEA solvers assume that elements are all the perfect shape. In the instance of a shell model, all the sides are equilateral, there is no warping (any nodes out of the plane of the element), and all the angles are 90 degrees. Any deviation from this perfections impacts the results. Most solvers will have warnings for poorly shaped elements and many pre-processors will plot the various measures for element distortion.

## Post-Processing and Results

Commonly referred to as post-processing, investigating results and preparing reports is done once the analysis is completed.

### *Wide range of Output*

Looking at Autodesk Nastran specifically, not including individual ply results, there are 814 output vectors to choose from. These vary depending on nodal, element type, and analysis type. The expansiveness comes from the various requests from customers over the years. Most companies develop methods for post-processing just like they have done for every other step in the finite element process. For linear static analysis of isotropic materials, common outputs are displacements and von Mises stresses and strains.

***Determining Your Limits***

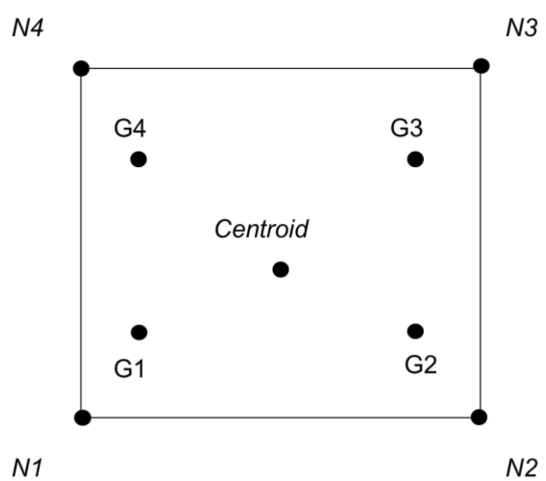
The real question that we are trying to answer with all the various output we receive is what is ultimately determining the failure of the part. Some companies will design to the yield of the material or a factor of safety tied to that. Others will try to keep stresses below the endurance limit for the material. We can also run nonlinear analysis with yielding, or fatigue studies to get more details about the failure of our models.

***Overview of Output Calculations***

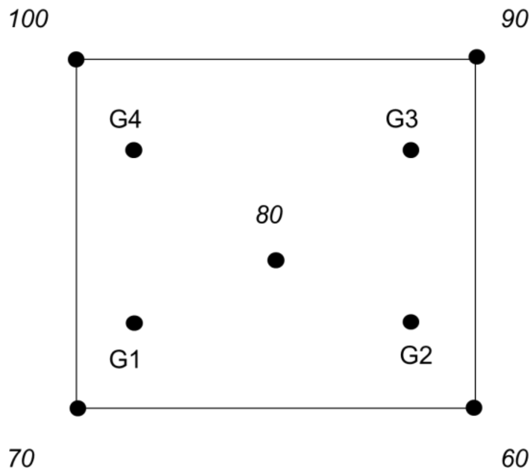
For things like forces, displacements and other values which can be calculated at a point, the output is reported at the nodes. Other values like stresses and strains are calculated on the elements. It is important to understand how these values are calculated.

Finite element analysis codes, really calculate these values at Gauss points. Since we typically are looking for output at the element centroid and at the nodes, we interpolate. Nastran can extrapolate to get “nodal” stresses, which are calculated independently for each element.

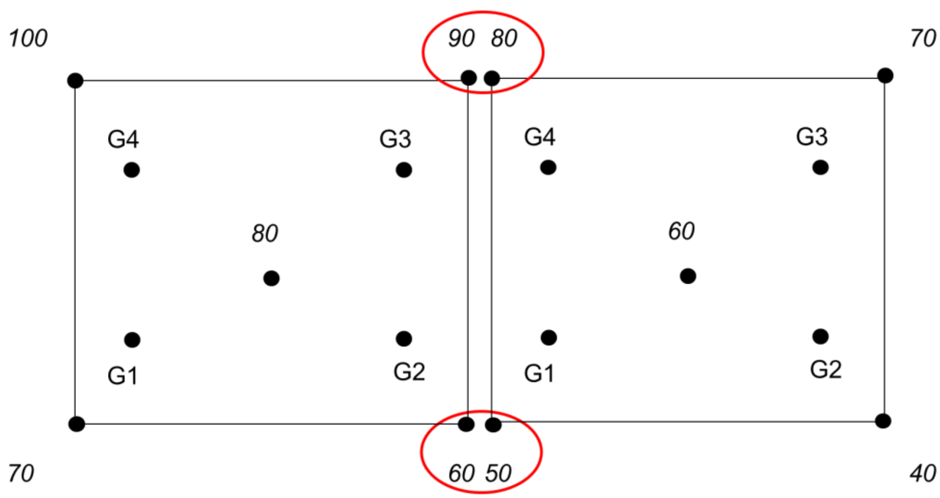
Let’s look at a shell element as an example.



Results are reported at element corners and centroids like this:



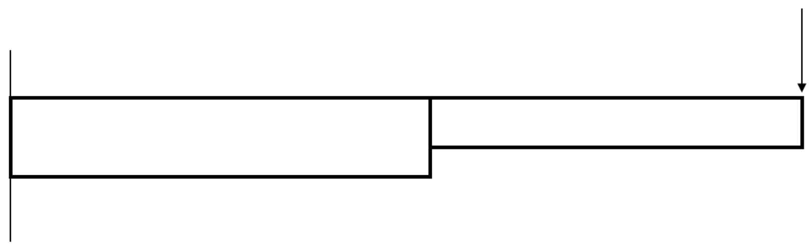

When two elements are connected together, we can start to understand how the post-processor needs to interpret the results.

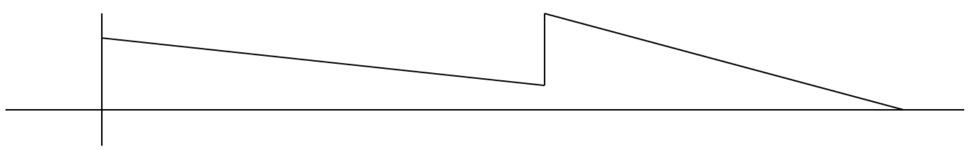


At our connected nodes, there are two different answers. When more elements are connected, there can be even more output. To come up with our “nodal” value, we need to come up with a best guess. There are a couple different approaches we can use.

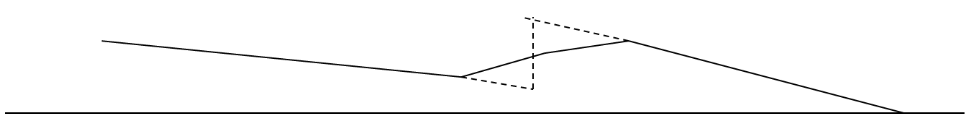
First, we can average the corner values. This is the typical default value used by most post-processors and ensures nice smooth results.

Another method which is used is calculating the maximum value. Since most of the time in FEA we are looking for the conservative answer, this will give the maximum calculated stress in that location. In our example, what would happen if the two elements had different thicknesses? You would expect a stress distribution like this:





However, if we use elemental averaging, we would get something more similar to this:



The peak stresses are in the wrong location and have the wrong magnitude. Most post-processors will allow you to change how results are averaged so that you do not encounter these types of issues.

## Analysis Types

We've saved the best for last, although selecting our analysis is typically done as the very first step. To make sure we're selecting the right one, we need to consider what kind of physics are influencing our results. That being said, we can make some assumptions and limit our analysis type accordingly.

### *Linear/Nonlinear*

The decision between linear and nonlinear analysis can be a course all by itself. Typically we are looking for a few factors when considering nonlinear analysis. This includes large displacements, follower forces, nonlinear materials, and nonlinear effects like contact. Sometimes the easiest way to determine if a model requires a nonlinear analysis is just to run it.

### *Modal/Dynamics*

Another great example is modal analysis. It allows you to find the natural frequencies and see how the part or assembly will perform. Output like the modal effective mass will show how much of the model responds in each direction for each mode. This helps identify the key frequencies to avoid for our loading. The mode shapes displayed are scaled by mass and the values we get for displacements are just for information. They do not have any real world meaning other than the scaled mode shapes. To get more details, we would need to run frequency or transient response. Also included in advanced dynamics are things like random response (PSD/ASD) and complex eigenvalues.