Finite Element Analysis

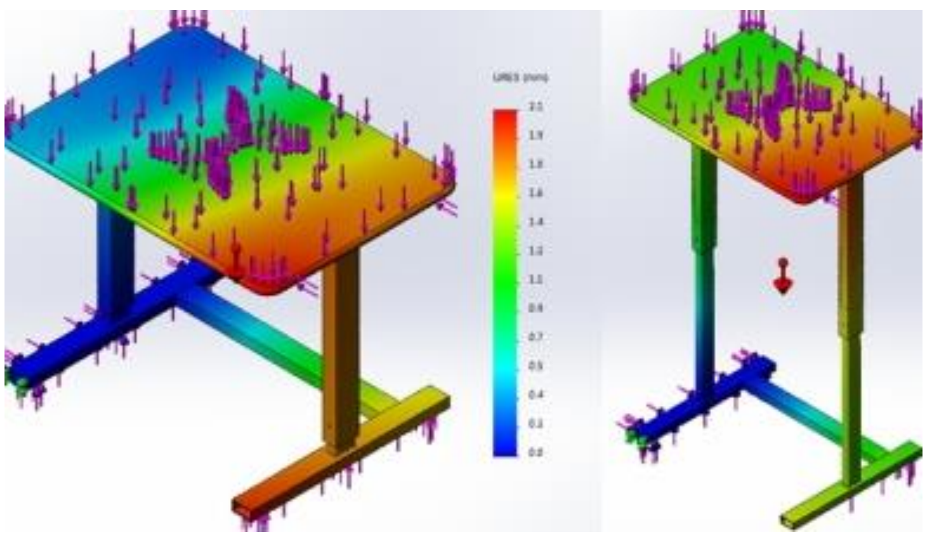
Annette Lopez Davila
Math 400, College of William and Mary
Professor Chi-Kwong Li

Abstract

This paper illustrates the importance of polynomial approximation and matrix theory in the engineering application of the Finite Element Method.

1 Introduction

In the world of engineering, simulation and modeling can tell us about the function of an object and how it interacts in the world. One such simulation system known as Finite Element Analysis (FEA) uses numerical approximations and discretization in order to predict how objects react to forces, temperature changes, fluid flow, and other conditions. When looking at the object, we define its shape, its components, and how it responds to the environment. Each element of the object must be at equilibrium in order for the object to function correctly. If one element is out of balance, the entire structure may fail. By accounting for the material properties and design of a structure, FEA may approximate the behavior of an object under stress, temperature changes, pressure changes, loads, etc.



2 History of FEA

Before technological methods such as FEA were created, engineers relied on physical tests to determine whether a design could withstand the environment it was created for. However, such conditions cannot always be simulated and are very expensive. The Finite Element Method was created to solve aeronautical structural design problems that surfaced after World War II. Many mathematical contributions to the field of partial differential equations helped advance the development of the Finite Element Method.

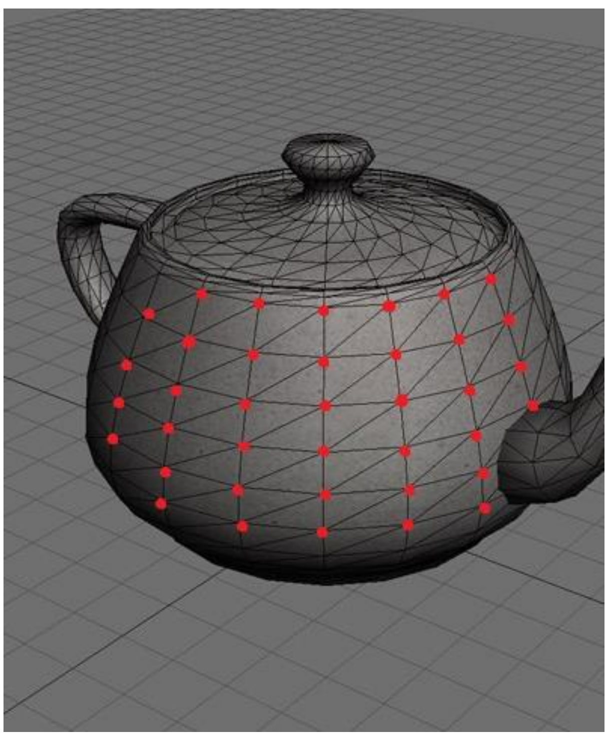
In the 1940s, Hrennikoff introduced the concept of using a lattice like approach to objects, while Richard Courant divided an object into finite subregions to solve second order elliptic partial differential equations. In the 1950s, a Chinese mathematician Feng Kang proposed a finite difference method based on variation principle in order to also solve PDEs. It wasn't until the late 60s and 70s that NASA applied FEA to non-linear problems using software. This was later given a rigorous mathematical basis.

3 The Finite Element Method

Most engineering applications are represented with differential equations with constraints called boundary conditions. For simple geometries, we use analytical methods such as integration and Laplace Transforms. However, complex geometries such as machinery need to be handled differently. The Finite Element Method will take a system of differential equations and boundary constraints and turn them into linear functions.

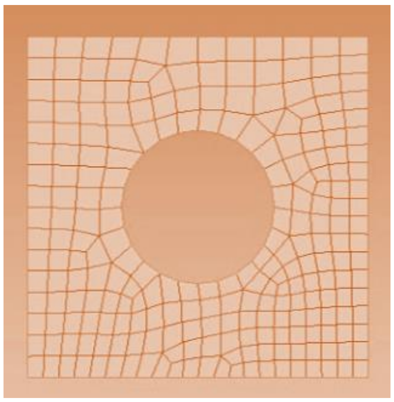
There are three overarching steps in the Finite Element Method: the discretization strategy, the solution algorithm, and post processing procedure. Within these steps, there are five mathematical sub steps: establishing a strong formulation of equations, obtaining a weak formulation, choosing approximations for unknown functions, choosing a basis function, and solving. In essence, the Finite Element Method takes a system governed by differential equations and partitions it into elements that can individually be solved with simple linear equations. With these partitions, we can attempt to make a linear approximation from the continuous solution.

3.1 Discretization Strategy: Finite Elements

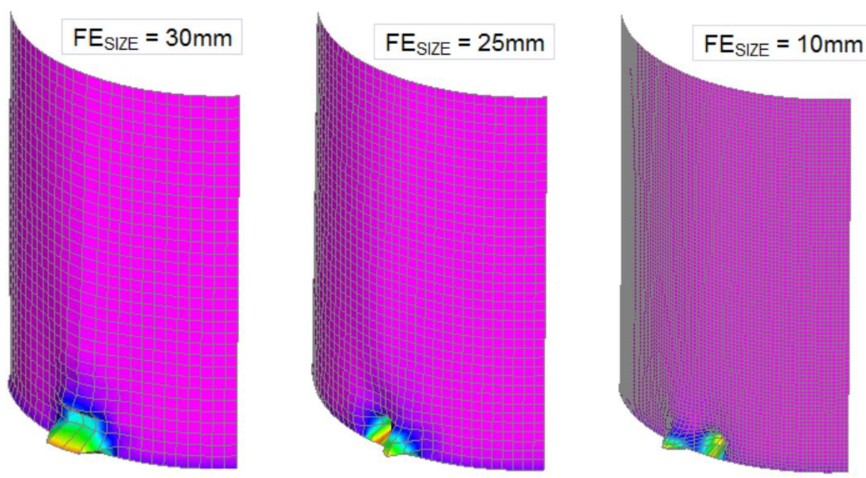


If we consider a solid of the n-th dimension, we can hypothetically divide this object into smaller components or geometrical shapes. These components are referred to as finite elements, a term coined in 1960 by Ray William Clough. In mathematics, discretization is defined as a process in which a continuous function, data, model, or variable is defined into a finite set of values. The word finite in finite element analysis defines the method as using limited degrees of freedom when modeling the behavior of an element at certain conditions. In FEA, discretization models the body of an object by dividing it into finite elements, thus creating a mesh. Each shape is interconnected at two or more nodes. Boundary lines or planes define the complete set. The nodes are the joint points of elements; they are specific reference points in some object. By taking these discrete points as data, the laws of physics may be applied to these objects.

3.11 Mesh Dimensionality, Shape, & Size

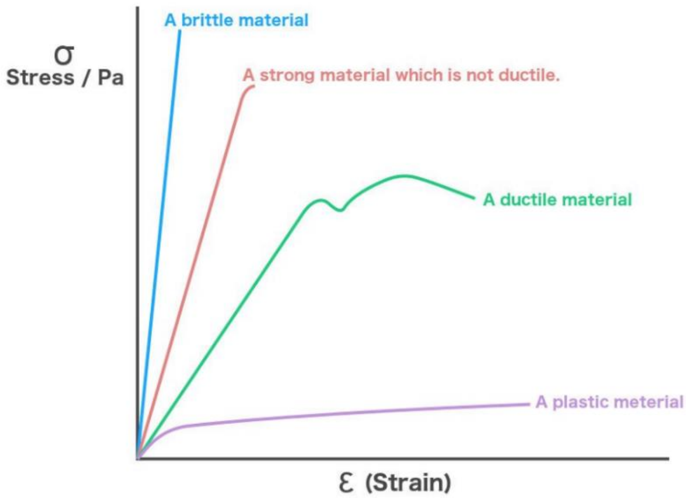


There are many ways of discretizing an object. Triangles, squares, pentagons, and even multi-dimensional objects may be defined as the discretization method. One-dimensional elements will include points on a line. Two dimensional elements may be planes, shells, membranes, and geometric shapes. Three dimensional elements may include cubes, tetrahedrons, triangular prisms, and pyramidal structures. It is important to note that discretization strategies for FEA can leave no empty space between elements. Dimensionality and shape are not the only choices to be made when choosing a discretization strategy. We must also choose a distribution of elements within a mesh. Structured meshes are meshes in which all vertices are topologically alike. They are used for simplicity and easy data access. Unstructured meshes are meshes in which vertices have arbitrarily varying local neighborhoods. In other words, the vertices are adapted to the object itself and are not uniform. A hybrid mesh is one with small patches of structured meshes which are all combined to form an unstructured mesh. The size of the mesh also plays an important role in discretization strategy. The accuracy of our findings depends on the number of elements we choose to have in the mesh. The smaller we make our elements, the larger the number of elements. In the same way we can describe a curve by breaking it up into smaller lines, we can approximate the shape of our object with smaller units. The smaller the element, the more accurate our predictions may be. This however may create longer runtimes for the FEA software. Depending on the shape of the object, the discretization method must be chosen adequately.



3.12 Object Materials

FEA analysis also allows one to define the materials of an object in order to properly calculate physical forces. For example, it is known that glass and metal have different elasticities than each other, and thus one is less likely to break than the other. Stress and s train are important



physical properties in the world of engineering. Stress is the deforming force per unit of area while strain is the actual change in shape of the original object. The graph below shows the relationship between stress and strain when different object materials are chosen. For example, we can note that brittle material undergoes a lot of stress with little strain while a fairly ductile material such as plastic undergoes a lot of strain at little stress. By choosing the materials in the FEA software, the calculations will more likely simulate the actual physical object.

$$\text{Stress } (\sigma) = \frac{Force\ (F)}{Area(A)} \qquad \text{Strain}(\epsilon) = \frac{Change\ in\ length(\Delta l)}{Original\ length(l)}$$

3.13 Solution Approximations

Although in practice, FEA software allows the user to define boundaries and alter the mesh pattern, the mathematics behind this software is often "sanity checked" by engineers. Let us assume we have a quadrilateral shape with an outer boundary Γ and inner region Ω.

The first step in the Finite Element Method is to establish a strong equation. A strong form is a conventional differential equation, while the weak form is an alternate representation of the differential equation. While strong form solutions must be continuous and differential, weak form solutions have larger sets of functions. Let us say our strong form differential equation for this particular problem is a form of the Poisson equation.

$$\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} = \phi \quad (+\, boundary\ conditions)$$

Let us assume a particular $u^e(x, y)$ is an exact solution to the equation:

$$\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} - \phi = 0$$

This exact solution is almost impossible to determine in the real world. Thus, an approximate solution must be found. Let $\hat{u}$ be this approximate solution. $\hat{u}$ will not satisfy the Poisson equation exactly, and thus a residual will be left over.

$$\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} - \phi \neq 0$$

$$Residual\ R = \frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} - \phi$$

Thus, we need to find an approximate solution $\hat{u}$ such that R is close to zero at each point within Ω. In order to find an R close to zero, we must find the weighted residual statement, otherwise known as the weak formulation. By integrating the equation over the domain, we can find our weak form:

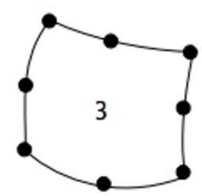
$$\int W(x,y)(\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 u}{\delta y^2} - \phi)\, \delta\Omega = 0$$

$$\int W(x,y)(\nabla^2 \hat{u} - \phi)\, \delta\Omega = 0$$

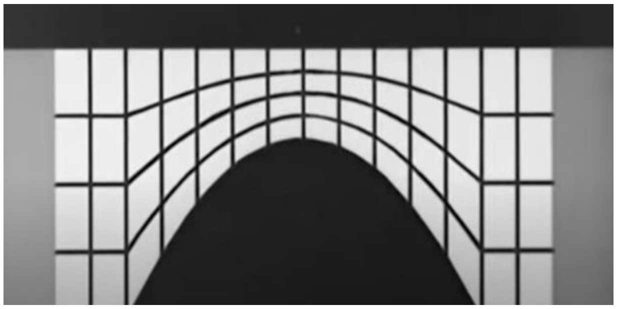
$$\int W \nabla \hat{u} \hat{n}\, d\Gamma - \int \nabla W \nabla \hat{u}\, d\Omega - \int W \phi\, d\Omega = 0$$

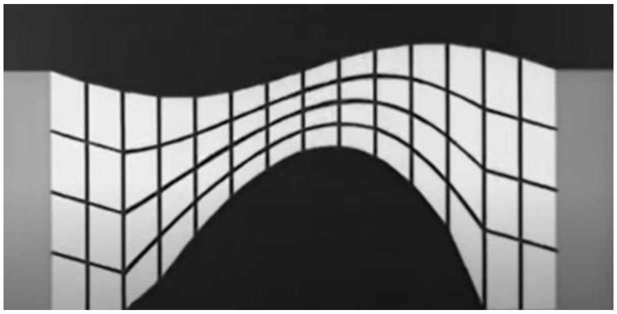
3.2 Solution Algorithm

With the discretization strategy selected, the governing equations for the physical property being tested are applied to each element. By calculating the physical property for each element using the nodes, a system of equations can be found. Let us look at an example object with 2d quadrilateral nodes in which we want to apply external pressure on. As can be seen, the element has 8 nodes. Because this is a 2d element, when external pressure is applied, each node can be displaced horizontally or vertically. Certain nodes may be fixed if other structures will hold this part of the object in place, assuring that they will not be displaced by the external applied pressure.

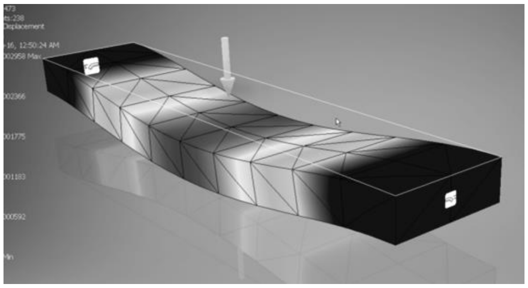


8 Node Quadrilateral





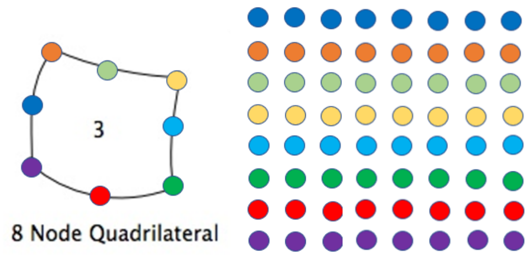
An example of such a system is a 2d bridge. If the bridge cannot expand towards the left or the right because rock structures are exerting their own pressure on the bridge, the nodes on the left and right boundaries can be set as fixed nodes. All other nodes can be displaced if the conditions applied on the object make the object change structure.



Let us look at another example for further clarification. If a load is placed on an object at a particular point, the FEA will use a system of equations which can tell us about the overall structural changes caused by the downwards force.

Each node has an equation that describes its displacement as a function of its coordinates:

$$Displacement\ of\ Node\ 1 = a + bx + cy + dxy + ex^2 + fy^2 + gx^2y + hxy^2$$

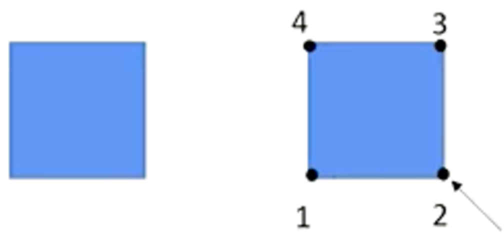


If we look back at our quadrilateral element, we can derive eight equations because it has eight nodes. These eight equations then form a matrix. The fundamental laws of mechanics are then applied to the matrix. Although displacement is the primary unit of measurement, stress, strain, and energy are all derived from displacement. Displacement is related to stress. Stress is related to strain, and thus potential energy can be described. Since FEA is used to determine whether an object design is stable, potential energy is a very important calculation. If the system is stable, potential energy will be at its minimum.

3.21 The Stiffness Matrix

The system of node equations is referred to as the stiffness matrix, a mathematical topic well discovered by the 1950s. The matrix operates on vectors whose components are the displacements for the whole elements. By looking at the displacement of nodes relative to their position, stress contours can be determined. This process is carried on for every element in the mesh. Every stiffness matrix is then combined into a single matrix representing the stiffness of the whole system using matrix reduction. Because nodes for two elements are shared, the values will be found in other matrices. In order to simplify the matrix, the equations are solved. Each equation is then substituted into the remaining ones. This continues until we are left with a solution for a single node. This result is then used to find the values for all other nodes.

3.22 Matrix Math



We earlier defined $\hat{u}$ to be a particular approximation solution to a differential equation describing a quadrilateral shape. Let us isolate one element with function $\hat{u}_i$. This function can be represented as an array of the element's nodes.

$$\hat{u}_i(x,y) = \{\hat{u}\}_i = \begin{pmatrix} 1 \\ u_1 \\ \pi_2 \\ \frac{\hat{n}_3}{u_4} \end{pmatrix}$$

Each node has its own equations that can be put into matrix form. In order to represent $\hat{u}(x,y)$ as a matrix, Piecewise extrapolation can be used.

$$\hat{u}(x,y) = N_1(x,y)\hat{u}_1 + N_2(x,y)\hat{u}_2 + N_3(x,y)\hat{u}_3 + N_4(x,y)\hat{u}_4$$
$$\hat{u}(x,y) = \sum N_k u_k = [N]\{u\}$$

N_1(x,y), N_2(x,y), N_3(x,y), and N_4(x,y) are the known basis functions that depend on the shape of the element. These basis functions are multiplied by the coefficients of the nodes and added to form the solution. Basis functions can be polynomials of any degree; they describe the shape, dimension, and size of the mesh elements. The solutions, $\hat{u}$, are the unknown solutions we are solving for. By substituting $\hat{u}$ into its weak form, a system of equations can be found.

$$\int W \nabla \hat{u} \hat{n}\, d\Gamma - \int \nabla W \nabla \hat{u}\, d\Omega - \int W f\, d\Omega = 0$$
$$[k]_i\{u\}_i = \{f\}_i$$

We have only calculated one element and transformed it into a linear form. This process must be repeated across all elements and then assembled by adding all of the $\hat{u}$ s together.
After adding all the elements, we are left with the following equation known as Hooke's Law:

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

K becomes our stiffness matrix, F is the global force, and U are the node values. Thus, with all the $\hat{u}$'s in the domain, contour lines can be defined.

3.3 Post Processing

The post processing step serves both to check errors throughout the solution algorithm and organize the data into its final presented form.

3.1 Error Detection

Error Detection is generally done quickly by hand ("sanity checks") or processed through post-processing software. In order to detect errors, we first must check the factors that affect accuracy the most. Because our approximations depend on the mesh we created, mesh convergency must be checked. In order to verify that the size of our elements is adequate, we must verify that increasing the quantity of elements will not drastically affect the solution. In other words, we must verify that the definition of approximations is held. Remembering that u^e (x, y) is our exact solution function and $\hat{u}$ is our approximate solution function, then:

$$\lim_{x\to\infty, y\to\infty} \mathrm{u^e(x,y)} - \mathrm{\hat{u}(x,y)} = 0$$

Since we do not know u^e (x, y), this can be rewritten such that:

$$\lim_{x\to\infty, y\to\infty} \mathrm{\hat{u}(x,y)} = C$$

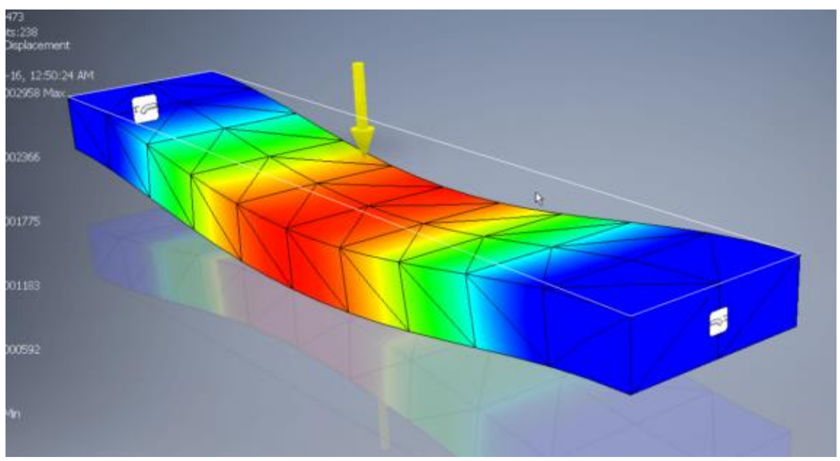
where C is some solution. The solution must converge to a specific value as the number of elements goes to infinity. In order to check this convergence solution, experimental results can be used or comparisons with other analytical solutions.

Errors can also be found in displacement, strains, and stresses. These errors are found by comparing the predicted value to the actual value.

$$Displacement\ Errors: e_u = u - u^h$$
$$Strain\ Errors: e_\epsilon = \epsilon - \epsilon^h$$
$$Stress\ Errors: e_\sigma = \sigma - \sigma^h$$

3.2 Data Organization

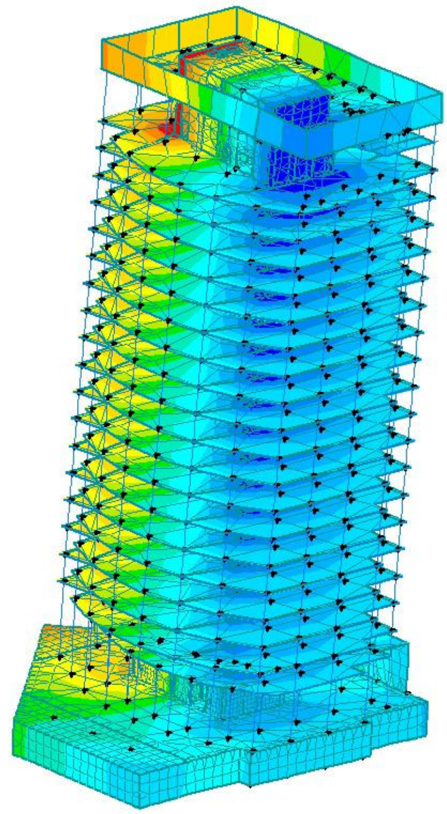
Plotting the results depends on the use and the software. Contour lines are often used to show points of strain or stress. In the image to the right, the point of added force is shown along with the magnitude of displacement. Animations of an object impacted by some force also may assist visualization.



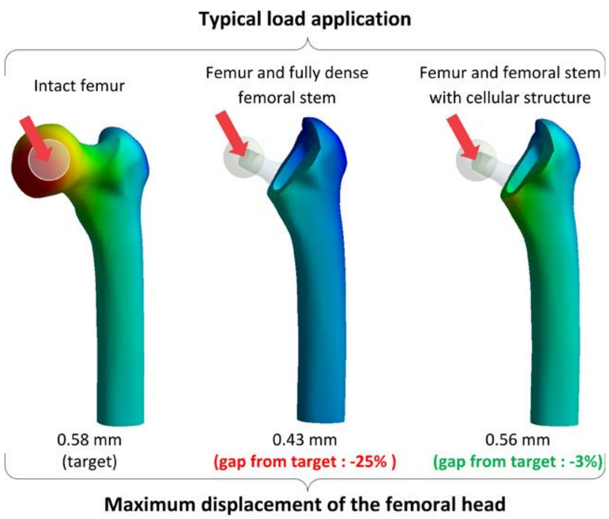
4 Applications

Finite Element Analysis has a varied use throughout the fields of science and engineering. FEA is used to solve Thermo-mechanical, thermo-chemical, ferroelectric, electromagnetic, fluid structure, flow analysis, structural analysis problems.

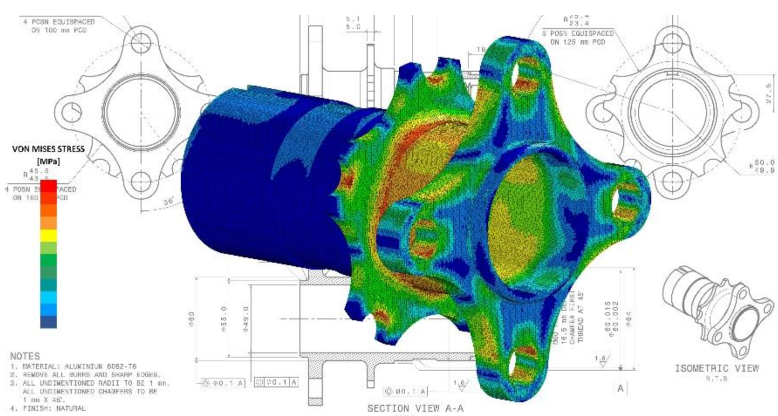
Civil Engineers and geologists may use FEA to test building materials and structures, sedimentation, erosion, hydrology, slope stability, and seepage of fluids in soils and rocks.



Bioengineers may use FEA to simulate human organs and test medical devices and implants.

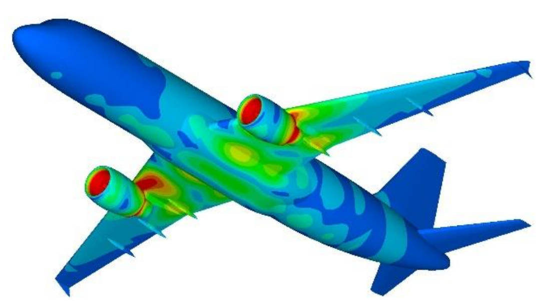


Mechanical engineers may use FEA in manufacturing process simulation, automotive design, heat analysis in electrical equipment, and insulation design analysis.



Meteorologists may use FEA for climate, wind, and monsoon predictions.

And of course, simulation software using FEA must be created by software engineers and computer scientists!



5 References

6, praveenDecember, et al. “Correct Mesh Size - a Quick Guide!” *Enterfea*, 16 Sept. 2019, enterfea.com/correct-mesh-size-quick-guide/.

Admin, and Mohamed Zaki says: “Difference Between Stress and Strain.” *Physicsabout*, 3 Oct. 2020, physicsabout.com/stress-and-strain/.

Associates, Roensch &. “FINITE ELEMENT ANALYSIS: Post-Processing.” *Finite Element Analysis: Post-Processing*, www.finiteelement.com/feawhite4.html.

Bern, Marshall, and Paul Plassmann. “Mesh Generation.” *UCIICS*, University of California, Irvine, www.ics.uci.edu/~eppstein/280g/Bern-Plassman-meshgen.pdf.

Caendkoelsch. “What Are Strong and Weak Forms in Finite Element Analysis (FEA)? Why Do We Need Them?” *Caendkölsch*, 9 Oct. 2018, caendkoelsch.wordpress.com/2018/05/24/what-are-strong-and-weak-forms-in-finite-element-analysis-fea-why-do-we-need-them/.

“Finite Element Method (FEM) - Finite Element Analysis (FEA): Easy Explanation.” *YouTube*, YouTube, 20 Mar. 2017, www.youtube.com/watch?v=aLJMDn_2-d8.

“Finite Element Method.” *Wikipedia*, Wikimedia Foundation, 27 Nov. 2020, en.wikipedia.org/wiki/Finite_element_method.

“Introduction to Finite Element Analysis (FEA) or Finite Element Method (FEM).” *UVIC*, University of Victoria Engineering, www.engr.uvic.ca/~mech410/lectures/FEA_Theory.pdf#:~:text=Introduction%20to%20Finite%20Element%20Analysis%20%28FEA%29%20or%20Finite,properties%20where%20analytical%20solutions%20can%20not%20be%20obtained.

“Introduction to Finite Element Method (FEM) for Beginners.” *YouTube*, YouTube, 20 July 2020, www.youtube.com/watch?v=C6X9Ry02mPU.

“Lec 1: MIT Finite Element Procedures for Solids and Structures, Linear Analysis.” *YouTube*, 10 June 2011, youtu.be/oNqSzzycRhw.

Lewis, Naama T.L. “Finite Element Analysis: Mathematical Theory and Applications.” *OpenSIUC*, Southern Illinois University Carbondale, 2011, opensiuc.lib.siu.edu/cgi/viewcontent.cgi?article=1226&context=gs_rp.

Mathematics of the Finite Element Method, math.nist.gov/mcsd/savg/tutorial/ansys/FEM/.

“Overview of Finite Element Method (FEM).” *YouTube*, 25 Apr. 2016, youtu.be/aX4_jIYpQbI.

Pointer, Jon. *Understanding Accuracy and Discretization Error in an FEA Model* . Woodward Governor Company, support.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/confpaper/2004-Int-ANSYS-Conf-54.PDF.

"What Is FEA: Finite Element Analysis? SimScale Documentation." *SimScale*, 10 Nov. 2020, www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-is-fea-finite-element-analysis/.

Pictures

Hanche6_ENG.jpg (645×552) (etsmtl.ca)

https://blog.spatial.com/hubfs/AdobeStock_296970902.jpeg

high-rise-building-3.jpg (465×784) (scia.net)

shell-2b-1.png (719×426) (enterfea.com)

https://www.google.com/url?sa=i&url=https%3A%2F%2Fwww.diva-portal.org%2Fsmash%2Fget%2Fdiva2%3A1222037%2FFULLTEXT02.pdf&psig=AOvVaw0B4yc0Jxvwo3oV85mJ3kIC&ust=1606854126811000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCOjj2p2Mq-0CFQAAAAAdAAAAABAJ

https://www.google.com/url?sa=i&url=https%3A%2F%2Fwww.quora.com%2FIf-a-stress-strain-graph-does-not-pass-through-the-origin-then-on-which-side-should-a-correction-be-applied-stress-or-strain&psig=AOvVaw1-9rcKbrixvWEHvLk8ABxa&ust=1606854530716000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCLCBpN6Nq-0CFQAAAAAdAAAAABAD

https://www.google.com/url?sa=i&url=https%3A%2F%2Flindenreid.wordpress.com%2F2018%2F02%2F24%2Fintro-to-procedural-geometry-part-2%2F&psig=AOvVaw0aLCUWETYJnbmHmskzJUKs&ust=1606854567707000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCOD-v--Nq-0CFQAAAAAdAAAAABAK

https://www.google.com/url?sa=i&url=https%3A%2F%2Fe3k.com%2Ffinite-element-analysis%2F&psig=AOvVaw1moNzkx2vT1wrD4I3rpmn6&ust=1606854607044000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCLiNvoKOq-0CFQAAAAAdAAAAABAD

https://www.google.com/url?sa=i&url=http%3A%2F%2Fwww.todohumor.com%2Fpowerpoint%2F178%2F&psig=AOvVaw1XXcHzSogETC82TFcb0_sx&ust=1606854674192000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCNDKxKKOq-0CFQAAAAAdAAAAABAD

https://www.google.com/url?sa=i&url=https%3A%2F%2Fdynamiccoachingcentre.co.in%2Ffinite-element-

analysis.php&psig=AOvVaw2ZQsFQEbUpyhXS6YLz273g&ust=1606854742943000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCPCOqcOOq-0CFQAAAAAdAAAAABAJ

https://www.google.com/url?sa=i&url=https%3A%2F%2Fopenfoamwiki.net%2Findex.php%2F2D_Mesh_Tutorial_using_GMSH&psig=AOvVaw0DIwnk_UTBI1YlS3pR3JiF&ust=1606854811630000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCIj5neSOq-0CFQAAAAAdAAAAABAD

https://www.google.com/url?sa=i&url=https%3A%2F%2Fwww.comsol.it%2Fmultiphysics%2Fmesh-refinement&psig=AOvVaw0UEiDivcQBd5MJCgdsSBch&ust=1606854862416000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCND8yfyOq-0CFQAAAAAdAAAAABAD