Structures and FEA

Purdue Space Program
Hybrid and Solid Rocket Teams

Agenda

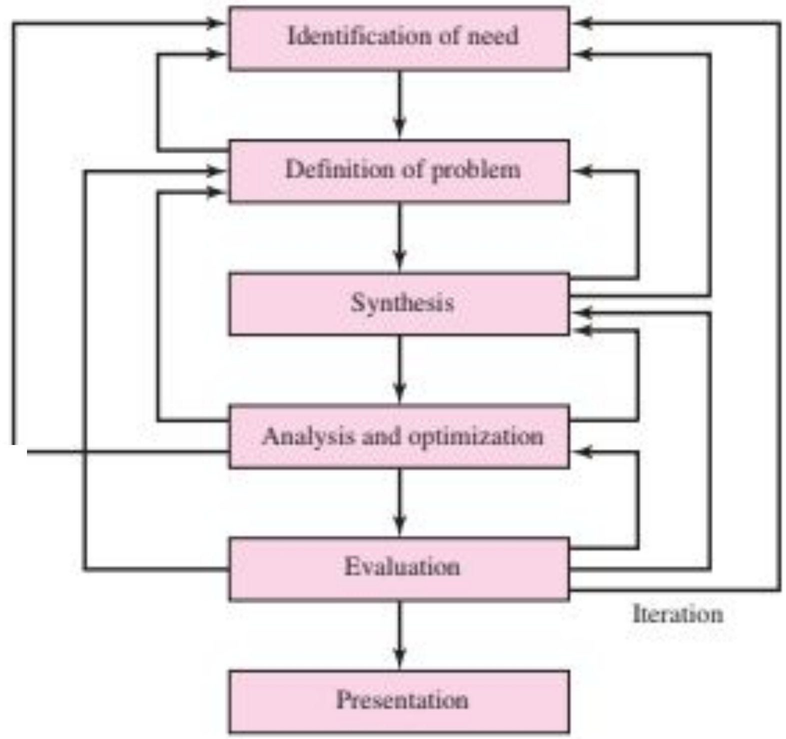
- Introduction to Structures
  - Design Process & Design Considerations
  - Basics of the Mechanics of Materials
  - Types of Loading
- Finite Element Analysis
  - Overview and Workflow
  - Basic Concepts of FEM
  - Nodes & Elements
  - Boundary Conditions and Material Properties
  - Determining Mesh Size using Mesh Convergence Study
  - Post-Processing Data
- ANSYS FEA Basics Tutorial
- Open **https://tinyurl.com/PSPANSYS**

PURDUE UNIVERSITY

Design Process

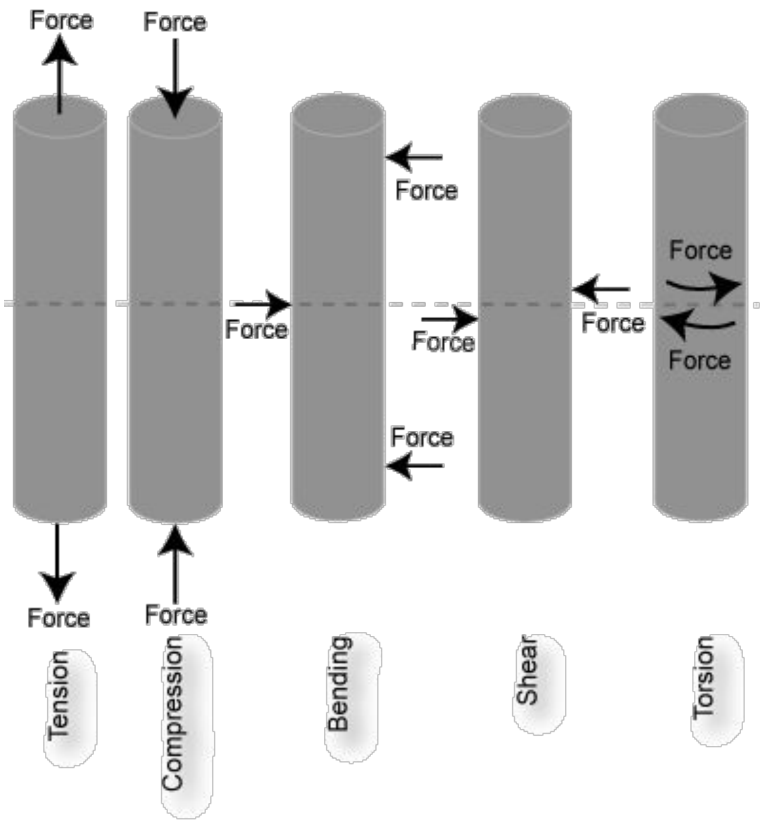
- Make sure to identify the original problem that needs to be solved
  - For our purposes we just have to make sure that the part is as structurally stable as possible with not added adequate weight
  - Strength vs Weight Ratio
- Know all loading conditions of the part being analyzed
- Design Considerations

1 Functionality
2 Strength/stress
3 Distortion/deflection/stiffness
4 Wear
5 Corrosion
6 Safety
7 Reliability
8 Manufacturability
9 Utility
10 Cost
11 Friction
12 Weight
13 Life
14 Noise
15 Styling
16 Shape
17 Size
18 Control
19 Thermal properties
20 Surface
21 Lubrication
22 Marketability
23 Maintenance
24 Volume
25 Liability
26 Remanufacturing/resource recovery

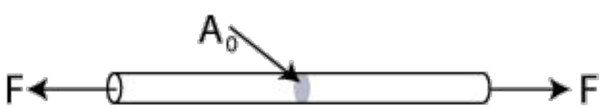


Types of Loads

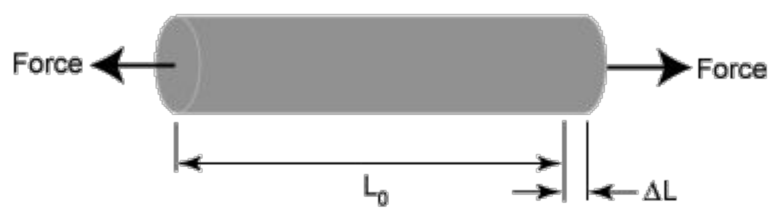
- Tension
- Compression
- Bending
- Shear
- Torsion



Stress vs Strain



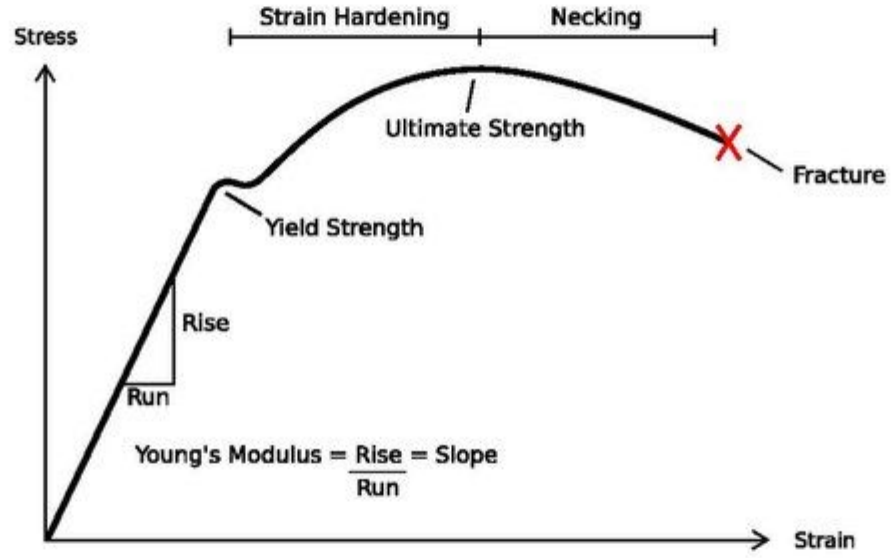
$$\text{Stress, } \sigma = \frac{\text{Force}}{\text{Cross-Sectional Area}} = \frac{F}{A_0}$$



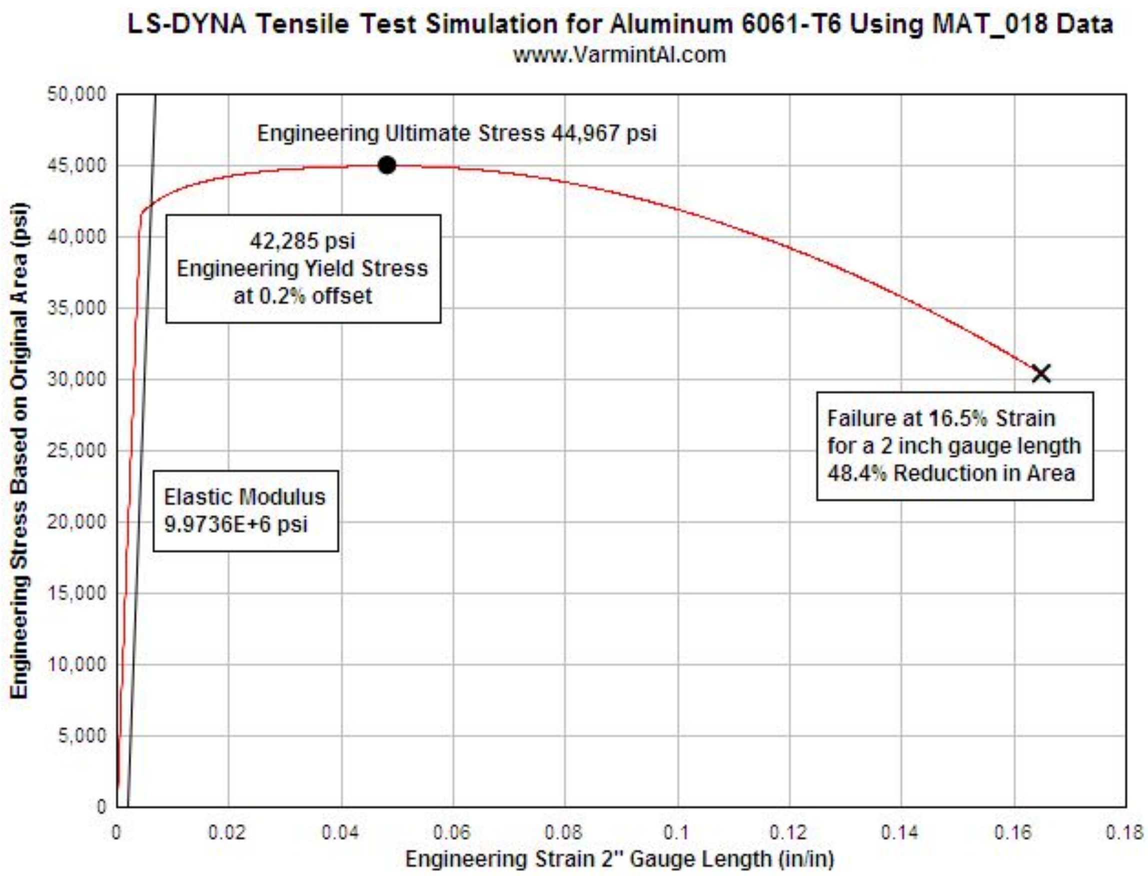
$$\text{Strain} = \frac{\text{Elongation}}{\text{Orininal Length}} = \frac{\Delta L}{L_0}$$

Stress Strain Curve

- ❑ Yield Strength
- ❑ Ultimate Strength
- ❑ Fracture



Engineering Yield Stress



Material Properties

- Metallic Material Properties Database 11 (MMPDS 11)
  - NASA Standard
  - Adopted by major companies such as SpaceX

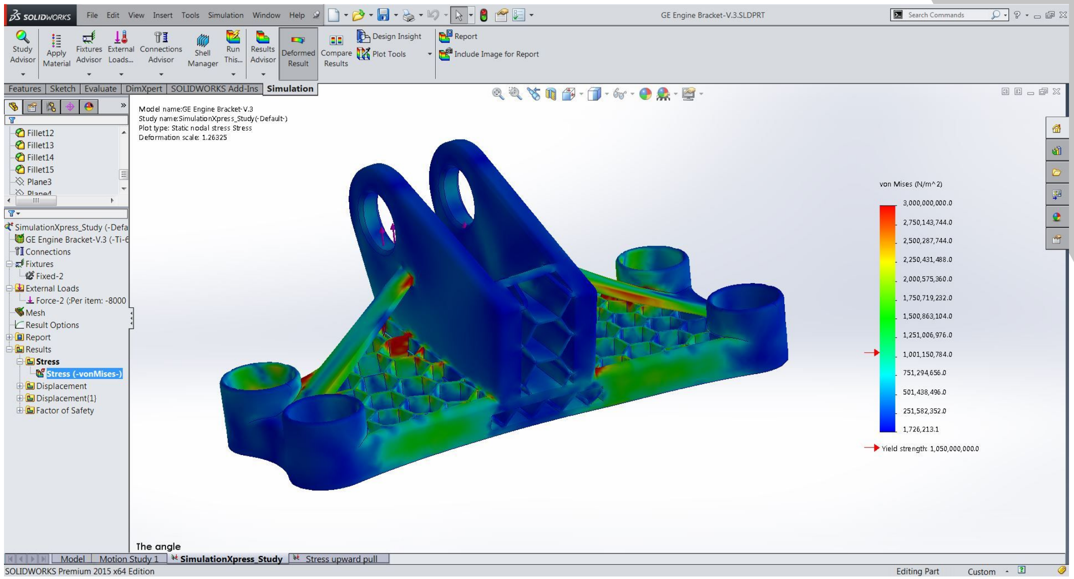
MMPDS-12
1 July 2017

Table 3.7.3.0(b). Design Mechanical and Physical Properties of 7040-T7451 Aluminum Alloy Plate

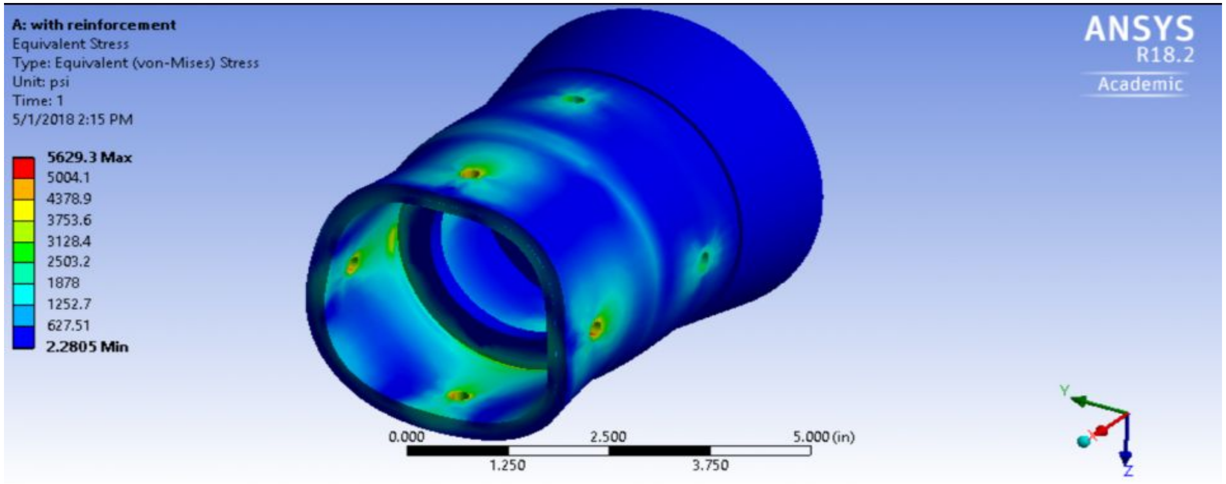
Specification	AMS 4211											
Form	Plate											
Temper	T7451											
Thickness, in.	3.001-4.000		4.001-5.000		5.001-6.000		6.001 - 7.000		7.001 - 8.000		8.001 - 8.500	
Basis	A	B	A	B	A	B	A	B	A	B	A	B
Mechanical Properties:												
F_{tu}, ksi:												
L	70	72	70	72	70[e]	71	69	70	68[b]	70	68[b]	70
LT	71[c]	74	71[d]	73	70[e]	72	69	70	69	69	68	69
ST	69	70	68[d]	70	68	69	66	67	66	67	66	67
F_{ty}, ksi:												
L	62[c]	65	62[d]	64	62[e]	64	62	62	61	62	61	63
LT	62[c]	65	62[d]	65	61[c]	63	60	62	60	61	59	61
ST	59[c]	61	58[d]	61	58[e]	61	57	58	57	58	56	58
F_{cy}, ksi:												
L	60	63	60	62	59	61	58	60	59	60	59	61
LT	64	67	64	67	63	66	62	64	62	64	61	63
ST	63	66	63	66	62	65	61	63	61	63	60	63
F_{su}, ksi:												
L-S	45	47	45	46	44	45	43	44	42	44	42	43
T-S	44	46	44	46	44	45	43	44	43	44	43	44
F_{bru}[a], ksi (e/D = 1.5):												
L	113	118	112	116	110	114	108	110	106	108	105	106
LT	112	117	112	115	110	114	108	110	105	108	105	106
F_{bru}[a], ksi (e/D = 2.0):												
L	143	150	143	147	140	145	137	140	134	136	133	134
LT	144	150	144	148	141	146	138	141	135	138	135	136
F_{bry}[a], ksi (e/D = 1.5):												
L	93	97	93	97	92	96	90	93	90	92	89	92
LT	94	98	94	98	93	96	91	93	90	92	87	91
F_{bry}[a], ksi (e/D = 2.0):												
L	114	119	114	119	112	117	110	113	110	113	108	112
LT	115	120	115	120	113	118	111	114	111	114	109	113
e, percent (S-Basis):												
L	9	—	9	...	8	—	7	—	6	—	6	—
LT	6	—	5	...	4	—	4	—	4	—	4	—
ST	3	—	3	...	3	—	3	—	3	—	3	—

Continued on the next page.

FEA



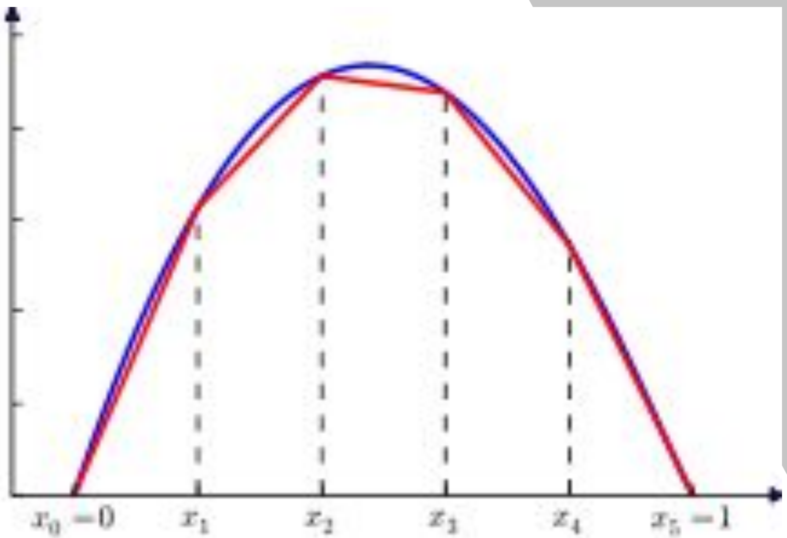
GE Engine Bracket-V.3.SLDPRT
Model name:GE Engine Bracket-V.3
Study name:SimulationXpress_Study(-Default-)
Plot type: Static nodal stress Stress
Deformation scale: 1.26325
von Mises (N/m^2)
3,000,000,000.0
2,750,143,744.0
2,500,287,744.0
2,250,431,488.0
2,000,575,360.0
1,750,719,232.0
1,500,863,104.0
1,251,006,976.0
1,001,150,784.0
751,294,656.0
501,438,496.0
251,582,352.0
1,726,213.1
Yield strength: 1,050,000,000.0
The angle
SOLIDWORKS Premium 2015 x64 Edition



A: with reinforcement
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: psi
Time: 1
5/1/2018 2:15 PM
5629.3 Max
5004.1
4378.9
3753.6
3128.4
2503.2
1878
1252.7
627.51
2.2805 Min
ANSYS
R18.2
Academic
0.000
2.500
5.000 (in)
1.250
3.750

FEA Overview

- Simple components can easily be calculated using hand calculations however many real world components are not so simple
- **Finite Element Method (FEM)**
  - FEM or Finite Element Method is a mathematical method for solving physical problems. FEM evaluates solutions at various points called nodes.
- **Finite Element Analysis (FEA)**
  - FEA or Finite Element Analysis on the other hand is the implementations of FEM. There are numerous different software programs available to implement FEM, known as FEA software
- **Basic Concepts of FEM**
  - Any continuous solution field such as stress, displacement, temperature, pressure, etc. can be approximated by a discrete model composed of a set of piecewise continuous functions defined over a finite number of subdomains



FEA Applications

- **Linear or Nonlinear**
  - All analyses can be classed as being linear or nonlinear. Whenever the 'initial conditions' change during an analysis, non-linearity exists.
  - Linear analysis have the ability to be directly 'scaled' … for example, if the load is doubled then the results (stress, deflection, etc) will simply double. The use of this knowledge can significantly reduce the number of FEA analysis that need to be undertaken.

Subtypes of FEA

Vibration and Impact: These are types of dynamic analysis that can be used to assess how a product will perform. For example, will the car steering wheel vibrate while driving? Would my product survive a drop onto floor from one metre up

Buckling: A length of wire can hold much less load in compression compared to tension due to a phenomenon called 'buckling'. It can occur in any object that is relatively thin/narrow in one direction, such as beams and sheet-metal parts. FEA can be used to predict the load at which an object will partially or fully buckle.

Contact: This nonlinear technique analyses the effect of parts contacting each other. For example, a car crashing into a flexible safety barrier or a bolted 'friction-grip' joint.

Fatigue: FEA is a powerful tool for assessing the complex effects of cyclic loading (fatigue) on components. A product life can be estimated in years and areas likely to crack highlighted.

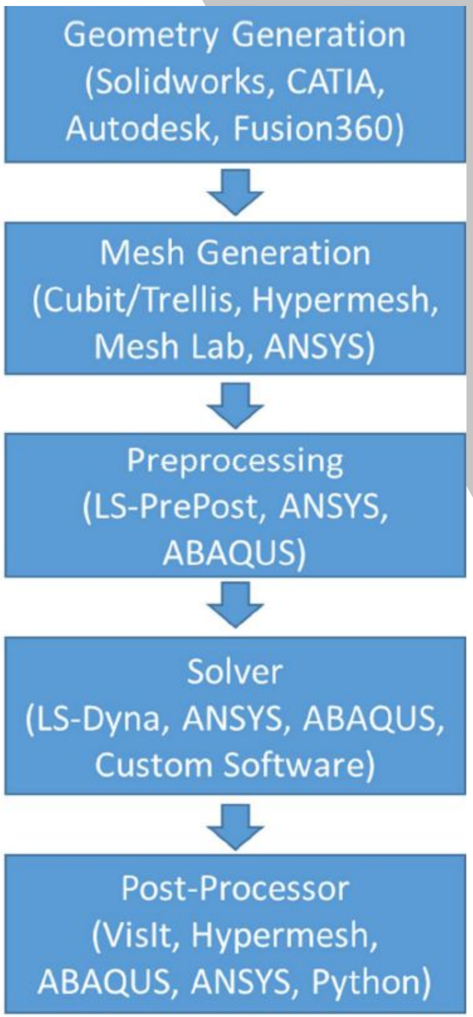
Heat transfer and Thermal Deflections and Stresses: FEA can be used to calculate the effect of heat on a components strength and temperature distribution.

Creep and Relaxation: A lot of engineering materials will tend to gradually stretch over time and can eventually rupture in a process called 'creep'. This is a key consideration for most plastic designs and is highly influenced by temperature. FEA can predict this behaviour.

FEA Workflow

Basics of FEA

1. **Preprocessing:** When you do an analysis you have to do modeling and meshing. This is essentially the process to set up the what type of FEA you will be conducting and
   a. **Meshing:** The process of breaking the geometric model into small pieces in order to create nodes and elements is called meshing. In other words meshing is the process of converting the geometric model to a FEA model.
   b. **Boundary Conditions:** In order to solve a FEA problem you have to apply force (these force could be structural, thermal, magnetic or of any form) and you have to resist at some points of the model from the effects generated by these forces. For example to see the stress in a cube you may think of applying force at the top surface and can constrain it at the parallel bottom surface. These set of loads and constraints are called boundary conditions.
2. **Solving:** After preprocessing we have to use a solver to find solution out of the FEA problem. In solidworks we could either use the native solidworks solver or can export or save the pre-processed mesh for solving with other external solver like ANSYS, SimScale, etc.
3. **Post-Processing:** After solving you view the results of your FEA problem. This section is called post-processing. Here we can see different types of images, plots and graphs of the result.



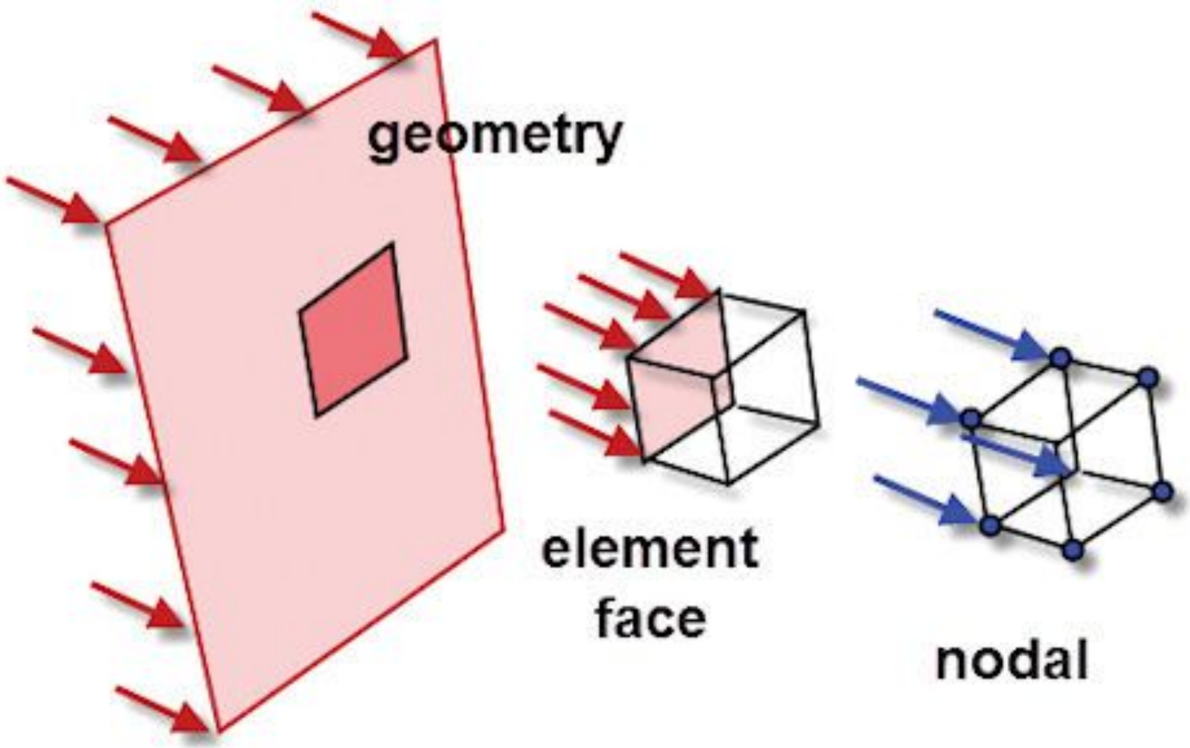
Nodes

- A node is simply a coordinate location in space where a DOF (degree of freedom) is defined
- Properties and Characteristics
  - Infinitesimally small
  - Defined with reference to a global coordinate system
  - Typically nodes are defined on the surface and in the interior of the component you are modeling
  - Form a grid work within component as a result of the mesh
  - Typically define the corners of elements
  - Where we define loads and boundary conditions
  - Location of our results (deformation, stress, etc.)
  - Nodes are the byproduct of defining elements

Elements

- An element is a mathematical relation that defines how a DOF of a node relates to the next.
- Properties and Characteristics
  - Point, 2D and 3D elements
  - Define a line (1D), area (2D) or volume (3D) on or within our model • Dimensions define an “Aspect Ratio”
  - A set of elements is known as the “mesh”
  - Mesh shape and density is critical to the analysis
  - Typically have many options that may be preset for the user
  - Elements are typically what we define

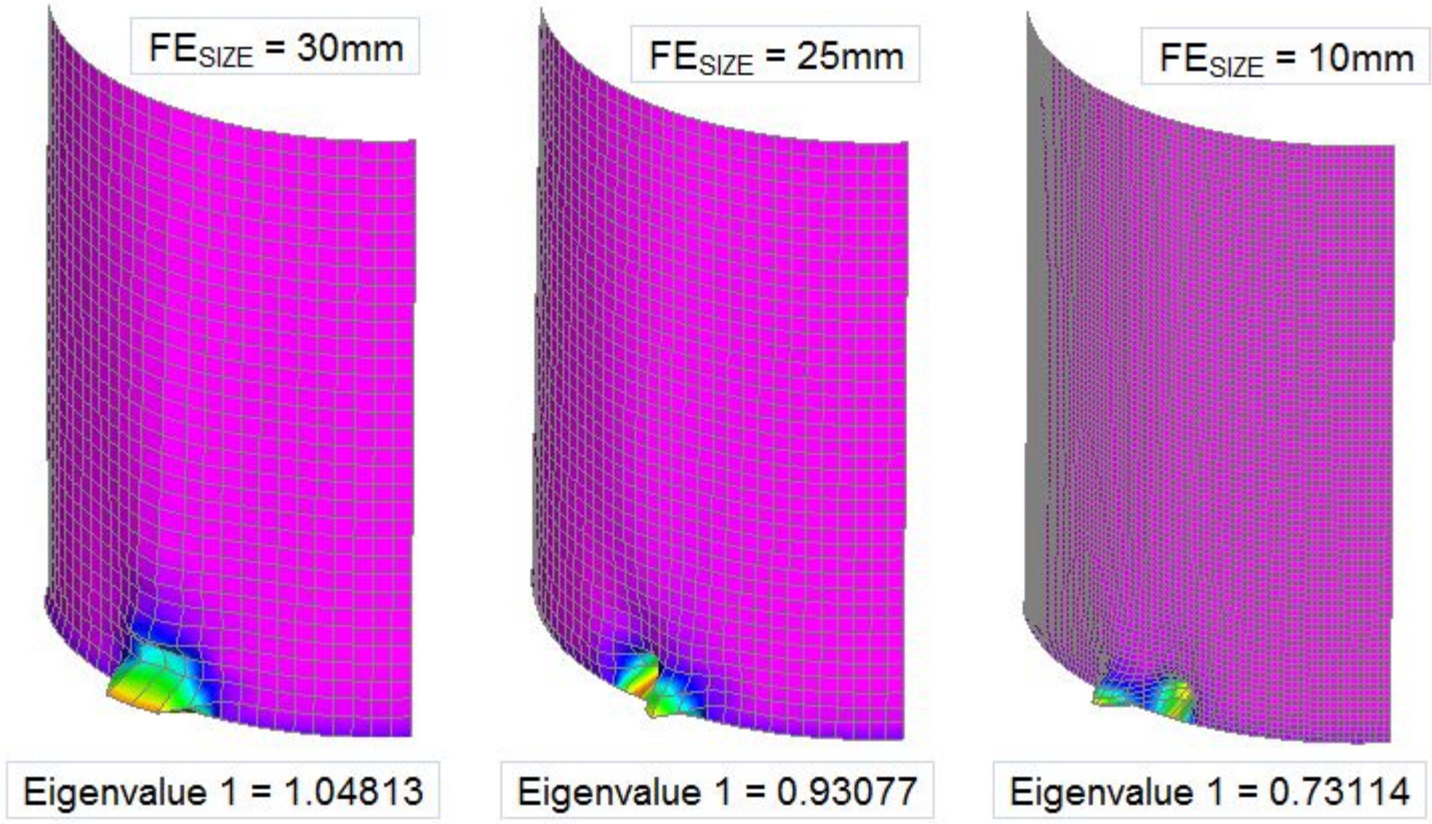
Overview



Determining Mesh Size

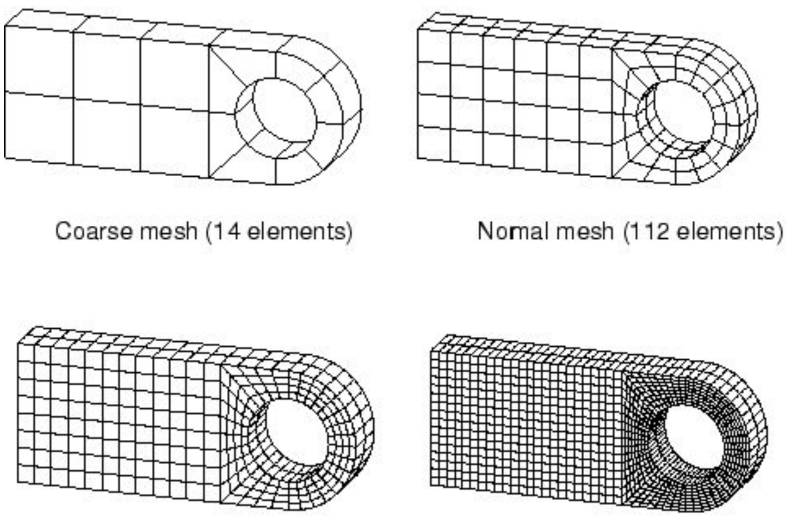
- Mesh size is one of the most common problems in FEA
- There is a fine line here:
    - Bigger elements give bad results, but smaller elements make computing so long you don’t get the results at all. You never really know where exactly is your mesh size on this scale.
    - Learn how to choose the correct size of mesh and estimate at which mesh size accuracy of the solution is acceptable.
- Usually smaller mesh means more accurate results, but the computing time gets significant as well
- Need to do a thing called Mesh Convergence
    - Essentially an iterative process of finding out what mesh size works best
    - In ANSYS you can parameterize the mesh size

Determining Mesh Size



Mesh Refinement

Mesh refinement



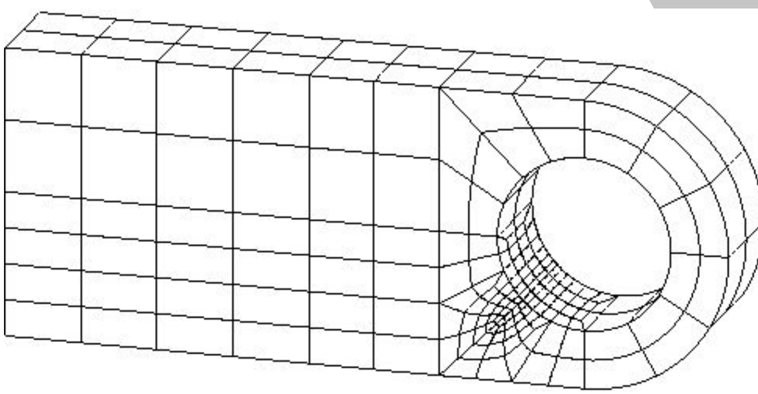
Coarse mesh (14 elements)

Nomal mesh (112 elements)

Fine mesh(448 elements)

Very fine mesh (1792 elements)

Localized Mesh refinement



Coarse to Fine simulation time increase:
22.5x

Coarse to Localized simulation time increase:
3.44x

THANK YOU

Contact Info

Arpit Agarwal | agarw162@purdue.edu | Hybrids Structures Team Lead

Cody Zrelak | czrelak@purdue.edu | Solids Structures Team Lead

DM in slack, or email with any questions.