INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.3 Issue 11,

November 2015

Pgs: 110-118

BUCKLING ANALYSIS AND MATERIAL SELECTION OF FUSELAGE FRAME USING CAD/CAM SOFTWARE

Arun M P[1]

[1]Assistant Professor, mparun3@gmail.com
Dept. of Mechanical Engineering
Eranad Knowledge city Technical Campus
Manjery, Malappuram, Kerala, India
+919946184306, mparun3@gmail.com

Abstract

The objective of this research is to study the strength of an aircraft fuselage when subjected to buckling due external load acting on it for different materials and composites. Literature survey designates some structural problems due to the external loads leads to this work. This involves design of an aircraft fuselage which has been modelled using CATIA and buckling analysis using ANSYS. This briefly explains the basics of the aircraft structures, composites, finite element method and its application using ANSYS. The analysis is carried out in ANSYS. Composite materials are used for the production of fuselage and that have very high range of load carrying capacity. From this work it is clear that there is a tremendous progress in the buckling load carrying capacity. To felicitate as a research, incorporated fuselage of an aircraft that would utilize the benefits of designing software, and structural analysis using ANSYS Software to designate the buckling analysis under external loads.

Keywords: Buckling analysis, fuselage and ansys.

1. Introduction

1.1 Composite materials

The word composite in the term composite materials signifies that two or more materials are combined on a macroscopic scale to form a useful third material. Different materials can be combined on a macroscopic scale such as in alloying of metals, but the resulting materials is, for all practical purpose, macroscopically homogeneous, i.e., the composite cannot be distinguished by the naked eye and essentially act together.

1.2 Classifications of composite materials

- Fibrous composite materials that consist of fibres in a matrix
- Laminated composite materials that consist of layers of various materials
- Particulars composite materials that are composed of particles in a matrix
- Combination of some or all of the first three types

1.3 Types of buckling analyses

Two techniques are available in the ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural, and ANSYS Professional programs for predicting the buckling load and buckling mode shape of a structure: *nonlinear* buckling analysis, and *eigenvalue* (or linear) buckling analysis. Because the two methods can yield dramatically different results, it is necessary to first understand the differences between them.

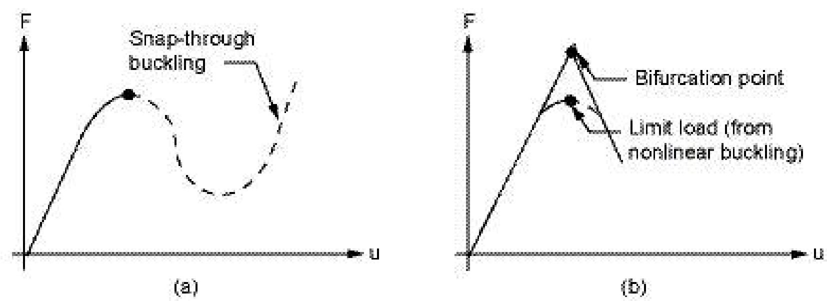

Figure1: BUCKLING CURVES **(**a) nonlinear load-deflection curve, (b) linear (Eigenvalue) buckling curve

1.3 Performing a Nonlinear Buckling Analysis

A nonlinear buckling analysis is a static analysis with large deflection active (NLGEOM,ON), extended to a point where the structure reaches its limit load or maximum load. Other nonlinearities such as plasticity may be included in the analysis. The procedure for a static analysis is described in "Structural Static Analysis", and nonlinearities are described in "Nonlinear Structural Analysis".

1.4 Performing a Post-Buckling Analysis

A post-buckling analysis is a continuation of a nonlinear buckling analysis. After a load reaches its buckling value, the load value may remain unchanged or it may decrease, while the deformation continues to increase. For some problems, after a certain amount of deformation, the structure may start to take more loading to keep deformation increasing, and a second buckling can occur. The cycle may even repeat several times.

1.5 Procedure for Eigenvalue Buckling Analysis

Again, remember that eigenvalue buckling analysis generally yields unconservative results, and should usually not be used for design of actual structures. If you decide that eigenvalue buckling analysis is appropriate for your application, follow this procedure:

1. Build the model.
2. Obtain the static solution.
3. Obtain the eigenvalue buckling solution.
4. Expand the solution.
5. Review the results.

2. Finite element analysis

The finite element method (FEM) (its practical application often known as finite element analysis (FEA)) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations.
In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input data and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages. The Finite Element Method is a good choice for solving partial differential equations over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness.

2.1 Model analysis

A technique used to determine a structure’s vibration characteristics:

- Natural frequencies
- Mode shapes
- Mode participation factors (how much a given mode participates in a given direction)

2.2 Mode Extraction Methods

Several mode extraction methods are available in ANSYS:

- Block Lanczos (default)
- Subspace
- Power Dynamics
- Reduced
- Unsymmetric

- Damped (full)
- QR Damped

2.3 Procedure for Modal Analysis

Four main steps in a modal analysis:

• Build the model
• Choose analysis type and options
• Apply boundary conditions and solve
• Review results

2.4 Modal stresses

• Available if element stress calculation is activated when choosing analysis options.
• Stress values have no real meaning; however these can be used to highlight hot spots.
• If mode shapes are normalized to unity, you can compare stresses at different points for a given mode shape.

2.4 Ansys

Ansys, Inc. is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software. While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multiphysics products.

ANSYS Mechanical and ANSYS Multiphysics software are non-exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a graphical user interface. These are general-purpose finite element modelling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

2.5 Analysis specification

The dispassionate of this research is to do buckling analysis of an aircraft fuselage and to simulate it for various boundary conditions. Static and modal analyses had been done and then buckling analysis had been carried out and their behaviour had been studied; the deflection, stress and mode shapes had been found.

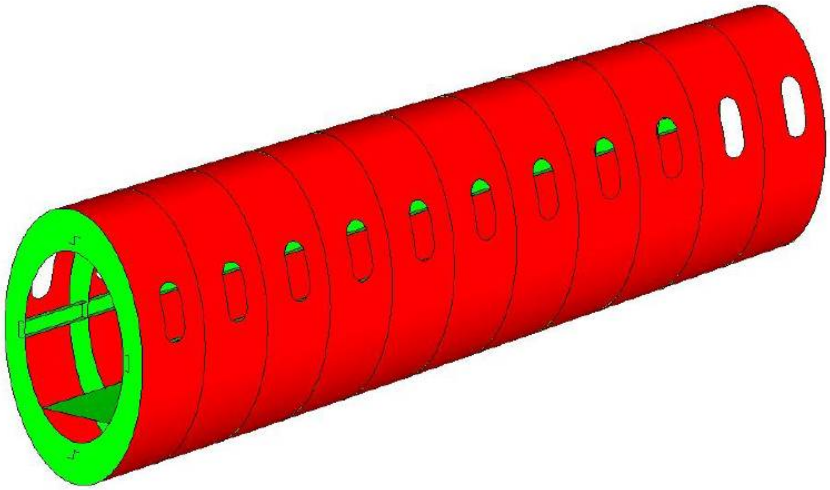

Figure2: CAD Model

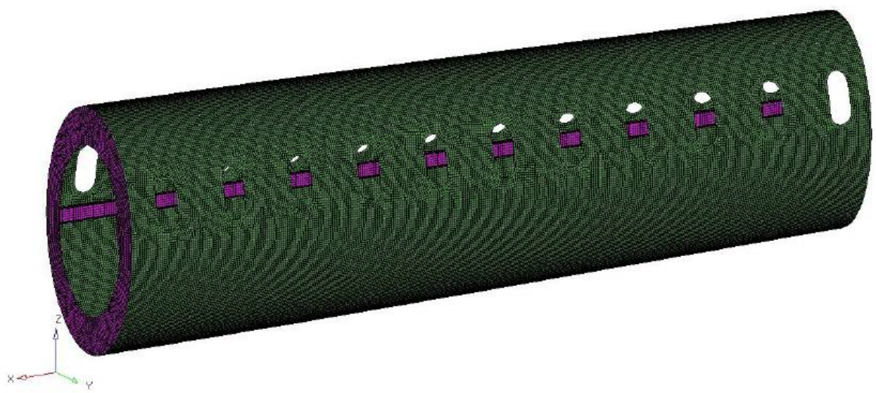

Figure3: MESH 1

Figure4: MESH 2

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.3 Issue 11, November 2015 Pgs: 110-118

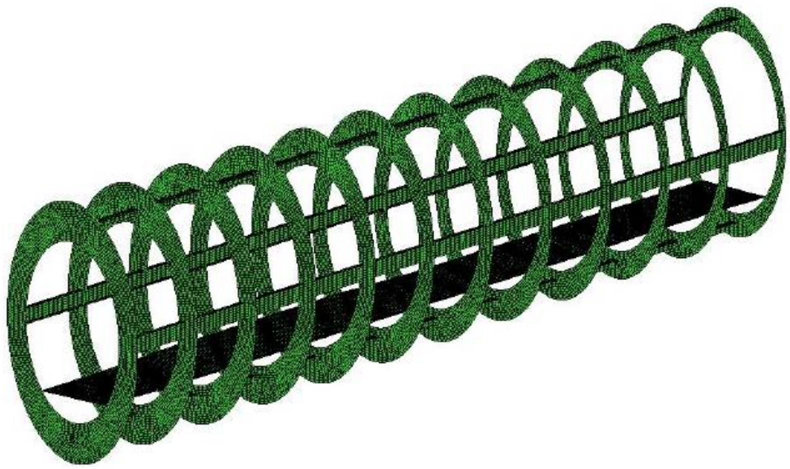

Figure5: MESH 3

2.6 Material properties

Inner components – Aluminium
Outer skin – Graphite Epoxy

Table 1: Material properties

Sl. No.	Material	Young's Modulus N/m^2	Density Kg/m^3	Poisson's Ratio
1	Aluminium	7.20E+10	2810.00	0.33
2	Graphite Epoxy	1.81E+11	1600.00	0.28

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.3 Issue 11, November 2015

Pgs: 110-118

3. Result analysis

On the comparison of various frequencies are evaluated below,

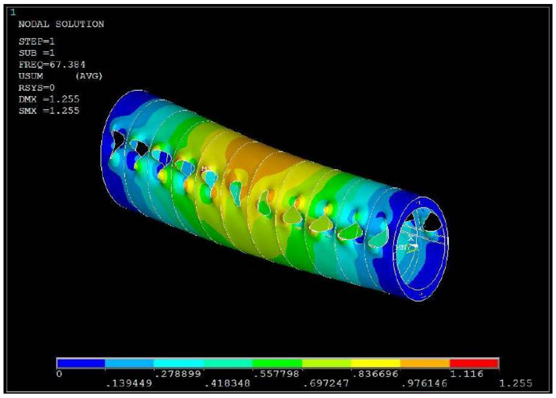

Figure6: Mode Shape 1

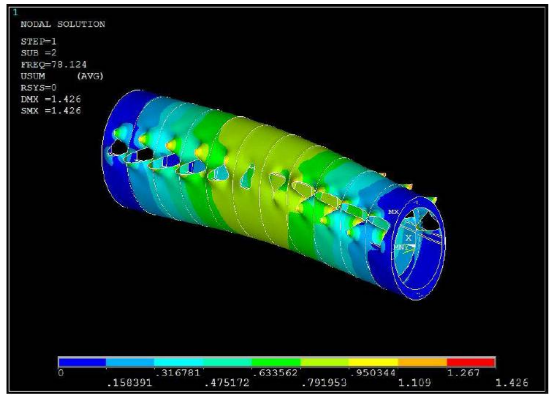

Figure7: Mode Shape 2

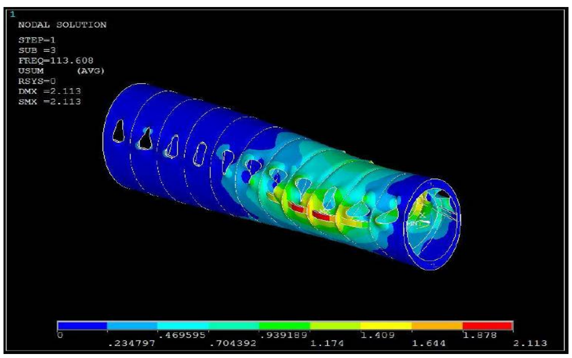

Figure8: Mode Shape 3

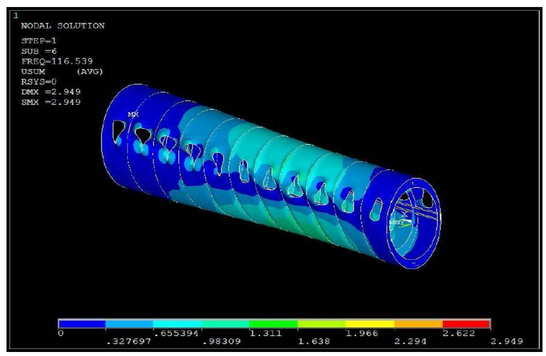

Figure9: Mode Shape 6

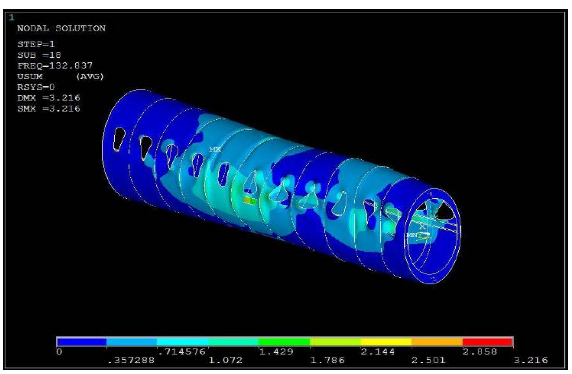

Figure10: Mode Shape 18

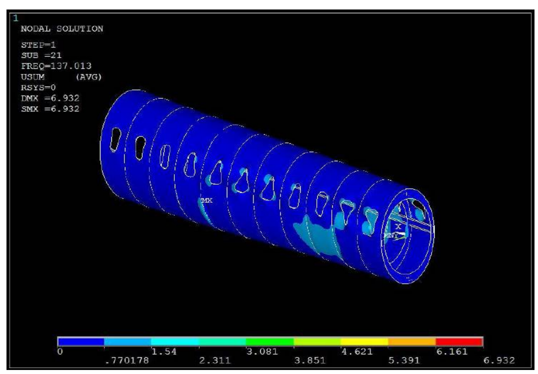

Figure11: Mode Shape 21

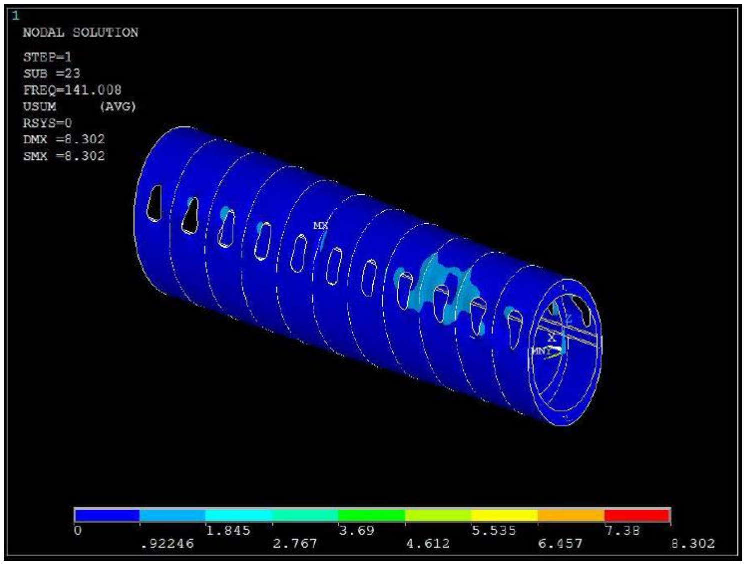

Figure12: Mode Shape 23

4. Conclusions

The modal analysis and buckling analysis of an aircraft fuselage have been done. The mode shapes, deflection and frequency values have been obtained and discussed. The fuselage has been analyzed using different material properties and all the results obtained are satisfactory. The fuselage structure can be redesigned in much better way using the obtained post-processing results.

References

[1] Black S., March 2002 "*A Grid-Stiffened Alternative to Cored Laminates*," High-Performance Composites.

[2] C. Bisagni and P. Cordisco, 2004 "Testing *of Stiffened Composite Cylindrical Shells in the Postbuckling Range until Failure*", AIAA Journal, 42 (9), 1806-1817.

[3] Gerdon G., Gurdal Z., November 1985 "*Optimal Design of Geodesically Stiffened Composite Cylindrical Shells*," AIAA Journal,; 23(11): 1753-1761.

[4] Graham J., March 1993 "*Preliminary Analysis Techniques for Ring and Stringer Stiffened Cylindrical Shells*," NASA report TM-108399,.

[5] Helms J.E., Li G., Smith B.H., 2001 "*Analysis of Grid Stiffened Cylinders*," ASME/ETCE.

[6] Hilburger M.W., 2001 "*Nonlinear and Buckling Behavior of Compression- loaded Composite Shells*," Proceedings of the 6th Annual Technical Conference of the American Society for Composite.

[7] Jaunky N., Knight N.F., Ambur D.R., June 1995 "*Formulation of An Improved Smeared Stiffener Theory of Buckling Analysis of GridStiffened Composite Panels*," NASA technical Memorandum 110162.

[8] Jaunky N., Knight N.F., Ambur D.R., March 1998 "*Optimal Design of General Stiffened Composite Circular Cylinders for Global Buckling With Strength Constraints*," Composite Structures.

[9] Jaunky N., Knight N.F., Ambur D.R., May-June 1998 "*Optimal Design of Grid-Stiffened Composite Panels Using Global and Local Buckling Analysis*," Journal of Aircraft, Vol. 35, No. 3.

[10] Joseph Clint, Santhosh Kumar, & Nazumuddin Shaik., 2014" *Buckling Analysis on Aircraft Fuselage Structure Skin*", IJEDR | Volume 2, Issue 4 | ISSN: 2321-9939

[11]Knight N.F., Stranes J.H., 1997 "*Development in Cylindrical Shell Stability Analysis*," NASA report.

[12] Phillips J.L., Gurdal Z., July 1990 "*Structural Analysis and Optimum Design of Geodesically Stiffened Composite Panels*," NASA Report CCMS-90-05.

[13]Poonam Harakarea and V.K. Heblikara, sept.2013 "*Evaluation of Static and Buckling load carrying capability of the Wing box through FEM approach*"

[14] Wang J.T.S., Hsu T.M., 1995 "*Discrete Analysis of Stiffened Composite Cylindrical Shells*," AIAA J, 23, 1753-1761.

[15]W. Rust, J. Overberg, 5–9 June 2006, "Accounting For Local Instabilities In The Coarse Model Of An Aircraft Fuselage By Means Of A Material Law", III European Conference on Computational Mechanics - Solids, Structures and Coupled Problems in Engineering.

A Brief Author Biography

Arun M P **-** Working as an Assistant Professor at Eranad Knowledge City Technical Campus, Manjeri, Malappuram, Kerala, India. Pursuing PhD at Noorul Islam Centre for Higher Education at Thuckley, Tamil Nadu. He has a Master of Engineering in CAD/CAM from Anna University, Chennai and Bachelor of Engineering in Aeronautical Engineering from Nehru Institute of Engineering and Technology at Coimbatore in Tamil Nadu.