

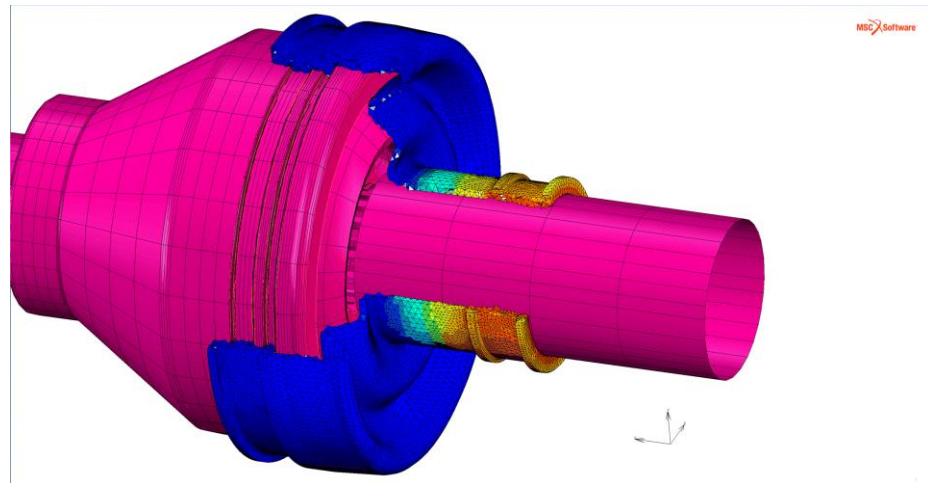
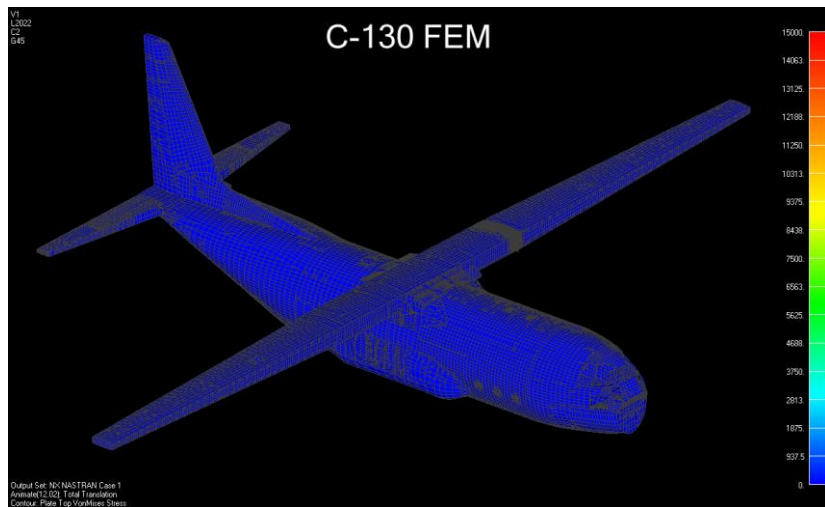


An Introduction to FEA

Jackson Talbert and Nick Garber



What is FEA?



Finite Element Analysis is a numerical approximation of how a structure will respond to a given environment.

Watch this [video](#) if you're new to FEA!



When should you use FEA?

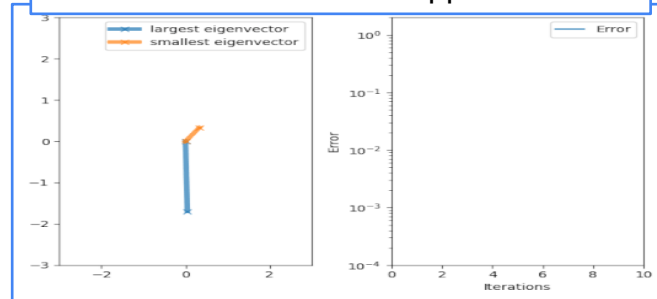
Numerical approximation: An approach to coming up with an approximate answer to a problem, whether its through **discretization** and/or an **iterative solver**.

Closed Form analytic solution: Finding a solution to a problem using a generally accepted set of **equations**. Much more accurate, and often empirically based!

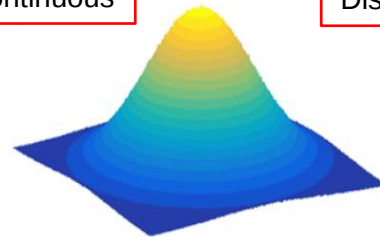
FEA is a numerical approximation and has inherent limitations on its accuracy. It should be used only when no analytic solution exists!

Analytic
Solution

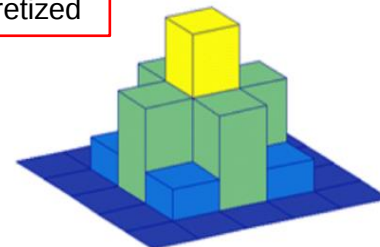
Power Method Iterative Approximation



Continuous



Discretized



The maximum in-plane load at which dimpling of the sandwich face sheet occurs, shown in Figure 14-7, is given in the empirical expression:

b_p typical potting radius;

t_s face sheet thickness;

E_s Young's modulus of face sheet;

ν_s Poisson's ratio of face sheet;

Sc core cell size.

KD dimpling coefficient; which depends on the plate geometry, boundary conditions and type of loading.

$$Q_d \leq \frac{2}{\pi} b_p t_s K_D \frac{E_s}{1 - \nu_s^2} \left(\frac{t_s}{S_c} \right)^2$$

[14.3-7]





Simulation Basics

This slide deck will cover a few main points important to becoming proficient with FEA.

Planning the analysis

- What information are you trying to get from this model and how will you get it?

Discretization

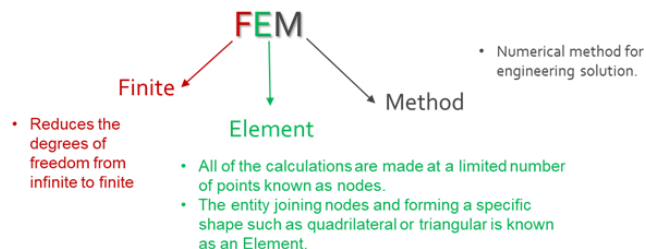
- How would you like simplify this model to make it solvable?

Boundary Conditions

- How will you model the environment that this part operates within?

Post Processing

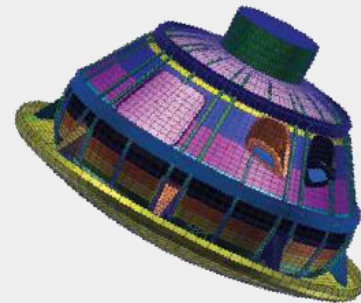
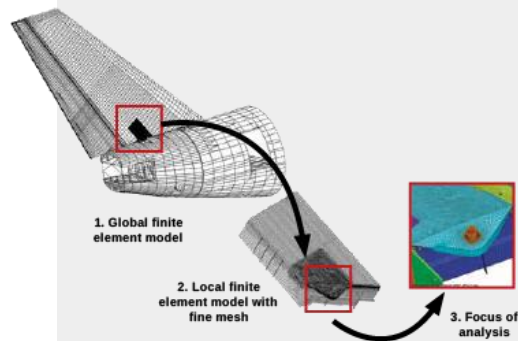
- How can you interpret and use the results?





Planning The Analysis

What are your goals?



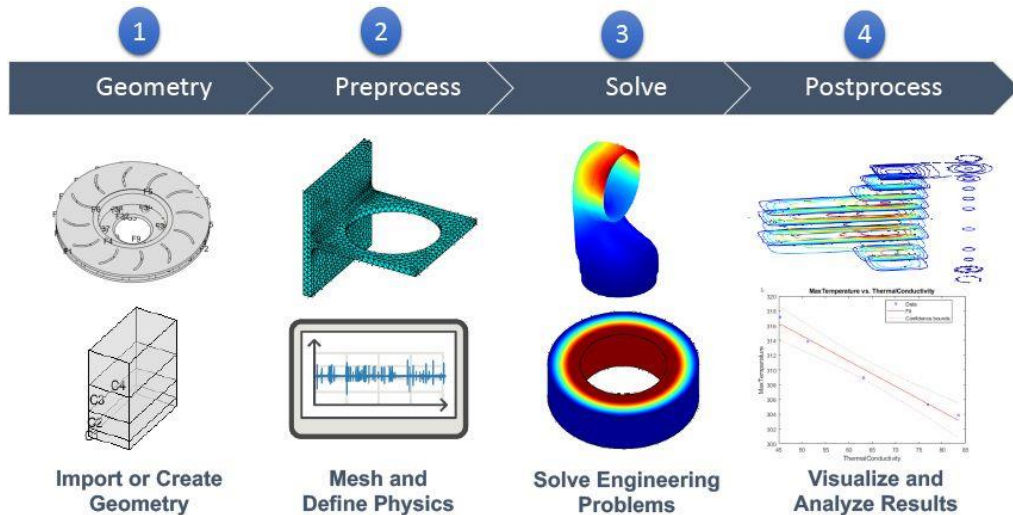


Planning The Analysis: Overview

FEA is split up into 4 phases shown to the right.
FEA software commonly consists of a pre-processor (idealization and meshing), solver (software for solving the model), and post-processor (visualizing the analysis results)

The first thing you should do when planning your FEA, is decide what information you want out of the model. Depending on that decision, you can decide how to set up your model.

For example, if you just want to predict the stiffness of a part you can generally use a simpler representation than if you want to predict the stress within a part.





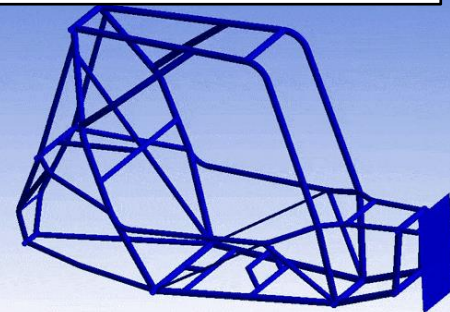
Planning The Analysis: Solution Types

FEA solvers commonly have many “solutions” to chose from. This allows the analyst to specifically chose what information they want out of a model.

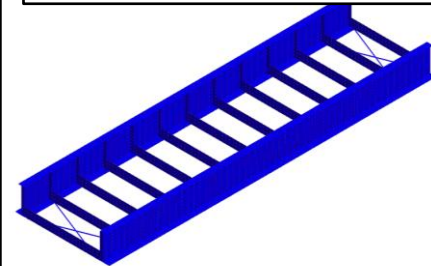
Linear vs. Non-linear	Linear: A linear solution assumes small displacements, and that your materials are operating exclusively within their elastic range. Non-linear: A non-linear solution allows for the modeling of large displacements and material behavior outside of the elastic range, but it is more computationally intensive.
Static vs. Dynamic	Static: A static solution assumes all loading is occurring at a steady state and is accurate for loading that occurs slowly and consistently. Dynamic: A dynamic solution is able to solve for the response of a structure to a dynamic load, for example an impact or vibration. These solutions are typically very computationally intensive.
Solution Examples	Buckling: A buckling solution predicts the stability of a structure and can capture compressive failures that wouldn't show up in a normal linear solution. This is also a non-linear solution. Modal: A modal solution predicts the natural frequencies of a structure. This is great for making sure your structure avoids resonance when subjected to vibrations.

And More! There are many different types of solutions in every solver.

Dynamic Impact Solution



Non-Linear Buckling Solution





Planning The Analysis: Materials

Just like in Mechanics of Materials class, materials can be modeled in different ways in FEA.

Isotropic: This material will have the same properties in all directions. Metals are generally isotropic materials.**

2D or 3D Orthotropic: This material has different properties in two or three principal directions. Carbon fiber and core can be modeled as orthotropic.

A Note on Composites:

Composite laminates can be modeled in many different ways, but most methods involve defining a stack of materials and thicknesses, and letting the solver translate this into overall material properties.

***Wrought metals are isotropic, while cold formed metals can be orthotropic.*

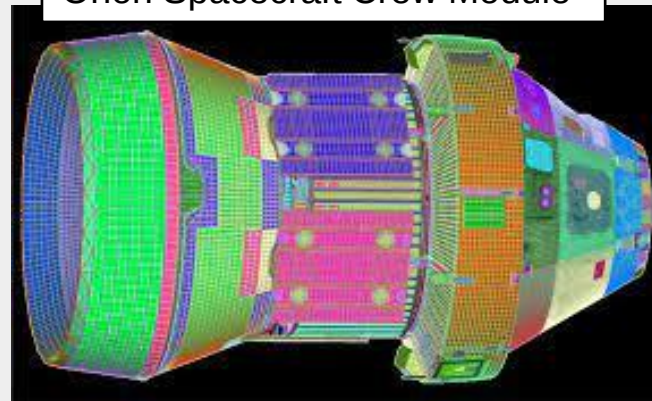




Discretization

How should you mesh the part?

Orion Spacecraft Crew Module



Space Shuttle Main Engine



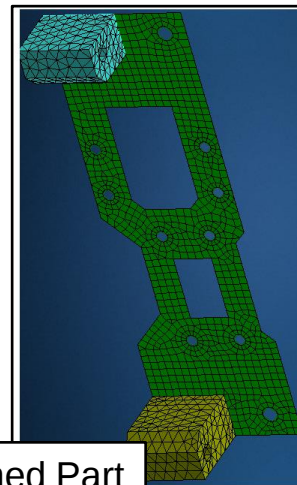
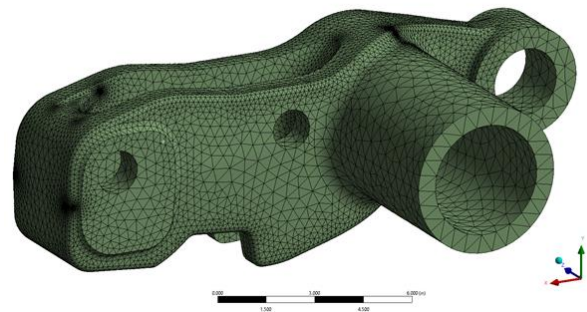
Discretization: Overview

Discretization is commonly referred to as meshing in finite element analyses. FEA works by splitting an object up into many simple pieces. It is then possible to solve a large system of simple problems instead of one large complex problem.

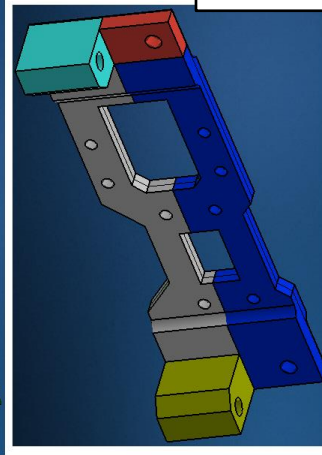
Each piece is called an element, and they are all connected at their vertices by nodes.

You should decide first how much you can idealize the part (remove small fillets, small holes, etc.), and then move onto meshing.

Meshing can be done in many different ways, and it is up to the analyst to decide the best method. These next few slides provide a basic outline of the element types you can use.



Meshed Part



Idealized Part



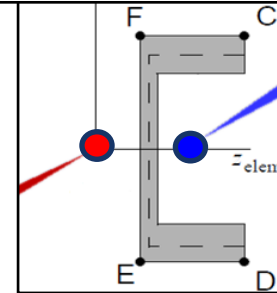
Discretization: 1D Elements

One dimensional elements are a great way to simplify slender objects like rods and beams! Use these when the expected root cause of failure is bending. Remember, orientation of these elements is critical!

They are extremely simple to make, and fast to solve, but are incapable of resolving detailed stresses within the elements.

Here are a few types commonly used.

Beam Element Cross Section



Bar Element	Rod Element	Beam Element	RBE2	RBE3	Spring
Tension, compression, torsion, shear, and bending supported. Neutral axis and shear center must coincide. Constant cross section.	Only has axial and torsional stiffness. Constant cross-sectional area.	Tension, compression, torsion, shear, and bending supported. Can define different sections at each end and even at some intermediate locations as well. Neutral axis and shear center can be non-coincident	This element couples all degrees of freedom of many dependent nodes to one independent node. Great for rigid connections.	This element is like an RBE2, but instead of a direct coupling of DOFs it geometrically averages them. Think of it as a 'soft' RBE2. Great for connections where you don't want extra stiffness	This element is commonly used to recover forces going through connections. Represents a simple spring.

Note: Don't connect super stiff springs or RBE2s to any elements you'd like to predict accurate stresses at, this will yield inaccurate results due to a discontinuity in stiffness!



Discretization: 2D Elements

Two dimensional elements are great for thin-walled parts like panels and sheet metal! If displacement is more than half the thickness of the shell, change to nonlinear analysis.

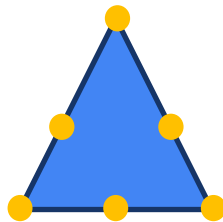
These are simple to make, and solve relatively fast, but are unable to resolve detailed through thickness stresses.

Generally, you can use quadrilateral or triangular elements. Triangular elements can fit geometry easier and thus are easier to mesh, but they have a fatal flaw: Linear triangular elements misrepresent stiffness. This means you must use *parabolic* triangles, which have more nodes and take longer to solve. Quadrilateral elements can be harder to mesh but solve much faster since they can be sufficiently accurate in linear form, which have fewer nodes.

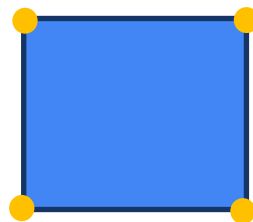
Most of the time with 2D meshing, you will use linear quadrilateral elements, and will be forced to include some linear triangular elements to avoid bad mesh quality. This is generally acceptable.



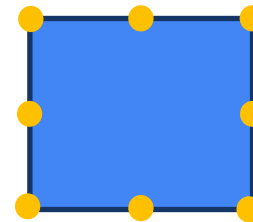
Linear



Parabolic



Linear



Parabolic

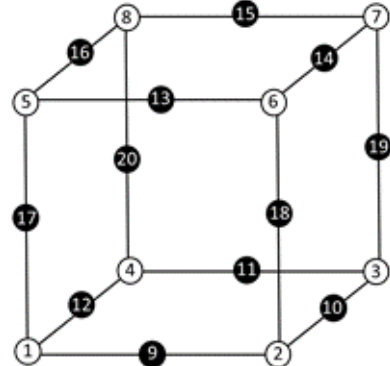
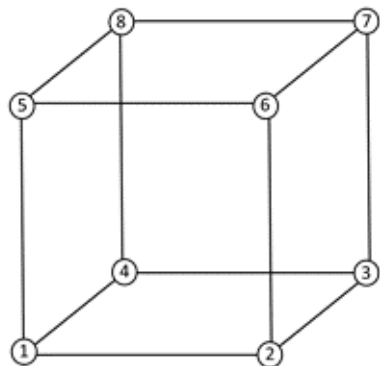
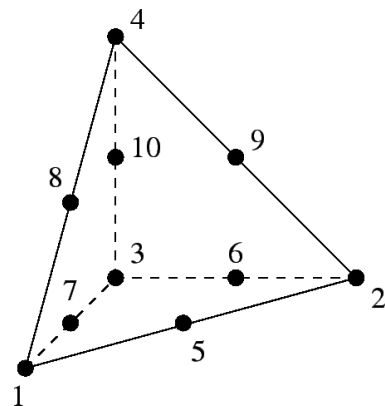
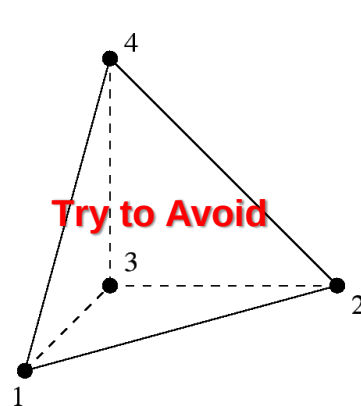


Discretization: 3D Elements

Three dimensional elements are used when meshing a solid part that can't be simplified further. Linear 3D elements ignore bending within the element, so you need at least three elements through the thickness of your part!

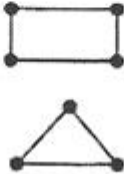
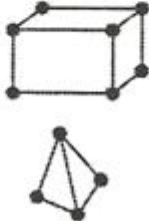
The main element types are tetrahedral (tet mesh, 4 sides) and hexahedral (brick mesh, 6 sides). Just like with 2D, linear tetrahedral elements should be avoided. As with 2D meshing, the hexahedral elements are harder to mesh, but solve faster, while parabolic tetrahedral elements are easier to mesh, but take much longer to solve.

Tip: Brick meshing is an art and will require you to split your part up into many sweepable bodies (Think of a sweepable body as one with a constant cross section). You will then have to extrude elements through the thickness of these split bodies, so plan out in which order you will mesh them.





Here's a great table summarizing elements!

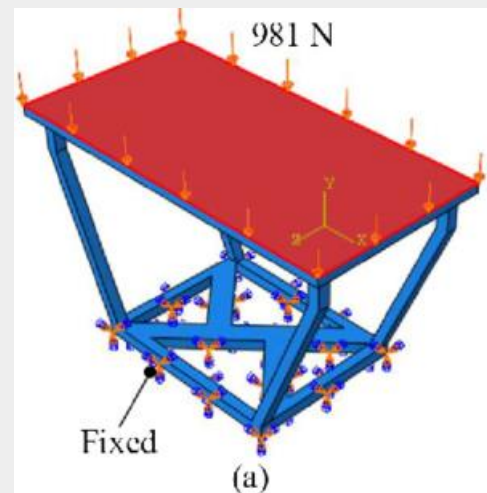
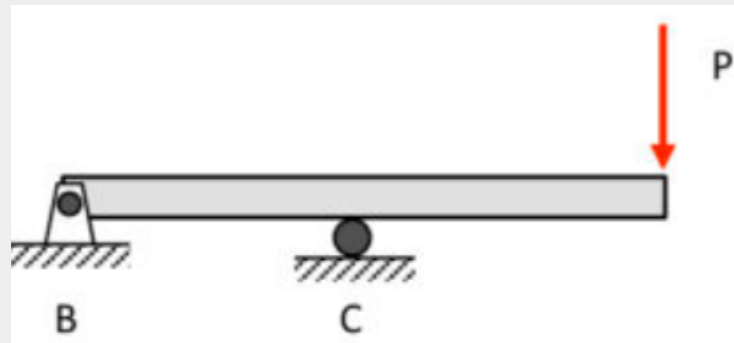
	Spring Elements	Line Elements	Surface Elements	Solid Elements	Multiple Constraint Elements (MPC)	
Physical Behavior	Simple Spring	Rod (axial load only), Beam	Shear, Membrane, Plate	Brick, Tetrahedron	Rigid Body Element	Interpolation Constraint Element
MSC/NASTRAN Element Name	CELAS1	CROD CBAR	CQUAD4 CTRIA3	CHEXA CTETRA	RBE1 RBE2	RBE3
Associated property entry	PELAS	PROD PBAR	PSHELL	PSOLID	None	None
Example of utilization	Connection between control surfaces and aircraft	Simple representation of fuselage stringer	Representation of fuselage and wing skin	Honeycomb of composite	Representation of engine	Load distribution
						

Note: This table references element names and property names corresponding to the FEA solver NASTRAN (CROD, CQUAD4, PBAR, etc.). These names correspond to how elements and properties are referenced in the files created by the pre-processor for the solver.



Boundary Conditions

How should you model the part's environment?



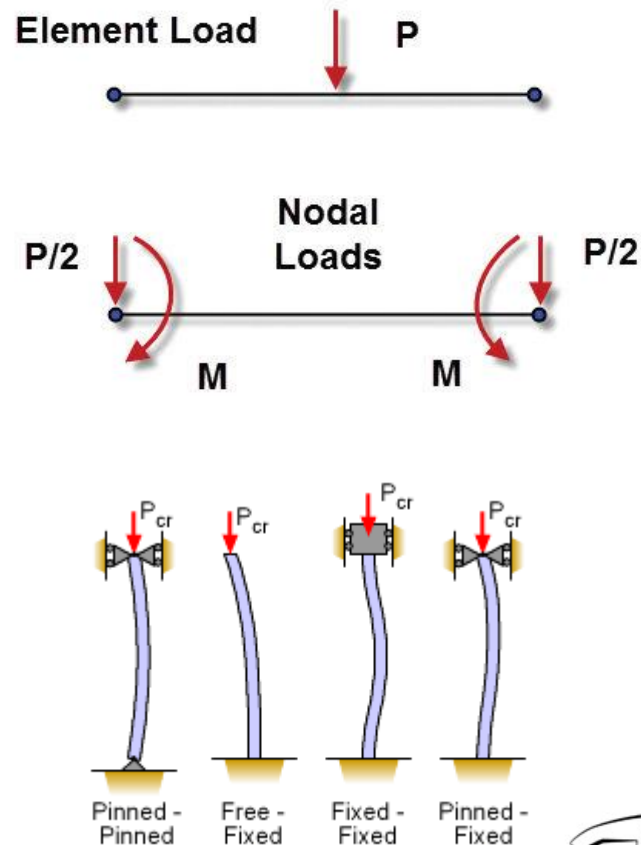


Boundary Conditions: Overview

Boundary conditions are how you choose to model the objects environment. This could mean representing things like bolted connections, pinned connections, pressure loading, line loading, inertial loading, and more.

FEA solvers have many different ways to represent boundary conditions, but in general they can be summarized into constraints (how your part is kept from moving) and loads (forces, pressures, and accelerations applied to your part).

Choosing the correct boundary conditions is one of **the most important parts** of any analysis and has large effects on results. Spend lots of time thinking and experimenting with these!





Boundary Conditions: Constraints

In general, constraints are how you keep the model from moving. For an FEA model to solve, the part needs to be fully constrained, meaning it can't rotate or translate freely. You constrain a part by fixing its DOFs. Here is some terminology you'll commonly hear.

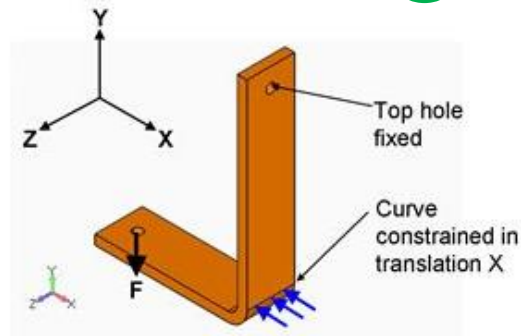
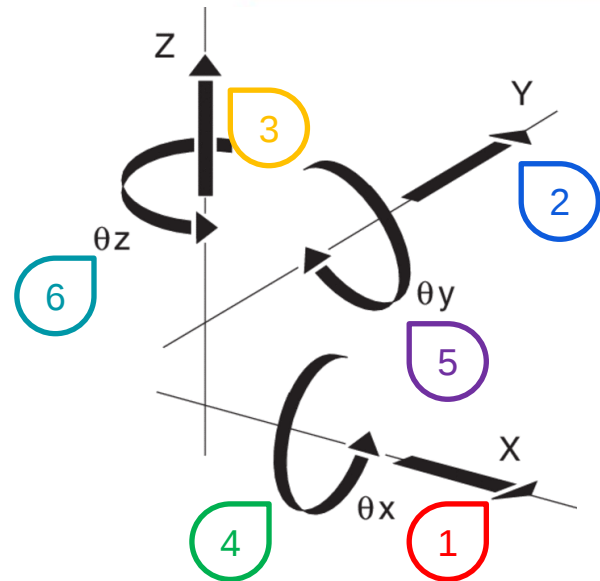
DOF: Degrees of freedom are the 6 types of motion a rigid body can experience. DOFs 1 through 6 are translation in **x**, **y**, and **z**, and rotation in **x**, **y**, and **z** respectively.

Fixed: This means all 6 DOFs are constrained.

Pinned: This means only translational DOFs are constrained.

You can constrain a model in many different ways depending on the connection. For example, a journal bearing might only have one rotational and one translational DOF released. While a spherical bearing might have all rotational DOFs released. Think wisely about how your part is supported before constraining it.

You can also constrain DOFs using spherical and cylindrical coordinate systems! You just have to make sure the constraints reference the right coordinate system.





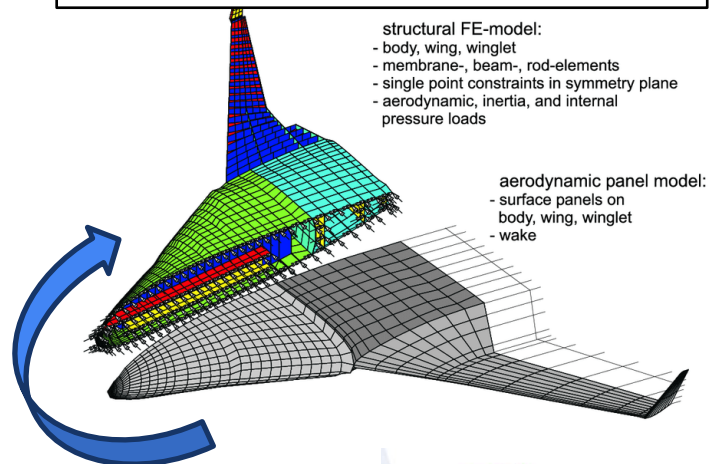
Boundary Conditions: Loading

Loads are how you represent the environment's influence on your part. For example, you could represent a wind load with a pressure distribution across your mesh. Or you could model a car going over a bump as an acceleration applied to your part (remember, acceleration is just a force distributed based on density).

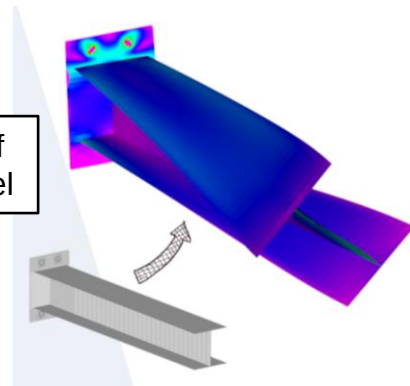
Your part may experience many different load cases within its working environment, so its important to include all of them in your analysis. As you check the severity of all the load cases, you can select the worst-case ones to drive your design.

A Note on Aerodynamic Components: Aerodynamic devices typically experience complex pressure loads. Often its necessary to derive these pressure loads from a CFD model and interpolate them onto your FEA mesh.

Pressure loading from CFD to FEA model



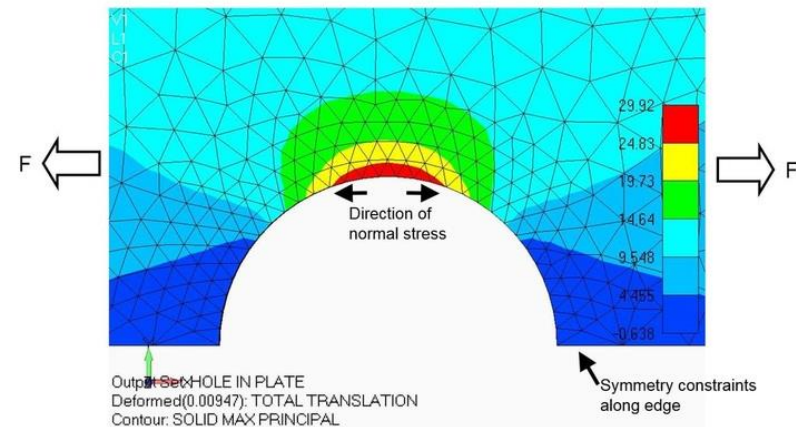
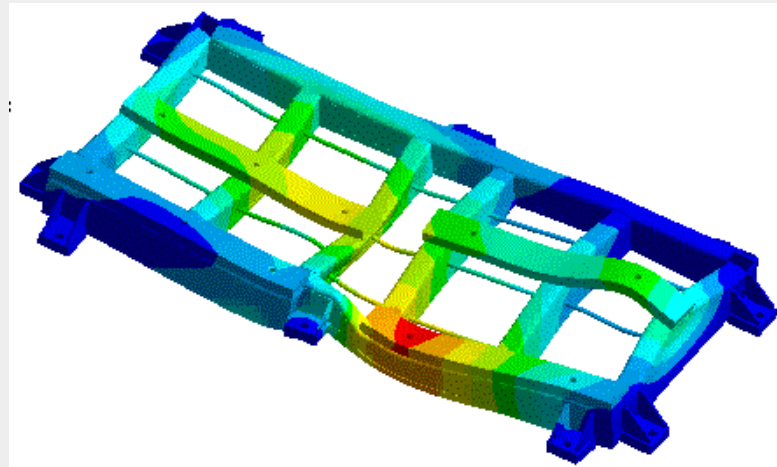
Torsional loading of detailed beam model





Post Processing

How should you interpret the results?





Post Processing: Knowing what to look for

The biggest part of any FEA, is knowing what you're looking for. As stated earlier, at a minimum you should know whether you're analyzing strength or stiffness. This will influence how you can process your model results.

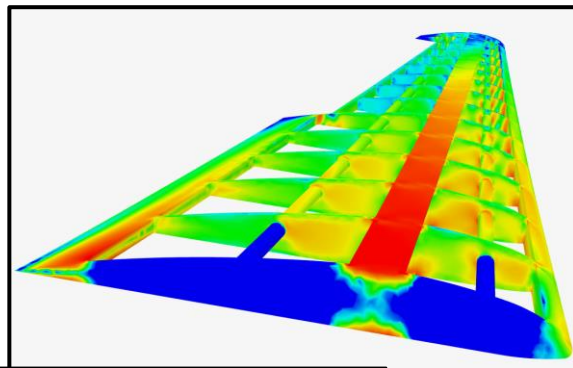
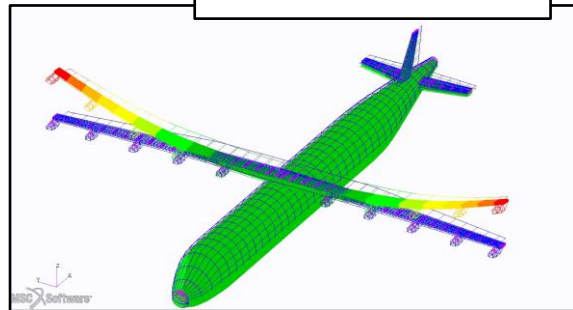
Stiffness: Stiffness models generally require lower levels of detail and may be poor for recovering detailed stresses. These models can be used for recovering maximum deflection, dynamic response, modes, buckling eigenvalues, forces going through joints and constraints, etc.

Strength: Strength models require high levels of detail and allow you to recover stresses and predict failure in your part.

Don't try and recover stresses from elements directly connected to RBE2s or stiff springs. These stresses will be unrealistic due to the large stiffness discontinuity. Recover loads at these connections using zero length spring elements and use analytic solutions or a more detailed FEA model to predict failure (this is the best way to use FEA)!

A Note on Composites: When analyzing a CFRP laminate, you must choose a failure index to use. There are many different failure indices, and some are empirically based.

Coarse aeroelastic stiffness model



Detailed strength model

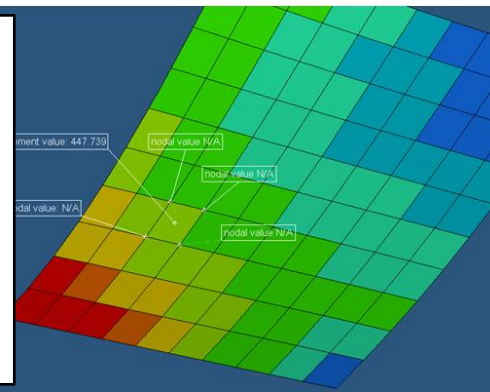


Post Processing: Plotting Best Practices

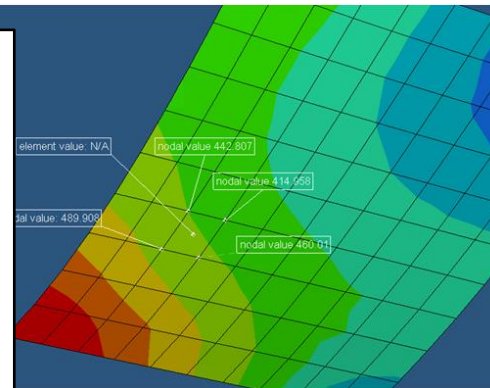
Color Bars: Never use magenta, it can be a misleading color in reports. One useful tip for strength analyses is to plot grey on elements experiencing failure (stress exceeding yield strength, failure index greater than 1, etc.). Also, turn off continuous plot colors to give a clearer result.

Elemental vs. Nodal Stresses: In general, element centroidal stresses are more accurate. This is because nodal stresses are extrapolated from the centroids for higher detail. So, for a sufficiently fine mesh, they should be similar, but only use nodal stresses if you need the extra level of detail.

Elemental Stresses



Nodal Stresses





And lastly, remember this:

“All models are wrong,
some are useful”

- *George Box (someone smart)*