ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 8 – No.2, September 2021

Review of Finite Element Modeling and Analysis Applications in Aerospace Structural Systems Design: Design Conceptualization Tool

Amir Javidinejad, Ph.D., P.E.
Structural Analysis Engineer (DAE)
The Boeing Company
4000 N Lakewood Blvd, Long Beach, CA 90808
amir.javidinejad@gmail.com

ABSTRACT

The paper presented hereby is an article with emphases on practical applications of the Finite Element Modeling (FEM) in aerospace structural systems design. This paper briefly describes the fields of analysis and its utilization via FEM. Briefly, general theory of the Finite Element Analysis (FEA) is discussed. General steps required for setting up FEA are elaborated. Several examples are introduced to illustrate the output from a typical FEA. In addition, it provides a comprehensive list of currently available software packages that perform pre/post and computational solver analysis in the international engineering markets. The information provided here could and should be a good starting point for technology assessment of the FEM for variety of aerospace design disciplines. It could be used as a reference in decision making process for aerospace design, testing and simulation.

Keywords: FEA, FEM, VIBRATION, IMPACT, LAMINATE ANALYSIS

1.0 Introduction:

Traditionally conceptual design of aerospace structures is followed by an exhaustive campaign of testing for prediction of failure in the design. Since there are a great deal of details involved in the design of every aerospace structural system, these testing efforts are time consuming and expensive. Furthermore, the qualification efforts for aerospace structural systems contain static, dynamics and thermal justifications. Each static test unit could be costly depending on the level of functionality required of the sample unit for qualification and certifications. Engineering hours required for support of testing and consequent analysis of data is tremendous as well. For these reasons an alternative method that is less time consuming and less expensive is desirable. Finite Element Analysis can be this exact solution for these short comings as they were in other engineering disciplines [1, 2, 3].

Finite Element Modeling and Analysis are discretization methods that model the structural and thermal systems in segments also known as finite elements [4]. Each individual element is placed in a global matrix, and by application of the global boundary conditions which includes the applied loads and the constraints, its response is approximated via numerical calculations from available raw material structural/mechanical data. By compilation of the responses of each of the individual elements, the overall system response is approximated.

Commercial FEA packages are available that provide the solver engines and pre and post processing tools for modeling of these discretization efforts. The cost associated with these FEA packages are mainly usage license fees that can be negligible as the number of units for required simulations increase. Unlimited number of product units can be modeled and analyzed via these packages.

In this paper several examples of the applications of the FEA for aerospace systems is discussed. The static, dynamic and thermal engineering approaches to the problems are mentioned. The required steps for FEA analysis, that is standardized by the engineering community and sometimes guided by the technical guidelines of the regulatory agencies, is elaborated to show that the FEA techniques are universal and thus acceptable in the international engineering community. It is worthy to note that the author mainly intends to review the applications of the FEA for aerospace structures rather than putting an emphasis in the engineering theoretical background of the FEA which is broadly available in many text books [6, 7, 8]. The general application of FEA for design is illustrated in Figure 1 following. A preliminary design is conceptualized. Simple hand estimations and calculations are done making sure the design is functional per design requirements. 2D and 3D Solid CAD models are generated. Finite Element Modeling is done and analysis is carried on the simulated design models. Optimized 2D and 3D models are generated and FEA iterations are performed again to achieve an optimal design that is the best feasible solution to the engineering problem.

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 8 – No.2, September 2021

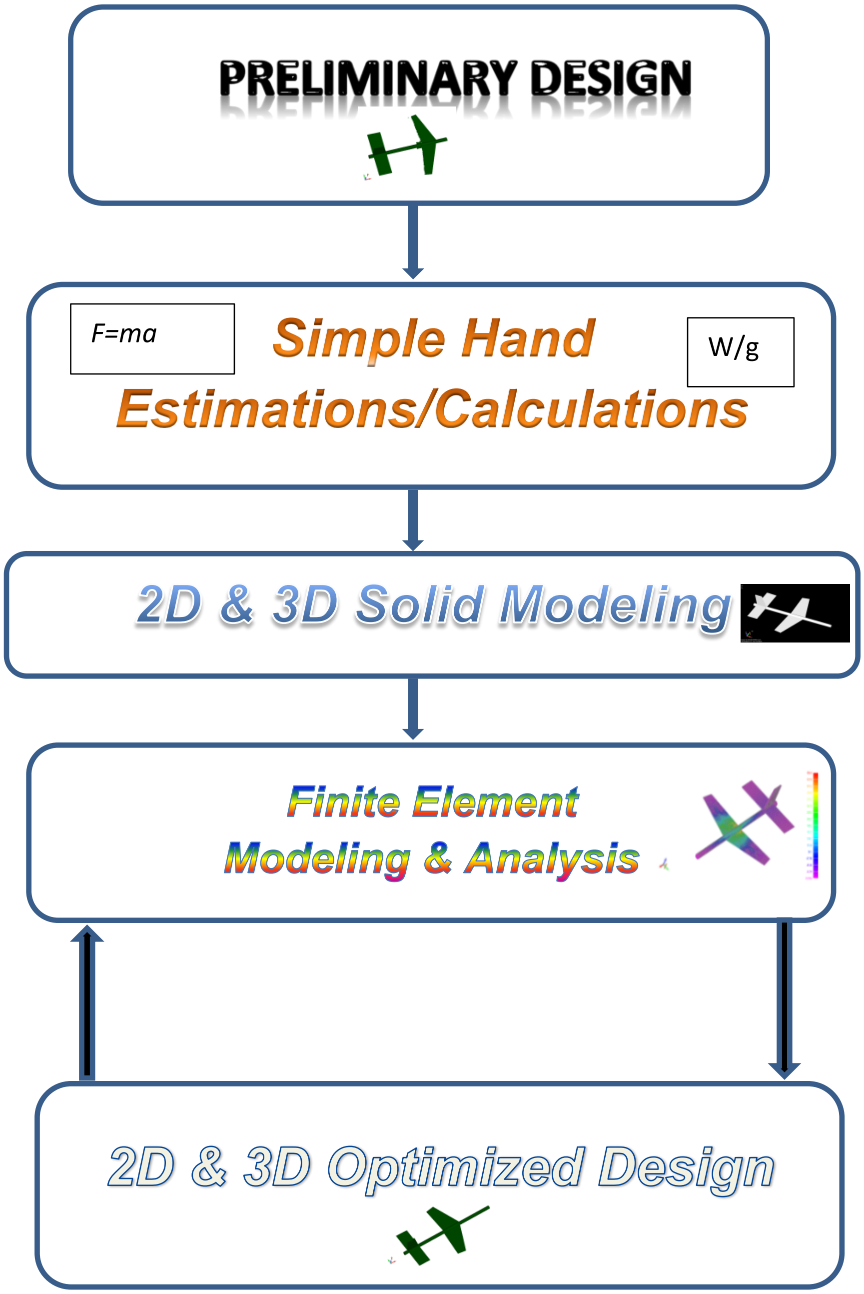

Figure 1. FEA Application for Design Iterations

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 8 – No.2, September 2021

1.1 Brief General Theory

The finite element method is one of the commonly used methods for calculation of the stresses and deflections of the large truss and beam systems as an example. The structural system is discretely divided into finite elements that each have their own basic equilibrium models. Each element model is defined and assembled into the larger global model that defines the system. Then the system is represented by simultaneous equations and solved by numerical methods. Hence, the nodal displacements are determined and the individual element forces are computed. The displacements can be translated into strains and the forces can be translated into stress values for each element [5].

Now let's consider a rod example under equilibrium conditions for illustrating the Finite Element Method. The summation of forces on the rod must equal to zero.

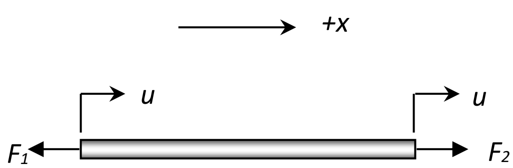

Figure 2. The rod, "truss" member under equilibrium conditions.

$$\sum F = 0, F_1 + F_2 = 0 \text{ or } F_1 = -F_2 \tag{1}$$

The same concept is applicable to the rod elongation. Thus, total rod elongation can be represented by the individual end displacements:

$$u = u_2 - u_1 \tag{2}$$

By expression,

$$F = ku \tag{3}$$

The force on the rod is,

$$F_2 = -F_1 = k(u_2 - u_1) \tag{4}$$

Or

$$F_1 = u_1 k - u_2 k \tag{5}$$

and

$$F_2 = u_2 k - u_1 k \tag{6}$$

The expressions (5) and (6) can be represented in the matrix form as,

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{7}$$

which follows of the form:

$$\{R\} = [K]\{D\} \tag{8}$$

where, $\{R\}$ is the load vector, $\{D\}$ is the displacement vector and $[K]$ is the stiffness matrix that is a function of the material properties of the rod.

$$[K] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \tag{9}$$

In finite element modeling, one models elements in this mentioned methodology and solves for loads and displacements of many or several meshes of elements all at once.

1.2 General Required Steps for FEA

FEA has been standardized by the international engineering community to follow a specific common approach for setup also known as preprocessing of the models. The setup is by definition similar for both analytical and computational methods of FEA. Whether the FEA is being carried out by hand calculations or via computer systems, it has to have the following numerated steps to be executed.

1) Material Property Definitions, which could be either:
   a) Isotropic
   b) Orthotropic
   c) Non-linear
2) Element Shape Properties which are either,
   a) Plate, Solid, Bar, Rod, Beam or
   b) Laminate Layup
3) Meshing the Geometry (Solid Model) by creating nodes and elements adhering to the solid model
4) Defining Applied Loads (Forces, Moments, Displacement, Temperature)
5) Defining Constraints (Degree of Freedom Boundary Conditions)
6) Analysis Runs
   a) Static (Linear or Non-Linear)
   b) Normal Modes (Modal)
   c) Heat Transfer
   d) Fluid Flow
   e) Buckling
   f) Optimization

All of the above mentioned steps are formulized by very user friendly software packages that could be carried out very quickly and efficiently by click of the mouse once a baseline CAD model is imported into the FEA package.

1.3 Software Packages

There are numerous FEM software packages available to the engineering community in the international market. More or less these packages follow the same pre-processing approach and are structured to easily convert from a solid model to a meshed model containing nodes and elements that can be used to solve the mathematical problem in hand. The main difference between them is the solver that is utilized within the package. There are many different solving techniques in reducing the property, stiffness and boundary condition matrices and/or equations that need to be solved in order to achieve a simulated displacement, load and heat transfer or dynamic mode results in a FEA package. Each and every one of these packages utilize their own unique computational code as solvers. The post processing of the results are also very similar in these FEA packages with slight differences in their methods of presentation. The results have to be the common engineering results that would be calculated using the theoretical mechanics approaches (i.e. normal and Von Mises stresses, strains, displacements, heat flux and mode shapes). The post processing presentation of these packages is normally in contour or fringe forms that give a visual presentation of the results field in color. Some examples would be presented in following sections of this paper. A brief list of some of the commercial FEM packages is shown following (Theses packages are copyright and trademark of their own respective development companies):

Abaqus, ADINA, ALGOR, ANSA, ANSYS, SAP2000, CSiBridge, ETABS, SAFE, PERFORM-3D, and CSiCOL, COMSOL Multiphysics, Femlab, CosmosWorks, Femap, FEMtools, Flexcom, FlexPDE, HyperSizer, JMAG, LS-DYNA, LUSAS, Nastran, NEi Software, OrcaFlex, StressCheck, VisualFEA, midasNFX, Patran and RFEM

2.0 Analytical Categories:

2.1 Linear Static Analysis

Aerospace system investigations fall under several major structures analysis categories which retrieve structural response of the system due to loads, displacements and or heat. The majority of the analytical work requires linear static analysis which simulates the material behavior of the systems in the linear range for both metallic and non-metallic materials. Stresses, deflections (displacements) and loads are retrieved from FEA models of the structural systems.

A typical example of such analysis work is modeling the airframe structure of an aircraft via FEM techniques. Figure 3 following illustrate the stress contours of an aircraft airframe under descent g-inertia loading. The aircraft is modeled using FEM by meshing the aircraft structure and applying the g-body inertia loading. The stress graduation is shown by a scale at the right side of the figure with the aircraft model exhibiting the various stress contour magnitudes over the airframe in color coding.

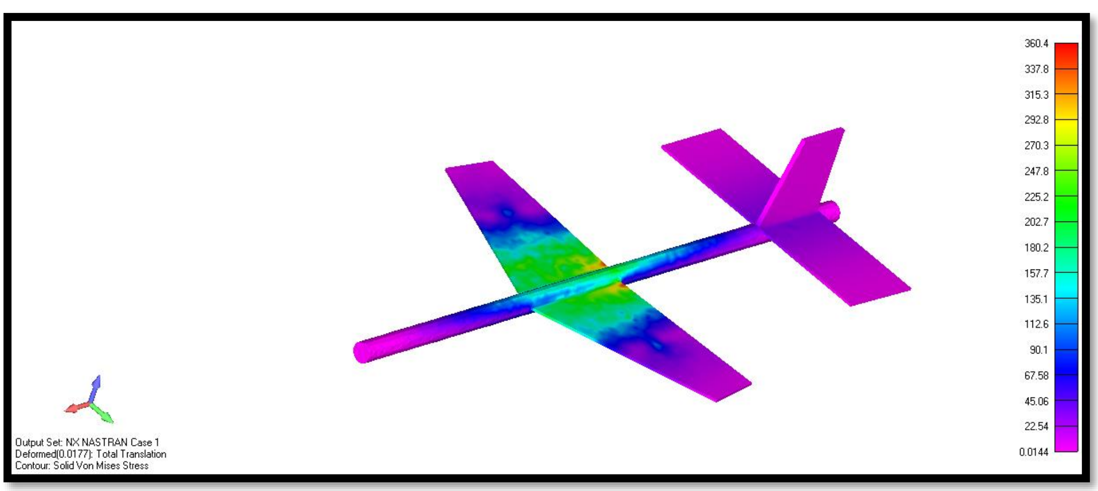

Figure 3. Stress Contours of the Aircraft Airframe Structure with Scaling

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 8 – No.2, September 2021

2.2 Non-Linear Static Analysis

To take advantages of non-linear material behavior of the materials where parts exhibit a smaller amount of stress for larger displacements the material non-linearity can be modeled in the FEA packages. Manually the strain-stress points representing the true material behavior are entered in the FEA package and the analysis are run. Also, large displacement calculations for sections can be produced from non-linear FEA analysis by simply activating the large displacement non-linear codes. Figure 4 following illustrates an FEA input module for material non-linearity inputting of the Elastic Modulus. Note the horizontal ("strains") axis unit is "in./in." and the vertical ("stress") axis unit is "psi".

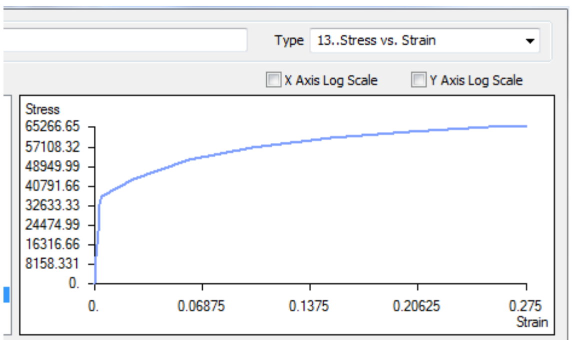

Figure 4. FEA Stress-Strain Input Module for Non-linear Representation of the Elastic Modulus

2.3 Vibrational Modal Analysis

FEM modeling can be utilized in determination of the natural resonance frequencies of any structure along with its mode shapes. Typically a structure is meshed and boundary conditions are defined by constraining the model at exact locations of the real life structure. Mass and stiffness matrices are generated internally via FEA and the natural frequencies of the structure are determined along with the corresponding mode shapes. Figure 5 following shows a FEM representation of an airplane galley structure with several compartments.

The natural frequencies are determined as the figures are showing the modes shape displacements of the galley at the corresponding resonance frequencies. The color coding is used in the FEM to show the scale of the displacement of various regions of the galley at specific resonance frequencies. Four mode shapes are illustrated in this figure. Natural frequencies are the frequencies at which the structural system oscillates or vibrates in the absence of any driving or damping force with further continuous force input the structure. The normal modes of the structure are the natural deflection motions/patterns of the structural system while oscillating or vibrating at its natural frequencies.

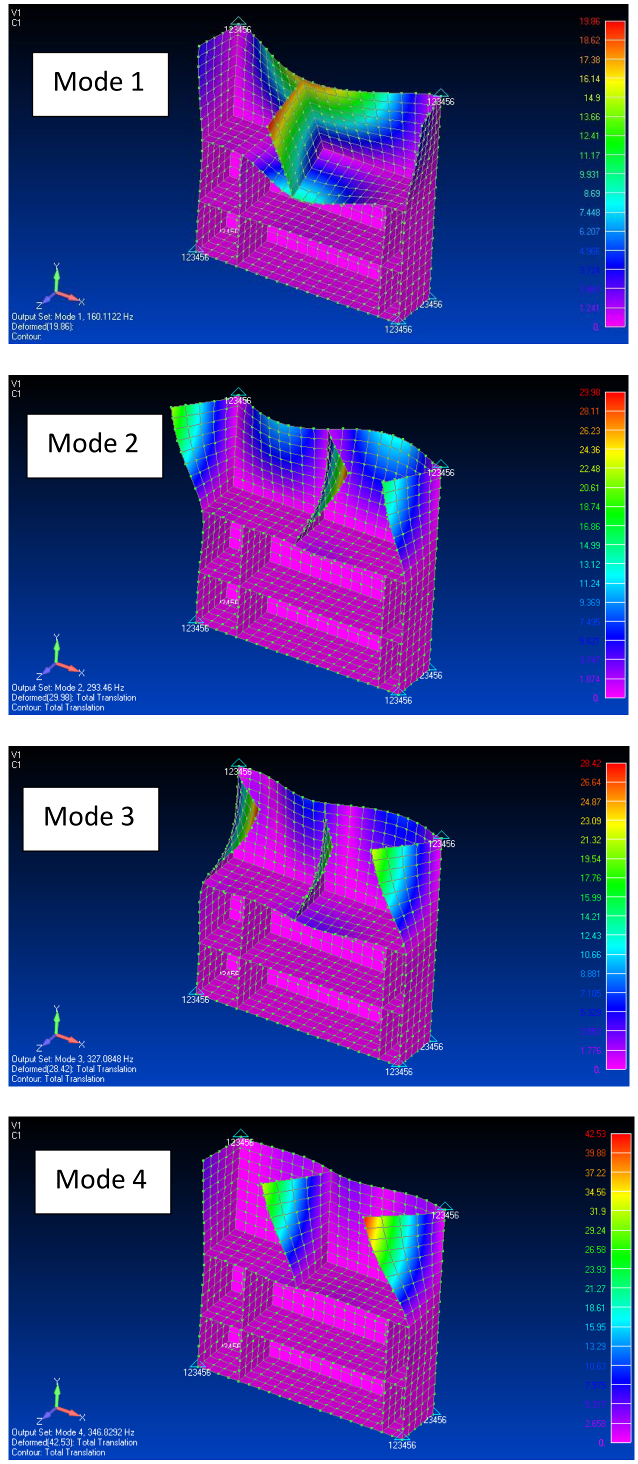

Figure 5. Mode Shapes and Natural Frequencies of the Airplane Galley

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 8 – No.2, September 2021

2.4 Buckling Analysis

FEM can also be used to determine a structure's critical buckling load. Similar to vibrational modal analysis where the resonance frequency is determined by use of similar algorithms the critical buckling load of a structure is determined by constraining the model to reality. Figure 6 following illustrates the use of FEA in determining the buckling load of an aircraft fuselage represented by its cross-section. The blue mesh in the background is the un-deformed cross-section of the fuselage and deformed color contoured mesh is the airframe section buckling. Both the buckling mode shape and the critical buckling loads are determined via FEA.

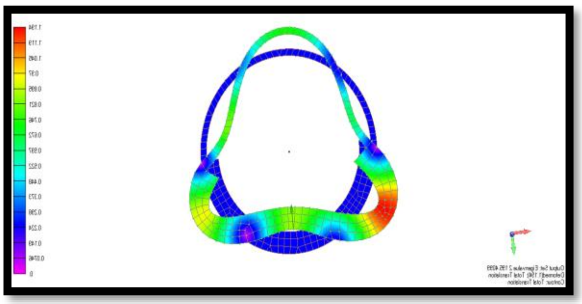

Figure 6. The Un-deformed and Deformed Buckled cross-section of a fuselage airframe.

2.5 Linear and Nonlinear Contact Surface Analysis

In conditions that either two or more elements are in contact and stresses and displacements are of interest for the design simulation or when one of the components in contact behaves totally different than other ones, one can use the Contact Surface Analysis. For instance where one would have attachment bushings in the lavatories or galleys that are fastened by bolts, one can take advantage of linear or non-linear contact analysis to simulate the load transfer and the actual interaction stresses. In general contact analysis is used when load transfer is essential but the components are only in contact and not secured together via a fastener system/mechanism.

The example above is mentioning attachment bushing as one example; other examples could be simply two aircraft monuments adjacent to each other and their respective wall panels are making contact which each other as the inertia g-loads are observed and due to displacements, the wall panels make contact with each other thus transferring the loads from one monument to another monument without even being connected together. Most FEA packages available provide such capabilities and the runs are systematic with definite procedural methods and are not time consuming to carry on.

2.6 Impact Analysis

FEA impact analysis comes in handy for impact analysis of instances where objects hit or impact the various airframe or engine components such as aircraft engine blades. This is typically done by introducing a time constant and displacing a mass of a variant shape to make contact with the section. This paper does not cover the details of the impact analysis since every FEA package has a different methodology for impact analysis and as such not all could be covered here.

2.7 Thermal Conduction Analysis

FEA can be used in determining the stress levels and displacement of aerospace structure where a temperature gradient is defined. Figure 7 following is showing a section of a rocket engine where the temperature of the core is at about 350 degrees F while the outside shell of the engine is at ambient temperature. The rocket engine section has an outer shell, insulation layer and the propellant inner core. As the engine ignites and propellant burns the internal temperature of the core reaches to 350 degrees F and that produces stress at different layers in the engine section. In Figure 7 the displacement due to conduction is shown and in Figure 8 the stress contours of the engine at this temperature gradient are shown.

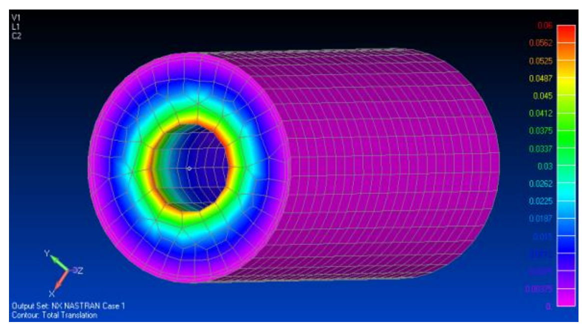

Figure 7. Total Displacement at the Different Rocket Engine Layers due to Propellant burn

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 8 – No.2, September 2021

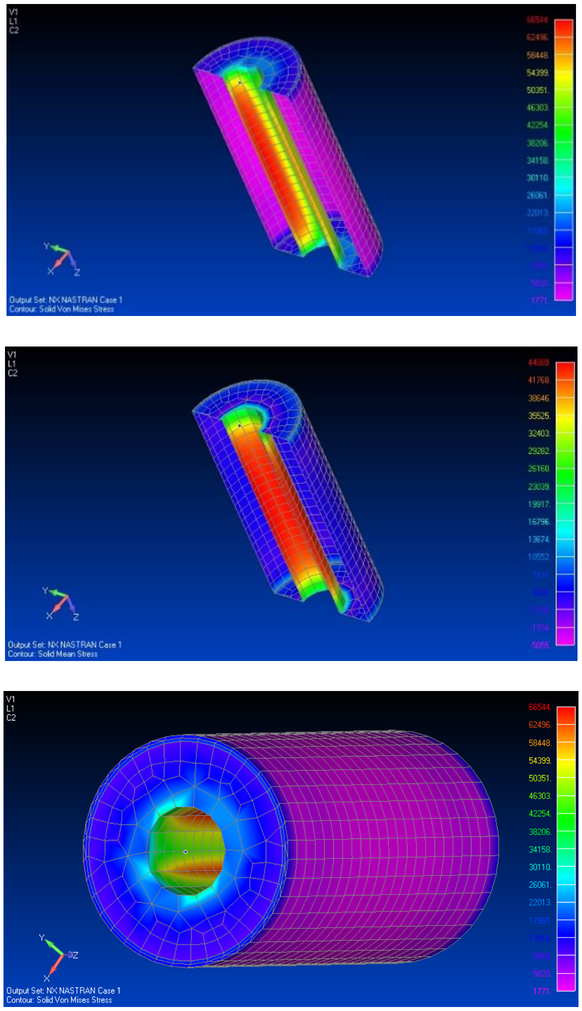

Figure 8. Various Stress Contours of the Rocket Engine (Also showing different cross-sections)

2.8 Design Optimization

Because the FEM packages are mainly parametric and part components such as airframe intercostal, ribs, webs and etc. can be modeled via variables; geometrical shape optimization can be achieved by Finite Element Methods. This is very useful in cutting down on the number of iterations required for an optimal design and cutting down on engineering hours. Strength and Stiffness Optimization can be done via FEA packages that may reduce overall weight of a structure as well.

2.9 Material Characterization

Composite Material Sizing (stiffness and strength) and modeling can be done via FEA packages where layup of an orthotropic material lamina can be entered and the FEA can use the layup in calculation of the properties of the model. The rest of the computations are carried out in the normal typical finite element method. Figure 9 following shows the input module of a FEA package for a typical composite ply layup.

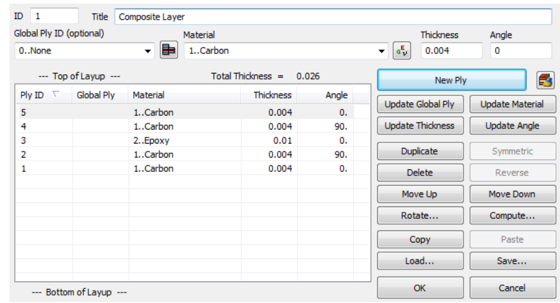

Figure 9. Typical Composite Layup Input for FEM

3.0 Conclusions:

In detail the application of the Finite Element Methods in the simulation, design and analysis for the aerospace mechanical systems was discussed. In brief the Finite Element Analysis theory was elaborated. The setup approach to a typical FEM modeling for simulation was shown and discussed. Several examples of the FEA applications for structural statics, structural dynamics and heat simulation were given. Some of the advanced material input capabilities of FEA packages were introduced. It is the perception of the author that, this paper provided an adequate reference for consideration of FEA for design simulation, test verification and response characterization of different aerospace mechanical systems. For further examples of the usages of the FEA in aerospace systems, one can explore the document references provided in this paper.

4.0 References:

[1] R. Courant (1943), Vibrational Methods for Solutions of Problems of Equilibrium and Vibrations, Bulletin of American Mathematical Society, Vol. 49.

[2] Reddy, J.N. (2005), *An Introduction to the Finite Element Method* (Third ed.). McGraw-Hill.

[3] Mahendran, Mahen (2007) Applications of finite element analysis in structural engineering, *Proceedings International Conference on Computer Aided Engineering*, pages pp. 38-46, Chennai, India.

[4] Robert D Cook (1995), Finite Element Modeling for Stress Analysis, John Wiley and Sons, Inc.

[5] Javidinejad, A. (2015), *Essentials of Mechanical Stress Analysis*, CRC Press.

[6] Strang, Gilbert; Fix, George (1973). *An Analysis of The Finite Element Method*. Prentice Hall.

[7] K. J. Bathe (1982), Finite Element Procedures in Engineering Analysis, Prentice-Hall.

[8] E.L. Wilson and A. Ibrahimbegovic, (1990), Use of Incompatible Displacements Modes for the Calculation of Element Stiffness or Stress, Finite Elements in Analysis and Design, Vol. 7.