International Journal of Innovation Sciences and Research
Vol.4, No, 7, pp.285-292, July- 2015

Available online at
http://www.ijisr.com

RESEARCH ARTICLE

A STUDY ON THE FINITE ELEMENT METHOD: AN ESSENTIAL TOOL FOR ENGINEERING ANALYSIS

***, [1]Wadzani Samuel Gadzama, [2]Nonyelu Helen Uzoamaka[3]Lilian Nneka Ejike-Uguwu and [4]Okwudiba Ifeanyi Nwachukwu**

[1]Department of Training, Scientific Equipment Development Institute, Enugu, Nigeria
[2]Department of Electrical R & D, Scientific Equipment Development Institute, Enugu, Nigeria
[3]Department of Advanced Manufacturing, Scientific Equipment Development Institute, Enugu, Nigeria
[4]Department of Training, Scientific Equipment Development Institute, Enugu, Nigeria

Accepted 24th June, 2015; Published Online 31st July 2015

ABSTRACT

The method of finite element is a numerical technique that solves or at least approximates enough to a system of differential equations related with a physical or engineering problem. This study looked into the usefulness of the finite element method as a gainful tool for engineering analysis; it equally seeks to widen the horizon of analysts on the use of this useful analytical tool. The FEM provides a standard process for converting governing energy principles or governing differential equations in to a system of matrix equations to be solved for an approximate solution. For linear problems such solutions can be very accurate and quickly obtained. Having obtained an approximate solution, the FEM provides additional standard procedures for follow up calculations (post-processing), such as determining the integral of the solution, or its derivatives at various points in the shape. When the FEM is applied to a specific field of analysis (like stress analysis, thermal analysis, or vibration analysis) it is often referred to as finite element analysis (FEA). FEA is the most common tool for stress and structural analysis.

Key Words: Finite element method (FEM), Finite element analysis (FEA), Boundary value problems (BVP), Degree of freedom (DOF), Computer Aided Design (CAD).

INTRODUCTION

The finite element method is one of the most powerful numerical techniques ever devised for solving differential and integral equations of initial and boundary-value problems (BVP) in geometrically complicated regions. There is some data that cannot be ignored when analyzing an element by the finite element method. This input data is to define the domain, the boundary and initial conditions and also the physical properties. After knowing this data, if the analysis is done carefully, it will give satisfactory results. It can be said that the process to do this analysis is very methodical, and that it is why it is so popular, because that makes it easier to apply. "The finite element analysis of a problem is so systematic that it can be divided into a set of logical steps that can be implemented on a digital computer and can be utilized to solve a wide range of problems by merely changing the data input to the computer program (Felippa *et al.,* 2006). Two fundamental concepts birth the advancement in the FEM approach. In 1963 it was shown that the FEM was a variation of Raleigh-Ritz Method (which produces a set of linear equations by minimizing the potential energy of the system). This lead to its application in different areas of heat transfer, fluid flow, etc. Also, in 1969 it was shown that element equations could also be derived using a weighted residual procedure such as Galerkin's Method or the least squares approach. This allows application to any BVP and therefore enlarged its use and application (Felippa *et al.,* 2006).

****Corresponding author: Samuel Wadzani Gadzama,***
Department of Training, Scientific Equipment Development Institute, Enugu, Nigeria.

When FEM started in the early seventies highly qualified and trained specialists were needed to use this method. Neither graphical preprocessing nor matured FEM-tools were available. Too, the processing speeds of the computers were low. From the viewpoint of today rather small problems required profound knowledge in mathematics, software development, and technical engineering. The effort needed for solving FEM problems was too high even to think about commercial application of FEM in industrial companies (Lee, 2009). Increase of the processing speed of computers and development of first graphic oriented preprocessors opened the door of the FEM into industrial companies. However, the handling of the programs retained difficult and still required highly trained and educated personal. Thus, this method was restricted to only to larger which could afford their own calculation group (Christoph, 2013).

Though CAD systems became convenient and widely spread tools in the same time, there was lack of integration between CAD- and FEM-systems. Data transfer was done via interfaces of very different quality. Not unusual, a complete scratch-up of the model was necessary prior calculation. The model had to be simplified in order to facilitate meshing and reduce computing time. Most often, weeks and months went by between the completion of the CAD model and the results of the calculation (Saad, 2011). Several attempts to integrate the designer into the calculations process were made. They failed because of great differences between the CAD and FEM-systems and difficulties in handling of FEM systems. The acceptance of these non-integrated FEM tools was very low.

The situation improved with the appearance of first integrated systems. Difficult geometry and data transfer into the preprocessor or rebuilding of the model was no longer necessary. However, the handling of the FEM tools remained complicated and required highly trained specialists (Saad, 2011). Actually, the situation has improved drastically. CAD system developers designed FEM menu structures especially tailored to be used by designers. The main goal was easy handling to solve standard problems (Saad, 2011).

Fundamental Concept of FEM

Any continuous quantity, such as temperature, pressure, or displacement, can be approximated by a discrete model composed of a set of piecewise continuous functions (polynominals) defined over a finite number of subdomains or elements. The finite element analysis can be done for one, two and three-dimensional problems. But generally, the easier problems are those including one and two dimensions and those can be solved without the aid of a computer, because even if they give a lot of equations, if they are handled with care, an exact result can be achieved. But if the analysis requires three-dimensional tools, then it would be a lot more complicated, because it will involve a lot of equations that are very difficult to solve without having an error. That is why engineers have developed softwares that can perform these analyses by computer, making everything easier. These softwares can make analysis of one, two and three dimensional problems with a very good accuracy (Song, 2009). The finite element method (FEM) rapidly grew as the most useful numerical analysis tool for engineers and applied mathematicians because of it natural benefits over prior approaches. The main advantages are that its can be applied to arbitrary shapes in any number of dimensions. The shape can be made of any number of materials. The material properties can be non-homogeneous (depend on location) and/or anisotropic (depend on direction). The way that the shape is supported (also called fixtures or restraints) can be quite general, as can the applied sources (forces, pressures, heat flux, etc.). The FEM provides a standard process for converting governing energy principles or governing differential equations into a system of matrix equations to be solved for an approximate solution.

For linear problems such solutions can be very accurate and quickly obtained (Sprague and Geers, 2007). Having obtained an approximate solution, the FEM provides additional standard procedures for follow up calculations (post-processing), such as determining the integral of the solution, or its derivatives at various points in the shape. The post-processing also yields impressive color displays, or graphs, of the solution and its related information. Today, a second post-processing of the recovered derivatives can yield error estimates that show where the study needs improvement. Indeed, adaptive procedures allow automatic corrections and re-solutions to reach a user specified level of accuracy. However, very accurate and pretty solutions of models that are based on errors or incorrect assumptions are still wrong (Wolf, 2003). When the FEM is applied to a specific field of analysis (like stress analysis, thermal analysis, or vibration analysis) it is often referred to as finite element analysis (FEA). FEA is the most common tool for stress and structural analysis. Various fields of study are often related. For example, distributions of non-uniform temperatures induce non-obvious loading conditions on solid structural members. Thus, it is common to conduct a thermal FEA to obtain temperature results that in turn become input data for a stress FEA. FEA can also receive input data from other tools like motion (kinetics) analysis systems and computation fluid dynamic (CFD) systems (Lee, 2009).

Basic concept of Integral Formulations

The basic concept behind the FEM is to replace any complex shape with the union (or summation) of a large number of very simple shapes (like triangles) that are combined to correctly model the original part. The smaller simpler shapes are called finite elements because each one occupies a small but finite sub-domain of the original part as explained earlier. They contrast to the infinitesimally small or differential elements used for centuries to derive differential equations. To give a very simple example of this dividing and summing process, consider calculating the area of the arbitrary shape shown in Figure 1 (left). If the user knew the equations of the bounding curves, in theory, could integrate them to obtain the enclosed area (Akin, 2009).

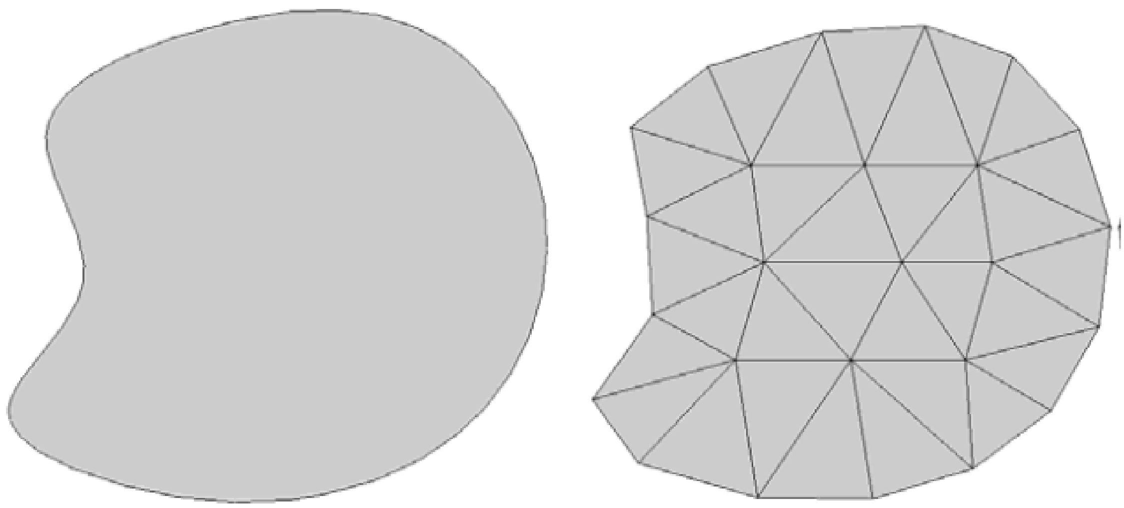

Figure 2. An area crudely meshed with linear and quadratic triangles

On the Alternative, split the area into an enclosed set of triangles (cover the shape with a mesh) and sum the areas of the individual triangles as expressed by equation 1 below:

$$A = \sum_{e=1}^{n} A^e = \sum_{e=1}^{n} \int_{A^e} dA \qquad \text{………………..} \; 1$$

Now, one has some choices for the type of triangles. The engineer could pick straight sided (linear) triangles, or quadratic triangles (with edges that are parabolas), or cubic triangles, etc. The area of a straight-sided triangle is a simple algebraic expression. The area of a curved triangle is relatively easy to calculate by numerical integration, but is computationally more expensive to obtain than that for the linear triangle. The first two triangle mesh choices are shown in Figure 1 for a large element size. Clearly, the simple straight-sided triangular mesh (on the left) approximates the area very closely, but at the same time introduces geometric errors along the boundary. The boundary geometric error in a linear triangle mesh results from replacing a boundary curve by a series of straight line segments. That geometric boundary error can be reduced to any desired level by increasing the number of linear triangles. But that decision increases the number of calculations and makes one to trade off geometric accuracy versus the total number of required area calculations and summations (Akin, 2009). Area is a scalar, so it makes sense to be able to simply sum its parts to determine the total value, as shown above. Other topics, like kinetic energy or strain energy, can be summed in the same fashion. Indeed, the very first applications of FEA to structures was based on minimizing the energy stored in a linear elastic material. The FEM always involves some type of governing integral statement. That integration is also converted to the sum of the integrals over each element in the mesh. Even if one starts with a governing differential equation, it gets converted to an equivalent integral formulation by one of the methods of weighted residuals (MWR). The two most common methods, for FEA, are the Galerkin Method and the Method of Least Squares Figure (Akin, 2009).

Thus, the boundary shape error is indeed reduced, at the expense of more complicated area calculations, but it is not eliminated. Some geometric error remains because most engineering curves are circular arcs, splines, or nurbs (non-uniform rational B-splines) and thus are not matched by a parabola. The most common way to reduce mesh geometric error is to simply use smaller elements, as shown in Figure 2 below. For instance, the default element choice in Solid Works Simulation is the quadratic element. Other systems offer a wider range of edge polynomial degree (e.g. cubic), as well as other shapes like quadrilaterals or rectangles. In three-dimensional solid applications some systems offer dozens of choices for the edge degree polynomial order, and shapes including hexahedral, wedges, and tetrahedral elements. Hexahedral elements are generally more accurate, but can be more challenging to mesh automatically. Tetrahedral elements can match hexahedral element performance by using more (smaller) elements, and tetrahedral elements are much easier to mesh automatically (Akin, 2009). An example of the small two-dimensional geometric boundary error due to different curved shapes is seen in Figure 3 where a circular arc and a parabola pass through the same three points. (A new method, called geometric analysis, can essentially eliminate all geometric errors, but it introduces new approximations in other study stages, such as in the restraint conditions.)

Stages of Analysis and Their Uncertainties

The typical stages of a FEA study are classified into three main stages

i. Build the model
ii. Solve the model
iii. Display the results

However, these three main stages are further sub-divided in this paper for more clarity and better understanding in simple stepwise approach that can conveniently guide the user of the FEM in running analysis:

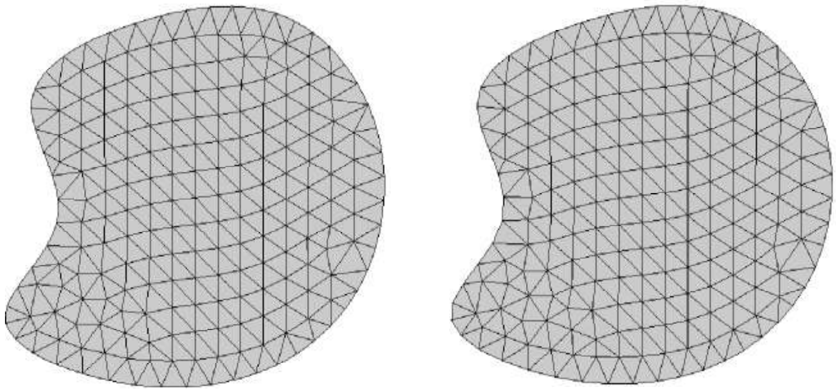

Figure 2. Mesh refinement quickly reduces geometric boundary errors for linear (left) or quadratic elements

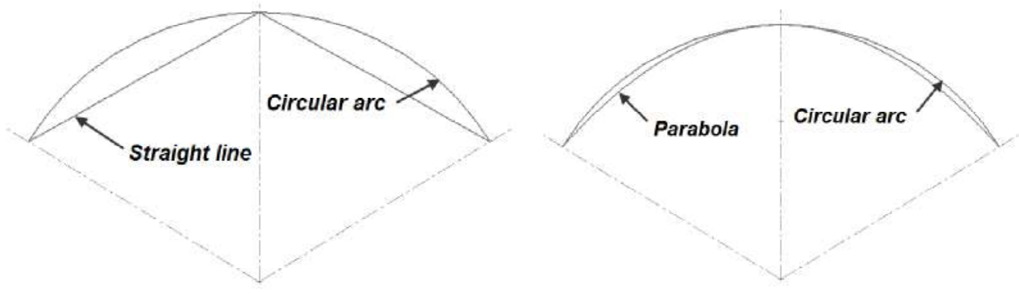

Figure 3. Linear or parabolic elements never exactly match circular shapes

- Firstly, construct the part(s) in a solid modeler. It is surprisingly easy to accidentally build flawed models with tiny lines, tiny surfaces or tiny interior voids. The part will look fine, except with extreme zooms, but it may fail to mesh. Most systems have checking routines that can find and repair such problems before one move on to a FEA study. Sometimes a user may have to export a part, and then import it back with a new name because imported parts are usually subjected to more time consuming checks than "native" parts. When multiple parts form an assembly, always mesh and study the individual parts before studying the assembly. Try to plan ahead and introduce split lines into the part to aid in mating assemblies and to locate load regions and restraint (or fixture or support) regions. Of late, construction of a part is probably the most reliable stage of any study.

- Defeature the solid part model for meshing. The solid part may contain features, like a raised logo, that are not necessary to manufacture the part, or required for an accurate analysis study. They can be omitted from the solid used in the analysis study.

- Then, combine multiple parts into an assembly. Again, this is well automated and reliable from the geometric point of view and assemblies "look" as expected. However, geometric mating of part interfaces is very different for defining their physical (displacement, or temperature) mating. The physical mating choices are often unclear and the engineer may have to make a range of assumptions, study each, and determine the worst case result. Having to use physical contacts makes the linear problem require iterative solutions that take a long time to run and might fail to converge.

- Select the element type. Some FEA systems have a huge number of available element types (with underlying theoretical restrictions). The Solid Works system for example, has only the fundamental types of elements. Namely, truss elements (bars), frame elements (beams), thin shells (or flat plates), thick shells, and solids. The system selects the element type (beginning in 2009) based on the shape of the part. The user is allowed to covert a non-solid element region to a solid element region, and vice versa. Knowing which class of element will give a more accurate or faster solution requires training in finite element theory. At times a second element type study is used to help validate a study based on what is thought to be the best element type.

- Mesh the part(s) or assembly. Remember that the mesh solid may not be the same as the part solid is an important guide. A general rule in FEA is that the computer never has enough speed or memory. Sooner or later the system will find a study that it cannot execute. Often at times, this means that one must utilize a crude mesh (or at least crude in some region) and/or invoke the use of symmetry or anti-symmetry conditions. Local solution errors in a study are proportional to the product of the element size and the gradient of the secondary variables (i.e., gradient of stress or heat flux).

- Assign a linear material to each part. Modern FEA systems have a material library containing the "linear" mechanical, thermal, and/or fluid properties of most standardized materials. They also allow the user to define custom properties. The values in such tables are often misinterpreted to be more accurate and reliable than they actually are. The reported values are accepted average values taken from many tests. Rarely are there any data about the distribution of test results, or what standard deviation was associated with the tests. Most tests yield results that follow a "bell shaped" curve distribution, or a similar skewed curve. Material data are usually more reliable than the loading values (considered next), but less accurate that the model or mesh geometries.

- Select regions of the part(s) to be loaded as well as assigning load levels and load types to each region. In mathematical terminology, load or flux conditions on a boundary region are called Neumann boundary conditions, or non-essential conditions. The geometric regions can be points (in theory), lines, surfaces, or volumes. If they are not existing features of the part, apply split lines to the part to create them before activating the mesh generator. Point forces, or heat sources, are common in undergraduate studies, but in a FEA they cause false infinite stresses, or heat flux.

Saint Venant's Principle states that two different, but statically equivalent force systems acting on a small portion of the surface of a body produce the same stress distributions at distantness large in comparison with the linear dimensions of the portion where the forces act as shown in the Figure 4 below. Assign a contribution to the total factor of safety to allow for variations in the uncertainty of the load value or actual spatial distribution of applied loads. Loading data are usually less accurate than the material data, but much more accurate that the restraint or supporting conditions considered next.

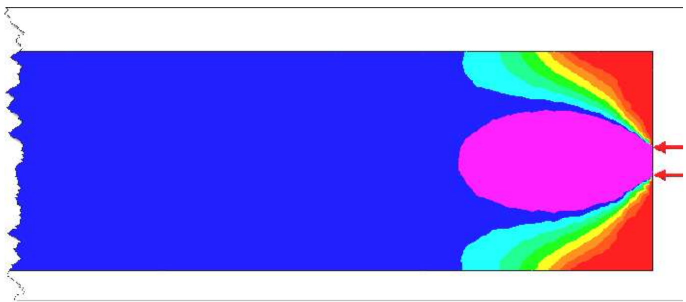

Figure 4. St. Venant's principle: local effects quickly die out

- Determine (or more likely assume) how the model interacts with the surroundings not included in the design model. These are the restraint (support, or fixture) regions. In mathematical terminology, these are called the essential boundary conditions, or Dirichlet boundary conditions. It is almost impossible to model everything interacting with a part. For many decades engineers have developed simplified concepts to approximate surroundings adjacent to a model to simplify hand calculations. They include roller supports, smooth pins, cantilever (encast, or fixed) supports, straight cable attachments, etc.

Those concepts are often carried over to FEA approaches and can over simplify the true support nature and lead to very large errors in the results. The engineer needs to assign a contribution to the total factor of safety to allow for variations in the uncertainty of how or where the actual support conditions occur.

- Solve the linear system of equations, or the eigen value problem. With today's numerical algorithms the solution of the algebraic system or eigen-system is usually quite reliable. It is possible to cause ill conditioned systems (large condition number) with meshes having large elements adjacent to small ones, but that is unlikely to happen with automatic mesh generators.

- Check the results. Are the reactions at the supports equal and opposite to the supposed sources you thought that you applied? Are the results consistent with the assumed linear behavior? The engineering definition of a problem with large displacements is one where the maximum displacement is more than half the smallest geometric thickness of the part. The internal definition is a displacement field that significantly changes the volume of an element. That implies the element geometric shape noticeably changed from the starting shape, and that the shape needs to be updated in a series of much smaller shape changes. Are the displacements big enough to require re-solution with large displacement iterations turned on?

- Post-process the solution for secondary variables. For structural studies the engineer should generally wish to document the deflections and stresses. For thermal studies, display the temperatures and heat flux vectors. With natural frequency models you show (or animate) a few mode shapes. Control the number of contours employed, as well as their maximum and minimum ranges. The latter is important if the desire is to compare two designs on a single page. Limit the number of digits shown on the contour scale to be consistent with the material modulus (or conductivity, etc.). Contour plots often do not reproduce well in a report, but graphs generally do, it is imperative to learn to include graphs in documentation.

- Determine (or more likely assume) what failure criterion applies to the study. This stage involves assumptions about how a structural material might fail. There are a number of theories. Most are based on stress values or distortional energy levels, but a few depend on strain values. If one has been accepted for the selected material then use that one (with a contribution to the overall factor of safety). Otherwise, you should evaluate more than one theory and see which the worst case is. Also keep in mind that loading or support uncertainties can lead to a range of stress levels, and variations in material properties affect the strength and unexpected failures can occur if those types of distributions happen to intersect, as sketched in Figure 5.

- Optionally, post-process the secondary variables to measure the theoretical error in the study, and adaptively correct the solution. This converges to an accurate solution to the problem input, but perhaps not to the problem to be solved. Accurate garbage is still garbage.

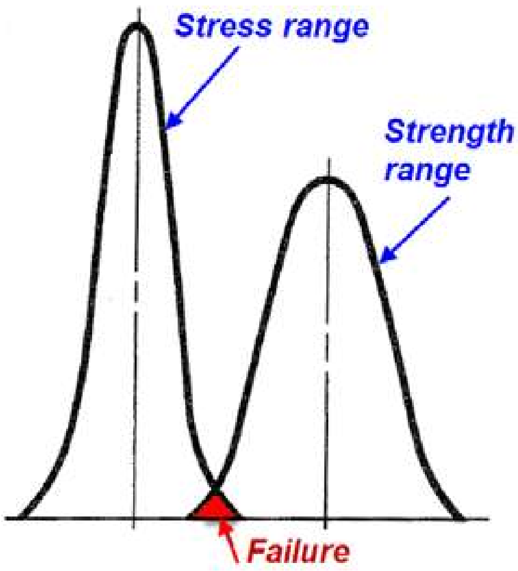

Figure 5. Distributions of loads/restraints and material strengths can cause failure

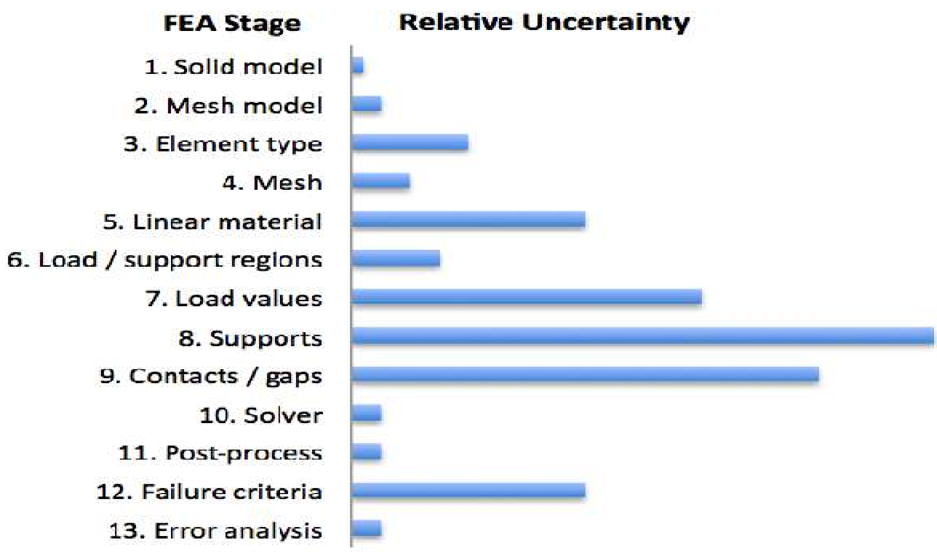

Figure 6. Relative uncertainties of major modeling stages

- Document, report, and file the study. The part shape, mesh, and results should be reported in image form. Assumptions on which the study was based should be clearly stated, and hopefully confirmed. The documentation should contain an independent validation calculation, or two, from an analytical approximation or a FEA based on a totally different element type. Try to address the relative uncertainties of the main analysis stages, as summarized in Figure 6.

The physical significance of the vectors u and f varies according to the application being modeled, as illustrated in Table 1 below. If the relation between forces and displacements is linear but not homogeneous, equation 2 above, generalizes to

$$\mathrm{Ku} = f_M + f_I \quad \ldots\ldots\ldots\ldots 3$$

Where $\boldsymbol{f_I}$ is the initial node force vector for effects such as temperaturechanges, and f_M is the vector of mechanical forces.

Table 1. Significance of u and f in Miscellaneous FEM Applications

Application Problem	State (DOF) vector u represents	Conjugate vector f represents
Structures and solid mechanics	Displacement	Mechanical force
Heat conduction	Temperature	Heat flux
Acoustic fluid	Displacement potential	Particle velocity
Potential flows	Pressure	Particle velocity
General flows	Velocity	Fluxes
Electrostatics	Electric potential	Charge density
Magnetostatics	Magnetic potential	Magnetic intensity

Finite element method and degree of freedom

The ubiquitous term "degrees of freedom," often abbreviated to freedom or DOF, as well as "stiffness matrix" and "force vector," originated in structural mechanics, the application for which FEM was invented. These names have carried over to non-structural applications as discussed below.

Classical analytical mechanics is that invented by Euler and Lagrange in the XVIII century and further developed by Hamilton, Jacobi and Poincar´e as a systematic formulation of Newtonian mechanics. Its objects of attention are models of mechanical systems ranging from material particles composed of sufficiently large number of molecules, through airplanes, to the Solar System. The spatial configuration of any such system is described by its degrees of freedom or DOF. These are also called generalized coordinates. The terms state variables and primary variables are also used, particularly in mathematically oriented treatments (ASEN 5007, 2014). If the number of degrees freedom is finite, the model is called discrete and continuous otherwise. Because FEM is a discretization method, the number of DOF of a FEM model is necessarily finite. They are collected in a column vector called u. This vector is called the DOF vector or state vector. The term nodal displacement vector for u is reserved to mechanical applications (ASEN 5007, 2014). In analytical mechanics, each degree of freedom has a corresponding "conjugate" or "dual" term, which represents a generalized force (Abramowitz and Stegun, 1993). In non-mechanical applications, there is a similar set of conjugate quantities, which for want of a better term are also called forces or forcing terms. They are the agents of change. These forces are collected in a column vector called f. The inner product $f^T u$ has the meaning of external energy or work (ASEN 5007, 2014). The relation between u and f is assumed to be of linear and homogeneous. The last assumption means that if u vanishes so does f. The relation is then expressed by the master stiffness equations:

$$Ku = f \quad \ldots\ldots\ldots\ldots 2$$

K is universally called the stiffness matrix even in non-structural applications because no consensus has emerged on different names.

Idealization

Idealization passes from the physical system to a mathematical model. This is the most important step in engineering practice, because it cannot be "canned." It must be done by a human. The word "model" has the traditional meaning of a scaled copy or representation of an object. And that is precisely how most dictionaries define it. The term is used here in a more modern sense, which has become increasingly common since the advent of computers:

A model is a symbolic device built to simulate and predict aspects of behavior of a system. ……… 4

Note the distinction made between behavior and aspects of behavior. To predict everything, in all physical scales, you must deal with the actual system. A model aspects of interest to them (Hughes, 2000) the qualifier symbolic means that a model represents a system in terms of the symbols and language of another discipline. For example, engineering systems may be (and are) modeled with the symbols of mathematics and/or computer sciences (Felippa, 2004).

Mathematical Models

Mathematical modeling, or idealization, is a process by which an engineer or scientist passes from the actual physical system under study, to a mathematical model of the system. The process is called idealization because the mathematical model is necessarily an abstraction of the physical reality. To give an example of the choices that an engineer may face, suppose that the structure is a flat plate structure subjected to transverse loading. Here is a non-exhaustive list of four possible mathematical models:

1. A *very thin* plate model based on Von Karman's coupled membrane-bending theory.
2. A *thin* plate model, such as the classical Kirchhoff's plate theory.
3. A *moderately thick* plate model, for example that of Mindlin-Reissner plate theory.
4. A *very thick* plate model based on three-dimensional elasticity.

The person responsible for this kind of decision is supposed to be familiar with the advantages, disadvantages, and range of applicability of each model. Furthermore the decision may be different in static analysis than in dynamics.

Discretization

Mathematical modeling is a simplifying step. But models of physical systems are not necessarily simple to solve. They often involve coupled partial differential equations in space and time subject to boundary and/or interface conditions. Such models have an infinite number of degrees of freedom.

Numerical or Analytical

At this point one faces the choice of going for analytical or numerical solutions. Analytical solutions, also called "closed form solutions," are more intellectually satisfying, particularly if they apply to a wide class of problems, so that particular instances may be obtained by substituting the values of free parameters. Unfortunately they tend to be restricted to regular geometries and simple boundary conditions. Moreover some closed-form solutions, expressed for example as inverses of integral trans forms, may have to be anyway numerically evaluated to be useful (ASEN 5007, 2014). Most problems faced by the engineer either do not yield to analytical treatment or doing so would require a disproportionate amount of effort (Õnate *et al.,* 2004) The practical way out is numerical simulation. Here is where finite element methods enter the scene. To make numerical simulations practical it is necessary to reduce the number of degrees of freedom to a finite number. The reduction is called discretization. The product of the discretization process is the discrete model. Discretization can proceed in space dimensions as well as in the time dimension (ASEN 5007, 2014).

Element Attributes

One can take finite elements of any kind one at a time. Their local properties can be developed by considering them in isolation, as individual entities. This is the key to the modular programming of element libraries. In the Direct Stiffness Method, elements are isolated by the disconnection and localization steps. The procedure involves the separation of elements from their neighbors by disconnecting the nodes, followed by referral of the element to a convenient local coordinate system (Idelsohn *et al*., 2009). After that one can consider generic elements: a bar element, a beam element, and so on. From the standpoint of the computer implementation, it means that one can write one subroutine or module that constructs, by suitable parametrization, all elements of one type, instead of writing a new one for each element instance (ASEN 5007, 2014).

Element Dimensionality

Elements can have intrinsic dimensionality of one, two or three space dimensions. There are also special elements with zero dimensionality, such as lumped springs or point masses. The intrinsic dimensionality can be expanded as necessary by use of kinematic transformations. For example a 1D element such as a bar, spar or beam may be used to build a model in 2D or 3D space.

Element Nodes

Each element possesses a set of distinguishing points called nodal points or nodes for short. Nodes serve a dual purpose: definition of element geometry, and home for degrees of freedom. When a distinction is necessary we call the former geometric nodes and the latter connection nodes. For most elements studied here, geometric and connector nodes coalesce. Nodes are usually located at the corners or end points of elements. In the so-called refined or higher-order elements nodes are also placed on sides or faces, as well as possibly the interior of the element (Olovsson *et al*., 2005).

Element Geometry

The geometry of the element is defined by the placement of the geometric nodal points. Most elements used in practice have fairly simple geometries. In one-dimension, elements are usually straight lines or curved segments. In two dimensions they are of triangular or quadrilateral shape. In three dimensions the most common shapes are tetrahedral, pentahedral (also called wedges or prisms), and hexahedra (also called cuboids or "bricks").

Element Degrees of Freedom

The element degrees of freedom (DOF) specify the state of the element. They also function as" handles" through which adjacent elements are connected. DOFs are defined as the values (and possibly derivatives) of a primary field variable at connector node points. For mechanical elements, the primary variable is the displacement field and the DOF for many (but not all) elements are the displacement components at the nodes.

Conclusion

The major concepts of FEM/FEA have been reviewed along with their history and their major applications. The study is not an all-encompassing textbook so to say, but it comfortably serves as an essential tool towards guiding the hands and mind of a beginner in the use of the FEM in engineering analysis, thus, doubling as tit bit for building the confidence of a beginner in the use of the FEM and aiding the beginners' confidence level to that of an everyday user with adequate practice.

REFERENCES

Felippa, C. A. and O'nate, E. 2006. *Nodally Exact Ritz discretization of 1D diffusion-Absorption and Helmholtz Equations by Variational FIC and Modified Equation Methods*, Comput. Mech.,

Lee, U. 2009. *Spectral Element Method in Structural Dynamics*, Wiley, Singapore.

Christoph W. Eckard, 2013. *Advantages and Disadvantages of FEM Analysis inan Early State of The Design Process.* Creative Engineering Services CES Eckard GmbH Heßbrühlstraße, Pg6.

Saad, Y. 2011. *Numerical Methods for Large Eigen value Problems,* second revised edition, SIAM, Philadelphia.

Song, C. 2009. *The Scaled Boundary Finite Element Method in Structural Dynamics*, *Int. J. Numer. Meth.,* Engrg., 77, 1139–1171.

Sprague, M. A. and Geers, T. L. 2007. *Legendre Spectral Finite Elements for Structural Dynamics Analysis*, Commun. Numer. Meth. Engrg., 24, 1953–1965.

Wolf, A. 2003. *The Scaled Boundary Finite Element Method*, Wiley, Chichester, U.K.

Akin, J.Ed 2009. *Finite Element Analysis Concepts via Solid Works*. World Scientific New Jersey Pg 12-25

Introduction to Finite Element Methods (ASEN 5007) Fall 2014 Department of Aerospace Engineering Sciences University of Colorado at Boulder

Abramowitz, M. and Stegun, L. A. (eds.), 1993. *Handbook of Mathematical Functions with Formulas, Graphs and Mathematical Tables*, *Applied Mathematics Series,* 55, Natl. Bur. Standards, U.S. Department of Commerce, Washington, D.C., 1964; reprinted by Wiley.

Hughes, T. J. R. 2000. *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*, Prentice Hall, Englewood Cliffs, NJ, 1987; Dover reprint.

Felippa, C. A. 2004.*A Compendium of FEM Integration Rules for Finite Element Work*, Engrg. *Comput.,* 21,867–890, 2004

Õnate, E., Rojek, J., Taylor, R. L., and Zienkiewicz, O. C., (2004).*Finite Calculus Formulation for Incompressible Solids Using Linear Triangles and Tetrahedra,* Int. J. Numer. Meth. Engrg., 59, 1473–1500.

Idelsohn, S. R., Del Pin, F., Rossi, R. and Õnate, E. 2009. *Fluid-Structure Interaction Problems with Strong Added-Mass Effect*, Int. J. Numer. Meth. Engrg., 10, 1261–1294.

Olovsson, L., Simonsson, K. and Unosson, M., 2005. *Selective Mass Scaling for Explicit Finite Element Analysis*, Int. J. Numer. Meth. Engrg., 631436–1445

****