Patrick Watson

Professor Rawlings

Math 208

18 June 2020

Finite Element Analysis with Linear Algebra Examples

Finite Element Analysis is the process of breaking down a large problem into smaller, more manageable, problems. It essentially relies on the decomposition of a complex domain into smaller subdomains which in turn creates a mesh with nodal points. The biggest benefit for Finite Element Analysis, or FEA, is that it can be evaluated through computer processing and with modern computers these computations can be carried out quickly. Applications, such as Solidworks, utilize FEA to solve problems by breaking it into smaller differential volumetric elements which in turn can be meshed to form a visualization. CAD, Computer Aided Design, files can be created and be analyzed for design flaws using these applications, essentially making hand computational methods obsolete. It provides a powerful tool for engineers to predict how an object will behave under real world scenarios such as forces, vibrations, heat flow, and fluid flow. As such, FEA is prevalent in the Engineering world for design analysis.

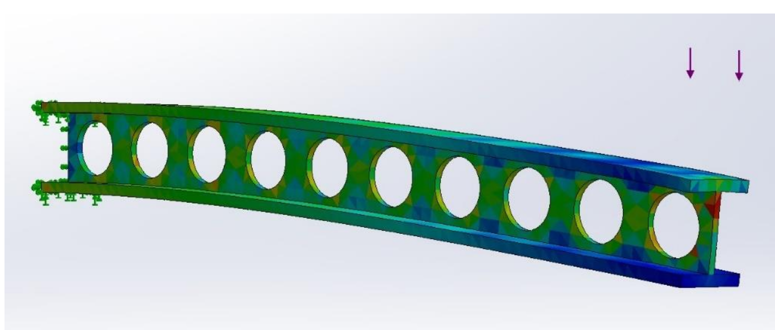


Figure 1. An I-beam created in Solidworks

History of Finite Element Analysis

Finite element analysis was born out of the necessity for solving structural mechanics problems. Finite element analysis techniques were in the works dating back to the 1800s and it only started to be formulated in the 1950s. Computational fluid dynamics came from the works of the finite element method and effectively shaped the world around us from cars to airplanes. CFD analysis started with the Navier-Stokes equations in the 1930s, but because computational power was limited back then, only drastically simplified problems could be solved. As time dwelled on, NASA and Boeing picked up the mantle of advancing finite element methods and computational fluid dynamics and it evolved through advanced mathematics and greater computer processing power. In the 90s Ford Motor Co. and General Motors realized the potential of using these techniques to make their cars more aerodynamic and in turn increase fuel economy.

How Linear Algebra is used for FEA

A classic example for Finite Element Analysis is the pin jointed Truss (fig.2) as a Truss can be broken up into a system of individual “Finite” elements. Each joint is a node and each beam is an element. A node represents a cartesian location with a degree of freedom, or DOF, value. It is important to realize that the DOF is a property of the node and is not only in terms of displacement. The elements should be considered what is resisting that node’s change.

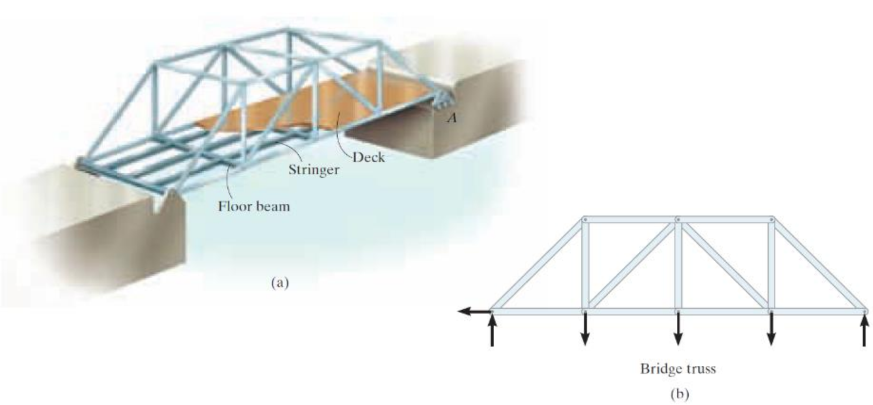


Figure 2. A bridge truss. Hibbeler. R. C. Statics pg274

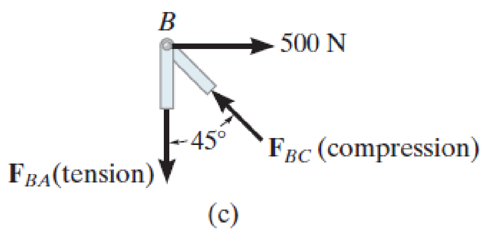


Figure 3. A Joint. Hibbeler, R. C. Statics(14th Edition) pg276

When using the method of joints to solve a truss it is assumed that the entire structure is under equilibrium and in turn each joint is under equilibrium. Thus, each joint can be drawn as a free body diagram(fig.3) and each element is represented by a vector force. Through equilibrium of a rigid body, a system of linear equations is formed and can be effectively written into a coefficients Matrix.

Often, frames and simple machines can be analyzed similarly to trusses, but as the complexity increases, a mesh must be used and heavy calculation is required. This is where computer applications come in to accurately model the structure through a CAD drawing. By setting the boundary conditions, force loads, and the material, the application can model the displacement, draw shear and moment diagrams, and show stresses.

Stiffness Matrix

Furthermore, Hooke's law can be used to model the elastic deformation of the axial loaded elements. As each element is under tension or compression, can analyze the elastic displacement using the equation:

$$\delta = \frac{NL}{AE}$$

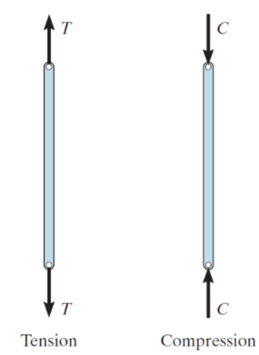


Figure 4. Hibbeler. R. C. Statics. pg275

Where N is the internal force in the element, L is the length, A is the cross-sectional area of the element, and E is the modulus of elasticity of the material. But to model the elastic displacement of the system under load, a matrix can be employed using the generalized formula $\{F\} = [K]\{u\}$. Where K we can set it equal to $\frac{AE}{L}$, which solely depends on the material properties. u is the nodal displacement. F is the internal axial force.

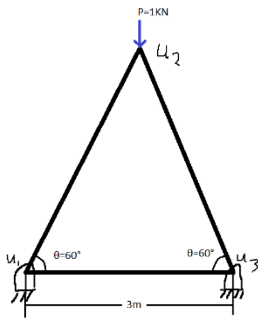
$$\begin{bmatrix} K_{11} & K_{12} & \dots & K_{1m} \\ K_{21} & K_{22} & \dots & K_{2m} \\ \vdots & \vdots & \ddots & \vdots \\ K_{n1} & K_{n2} & \dots & K_{nm} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{Bmatrix}$$

Other disciplines for FEA:

DISCIPLINE	DEGREE OF FREEDOM	FORCE VECTOR
STRUCTURAL/SOLIDS	Displacement	Mechanical Forces
HEAT CONDUCTION	Temperature	Heat Flux
GENERAL FLOWS	Velocity	Fluxes
ELECTROSTATICS	Electrostatics	Charge Density
MAGNETOSTATICS	Magnetic Potential	Magnetic Intensity

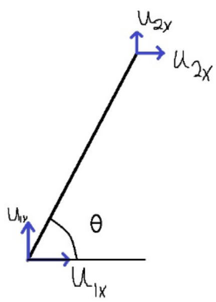
Table 1: Disciplines of FEA

Example



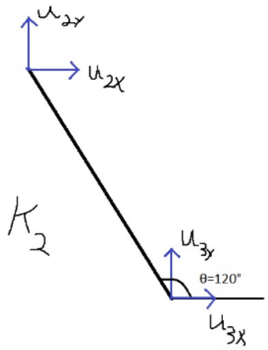
Determine the displacement of node U2 with force P=1KN using beams of cross-sectional area of $.0025m^2$ and a modulus of elasticity of $200GPa$.

First, break the problem into elements and define our variables in terms of our axis. Then define K_1: where Theta is 60 degrees. Since there are 6 Degrees of Freedom, the resultant matrix is 6x6.

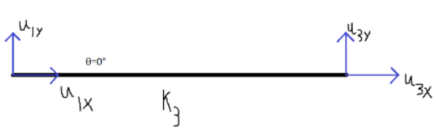


$$K = \frac{AE}{L}\begin{bmatrix} \cos^2\theta & \cos\theta\sin\theta & -\cos^2\theta & -\cos\theta\sin\theta \\ \cos\theta\sin\theta & \sin^2\theta & -\cos\theta\sin\theta & -\sin^2\theta \\ -\cos^2\theta & -\cos\theta\sin\theta & \cos^2\theta & \cos\theta\sin\theta \\ -\cos\theta\sin\theta & -\sin^2\theta & \cos\theta\sin\theta & \sin^2\theta \end{bmatrix}$$

$$K_1 = \frac{(.0025m^2)(200GPa)}{(3m)}\begin{bmatrix} k_{1x} & k_{1y} & k_{2x} & k_{2y} & \\ .250 & .433 & -.250 & -.433 & k_{1x} \\ .433 & .75 & -.433 & -.75 & k_{1y} \\ -.250 & -.433 & .250 & .433 & k_{2x} \\ -.433 & -.75 & .433 & .75 & k_{2y} \end{bmatrix}$$



$$K_2 = \frac{(.0025m^2)(200GPa)}{(3m)}\begin{bmatrix} k_{2x} & k_{2y} & k_{3x} & k_{3y} & \\ .250 & -.433 & -.250 & .433 & k_{2x} \\ -.433 & .750 & .433 & -.750 & k_{2y} \\ -.250 & .433 & .250 & -.433 & k_{3x} \\ .433 & -.750 & -.433 & .750 & k_{3y} \end{bmatrix}$$



$$K_3 = \frac{(.0025m^2)(200GPa)}{(3m)}\begin{bmatrix} k_{1x} & k_{1y} & k_{3x} & k_{3y} & \\ 1 & 0 & -1 & 0 & k_{1x} \\ 0 & 0 & 0 & 0 & k_{1y} \\ -1 & 0 & 1 & 0 & k_{3x} \\ 0 & 0 & 0 & 0 & k_{3y} \end{bmatrix}$$

Add all the elements together to form the global stiffness matrix

$$\Sigma K = \frac{AE}{L}\begin{bmatrix} k_{11}^{(1)}+k_{11}^{(3)} & k_{12}^{(1)}+u_{12}^{(3)} & k_{13}^{(1)} & k_{14}^{(1)} & k_{15}^{(13)} & k_{16}^{(3)} \\ k_{21}^{(1)}+k_{21}^{(3)} & k_{22}^{(1)}+k_{22}^{(3)} & k_{23}^{(1)} & k_{24}^{(1)} & k_{25}^{(3)} & k_{26}^{(3)} \\ k_{31}^{(1)} & k_{32}^{(1)} & k_{33}^{(1)}+k_{33}^{(2)} & k_{34}^{(1)}+k_{34}^{(2)} & k_{35}^{(1)} & k_{36}^{(1)} \\ k_{41}^{(1)} & k_{42}^{(1)} & k_{43}^{(1)}+k_{43}^{(2)} & k_{44}^{(1)}+k_{44}^{(2)} & k_{45}^{(1)} & k_{46}^{(1)} \\ k_{51}^{(3)} & k_{52}^{(3)} & k_{53}^{(1)} & k_{54}^{(1)} & k_{55}^{(2)}+k_{55}^{(3)} & k_{56}^{(2)}+k_{56}^{(3)} \\ k_{61}^{(3)} & k_{62}^{(3)} & k_{63}^{(1)} & k_{64}^{(1)} & k_{65}^{(2)}+k_{65}^{(3)} & k_{66}^{(2)}+k_{66}^{(3)} \end{bmatrix}$$

$$\Sigma K = \frac{(.0025m^2)(200GPa)}{(3m)}\begin{bmatrix} k_{1x} & k_{1y} & k_{2x} & k_{2y} & k_{3x} & k_{3y} & \\ 1.250 & .433 & -.250 & -.433 & -1 & 0 & k_{1x} \\ .433 & .750 & -.433 & -.75 & 0 & 0 & k_{1y} \\ -.250 & -.433 & .50 & 0 & -.250 & .433 & k_{2x} \\ -.433 & -.75 & 0 & 1.5 & .433 & -.750 & k_{2y} \\ -1 & 0 & -.250 & .433 & 1.250 & -.433 & k_{3x} \\ 0 & 0 & .433 & -.750 & -.433 & .750 & k_{3y} \end{bmatrix}$$

Points U1 and U3 are fixed, so their value does not contribute to the displacement. To ensure that the matrix contains a unique solution the determinate can be evaluated and should be nonzero.

$$\Sigma K = \frac{(.0025m^2)(200GPa)}{(3m)}\begin{bmatrix} k_{1x} & k_{1y} & k_{2x} & k_{2y} & k_{3x} & k_{3y} & \\ 0 & 0 & 0 & 0 & 0 & 0 & k_{1x} \\ 0 & 0 & 0 & 0 & 0 & 0 & k_{1y} \\ 0 & 0 & .5 & 0 & 0 & 0 & k_{2x} \\ 0 & 0 & 0 & 1.5 & 0 & 0 & k_{2y} \\ 0 & 0 & 0 & 0 & 0 & 0 & k_{3x} \\ 0 & 0 & 0 & 0 & 0 & 0 & k_{3y} \end{bmatrix}$$

Now that it is greatly simplified, the displacement of node U2 must be found using:

$$\{F\} = [K]\{u\}$$

$$\begin{Bmatrix} F_x \\ F_y \end{Bmatrix} = \frac{(.0025m^2)(200GPa)}{(3m)}\begin{bmatrix} .5 & 0 \\ 0 & 1.5 \end{bmatrix}\begin{Bmatrix} u_{2x} \\ u_{2y} \end{Bmatrix}$$

$$F_x = 0, F_y = -1000N$$

$$0 = 166666666.7\frac{N}{m}[.5u_{2x} + 0]$$

$$u_{2x} = 0$$

$$-1000N = 166666666.7\frac{N}{m}[0 + 1.5u_{2y}]$$

$$u_{2y} = -4 \times 10^{-6}m$$

Solidworks Example

In Figure 5 is a connecting rod for a car's engine out of 4340 Steel made in Solidworks. First start by finding dimensions online of a LS V8 connecting rod and create an outline. Then extrude the face to create the 3-dimensional shape. In Figure 6, this is the mesh with the nodes and elements that was created by the application.

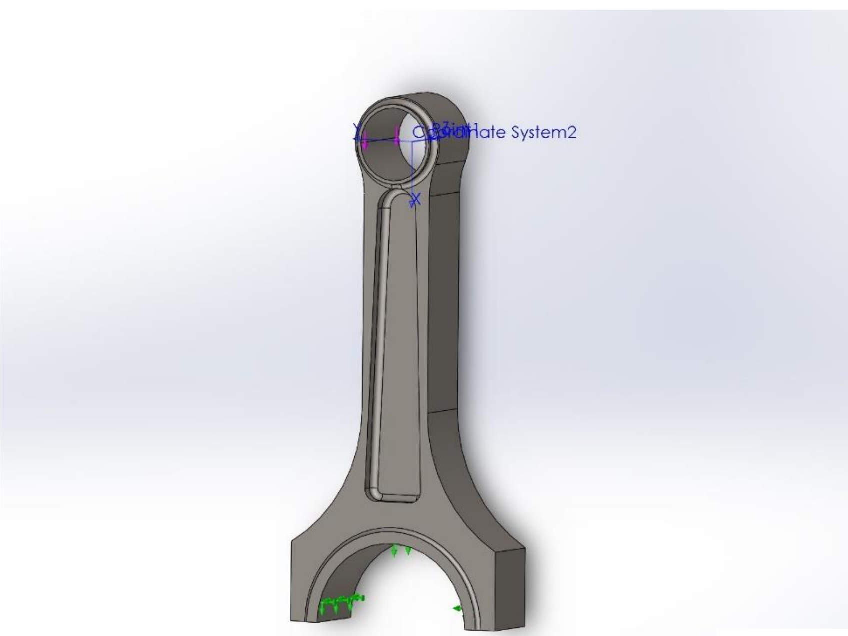


Figure 5. Connecting Rod in Solidworks

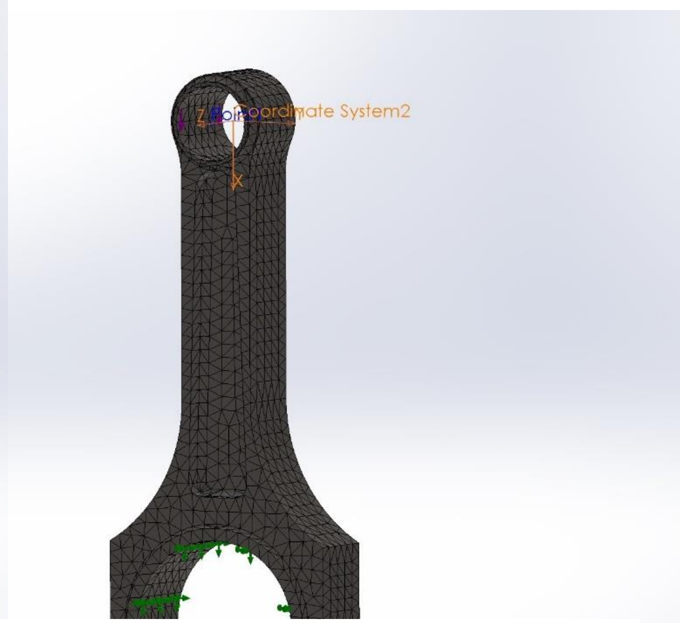


Figure 6. Connecting Rod Mesh in Solidworks

Using Solidworks Simulation a prescribed bearing load can be added on the surface of the top bearing. Usually there would be a wrist pin in that hole that would connect the rod to the piston, but it is removed for this problem. In this case a 10,000 pound load is added to make sure that the deformation was apparent. In figure 3 the deformation is apparent.

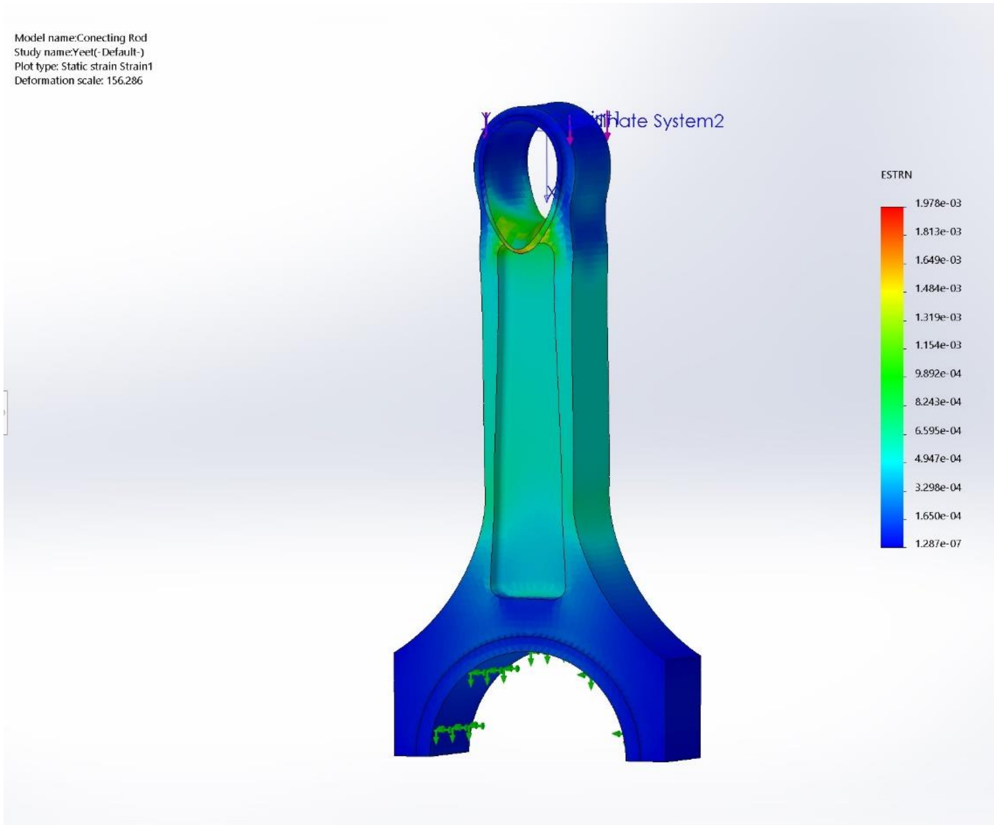


Figure 7. Deformed Connecting Rod

Conclusion

Finite element analysis has made great strides in the last century and when combined with computers, it has helped the world evolve past tedious hand calculations and has greatly increased efficiency. Through concepts from linear algebra complex engineering problems can be solved swiftly. Computer aided modeling is a powerful tool for engineers of all disciplines and when combined with finite element analysis it is invaluable.

Works Cited

Hibbeler, R. C. *Statics*. Pearson Prentice Hall, 2016.

"Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM)." *UVIC*, UVIC, www.engr.uvic.ca/~mech410/lectures/FEA_Theory.pdf.

Madenci, Erdogan, and Ibrahim Guven. *The Finite Element Method and Applications in Engineering Using ANSYS®*. Springer US, 2015.

Roylance, David. "Finite Element Analysis." *Massachusetts Institute of Technology*, 28 Feb. 2001, web.mit.edu/course/3/3.11/www/modules/fea.pdf.