

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Special Issue 3, March 2018

Review Modal Analysis of a Stepped Bar Using MATLAB & ANSYS

Dhiraj W. Ghatole¹

Assistance Professor, Department of Mechanical Engineering, Sanmati college of Engineering, Washim, India

ABSTRACT: Today we are living in the world of technology. I have different field of study. Engineering is one among such professions which provides students the caliber to deal with the problems in a different way.

Almost all the problems of various subjects which we deal in our mechanical engineering have solutions. Finite Element Methods is a subject where I deal with the elements but not the entire body at once. But still it involves a lot of calculations like Matrix multiplications, solving higher order equations etc. I have chosen a topic in Finite Element Methods i.e. Modal Analysis of finite elements and are working out to get results in a simpler way.

In this paper I use MATLAB software to simplify the tiresome calculations involved in Modal Analysis of Stepped bar. The code for solving the problem chosen was written in Mat lab and was tested for different inputs. Even though it is accurate and has a large number of built in functions, it is lagging applications in most of the mechanical fields. The code and the procedure I presenting here in our paper don't exist. The program written asks the user for inputs and gives output after making necessary calculations. Finally we are also using ANSYS 11.0 workbench software for verifying the results obtained in the matlab.

KEYWORDS: ANSYS, MATLAB, Modal analysis, mode, frequency.

I. INTRODUCTION

Finite Element Method is considered as one of the well-established and convenient technique for the computer solution of complicated problems in different fields of engineering: mechanical engineering, civil engineering, nuclear engineering, biomedical engineering, hydrodynamics, heat conduction, geo-mechanics, etc. From other side, FEM can be examined as a powerful tool for the approximate solution of differential equations describing different physical processes. The success of FEM is based largely on the basic finite element procedures used: the formulation of the problem in variation form, the finite element discretization of this formulation and the effective solution of the resulting finite element equations. These basic steps are the same whichever problem is considered and together with the use of the digital computer present a quite natural approach to engineering analysis. The objective of this project is to present briefly each of the above aspects of the finite element analysis and thus to provide a basis for the understanding of the complete solution process.

MATLAB (Matrix laboratory) integrates mathematical computing, visualization, and a powerful language to provide a flexible environment for technical computing. The open architecture makes it easy to use MATLAB and its companion products to explore data, create algorithms, and create custom tools that provide early insights and competitive advantages. MATLAB is an interactive software system for numerical computations and graphics. As the name suggests, MATLAB is especially designed for matrix computations: solving systems of linear equations, computing Eigen values and Eigenvectors, factoring matrices, and so forth. In addition, it has a variety of graphical capabilities, and can be extended through programs written in its own programming language.

ANSYS, Inc. is an engineering simulation software (computer-aided engineering, or CAE) developer that is headquartered south of Pittsburgh in Canonsburg, Pennsylvania, United States. ANSYS offers a comprehensive range of engineering simulation solution sets providing access to virtually any field of engineering simulation that a design process requires. The tools in ANSYS put a virtual product through a rigorous testing procedure such as crashing a car into a brick wall before it becomes a physical object. The ANSYS Workbench platform is the framework upon which the industry's broadest and deepest suite of advanced engineering simulation technology is built. With bi-directional

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

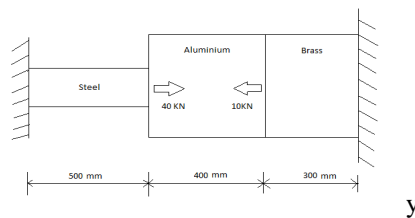
Vol. 7, Special Issue 3, March 2018

CAD connectivity, powerful highly-automated meshing, a project-level update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity, enabling simulation driven product development.

II. RELATED WORK

PROBLEM

Find the displacement and element stresses for the given



Composite Stepped Bar

Table : Properties of Elements

	Steel	Aluminium	Brass
Area (mm ²)	400	350	350
E(Gpa)	200	70	83

Solution-

Formation of element stiffness matrix

$$K_1 = (A_1 \times E_1) \div L_1 = \frac{200 \times 200000}{500} \times \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^3 \times \begin{bmatrix} 80 & -80 \\ -80 & 80 \end{bmatrix}$$

$$K_2 = (A_2 \times E_2) \div L_2 = \frac{350 \times 70000}{400} \times \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^3 \times \begin{bmatrix} 61.25 & -61.25 \\ -61.25 & 61.25 \end{bmatrix}$$

$$K_3 = (A_3 \times E_3) \div L_3 = \frac{350 \times 83000}{300} \times \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^3 \times \begin{bmatrix} 96.83 & -96.83 \\ -96.83 & 96.83 \end{bmatrix}$$

Global stiffness matrix.

$$K = [80 \ -80 \ 0 \ 0; -80 \ 141 \ -61.25 \ 0; 0 \ -61.25 \ 158.08 \ -96.83; 0 \ 0 \ -96.83 \ 96.83]$$

Formation of global force matrix-

$$[F] = \begin{bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{bmatrix} \text{ or } 10^3 \times \begin{bmatrix} 0 \\ 40 \\ -10 \\ 0 \end{bmatrix}$$

Formation of nodal displacement-

$$[U] = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}$$

Assembly of global stiffness nodal displacement-

$$[K] \times [U] = [F]$$

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Special Issue 3, March 2018

$$10^3 \times \begin{bmatrix} 80 & -80 & 0 & 0 \\ -80 & 141 & -61.25 & 0 \\ 0 & 0 & -61.25 & 158.08 \\ -96.83 & 0 & 0 & -96.83 \end{bmatrix} \times \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = 10^3 \times \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{Bmatrix}$$

Apply boundary conditions- $u_1=0$ and $u_4=0$

Erase first and fourth row and first and fourth column.

$$\begin{bmatrix} 141.25 & -61.25 \\ -61.25 & 158.08 \end{bmatrix} \times \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 40 \\ -10 \end{Bmatrix}$$

$$141.25 \times u_2 - 61.25 \times u_3 = 40 \quad \text{----- (1)}$$

$$-61.25 \times u_2 + 158.08 \times u_3 = -10 \quad \text{or}$$

$$61.25 \times u_2 - 158.08 \times u_3 = 10 \quad \text{----- (2)}$$

After solving we get

$$u_2 = 0.3074 \text{ mm}$$

$$u_3 = 0.055 \text{ mm}$$

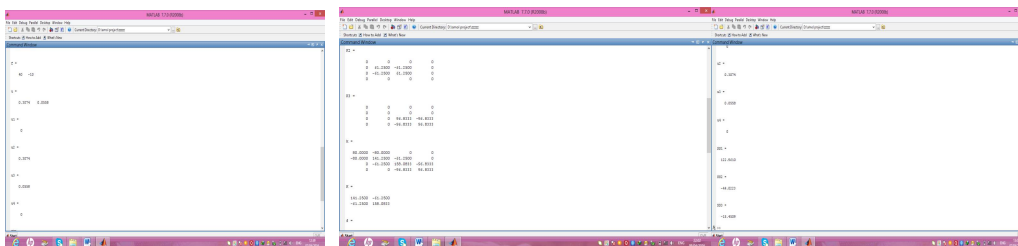
Stress calculation-

$$(1) \text{ Stress in element 1} = E_1 \div L_1 \times (u_2 - u_1) = \frac{200000}{500} \times (0.3074 - 0) = 122.96 \text{ N/mm}^2 \text{ (Tensile)}$$

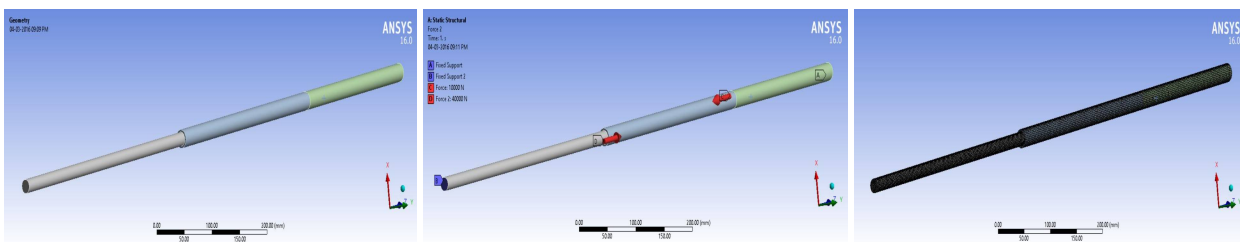
$$(2) \text{ Stress in element 2} = E_2 \div L_2 \times (u_3 - u_2) = \frac{70000}{400} \times (0.055 - 0.3074) = -44.17 \text{ N/mm}^2 \text{ (Compressive)}$$

$$(3) \text{ Stress in element 3} = E_3 \div L_3 \times (u_4 - u_3) = \frac{30000}{200} \times (0 - 0.055) = -15.21 \text{ N/mm}^2 \text{ (Compressive)}$$

MATLAB Output



ANSYS Output



(a)

(b)

(c)

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Special Issue 3, March 2018

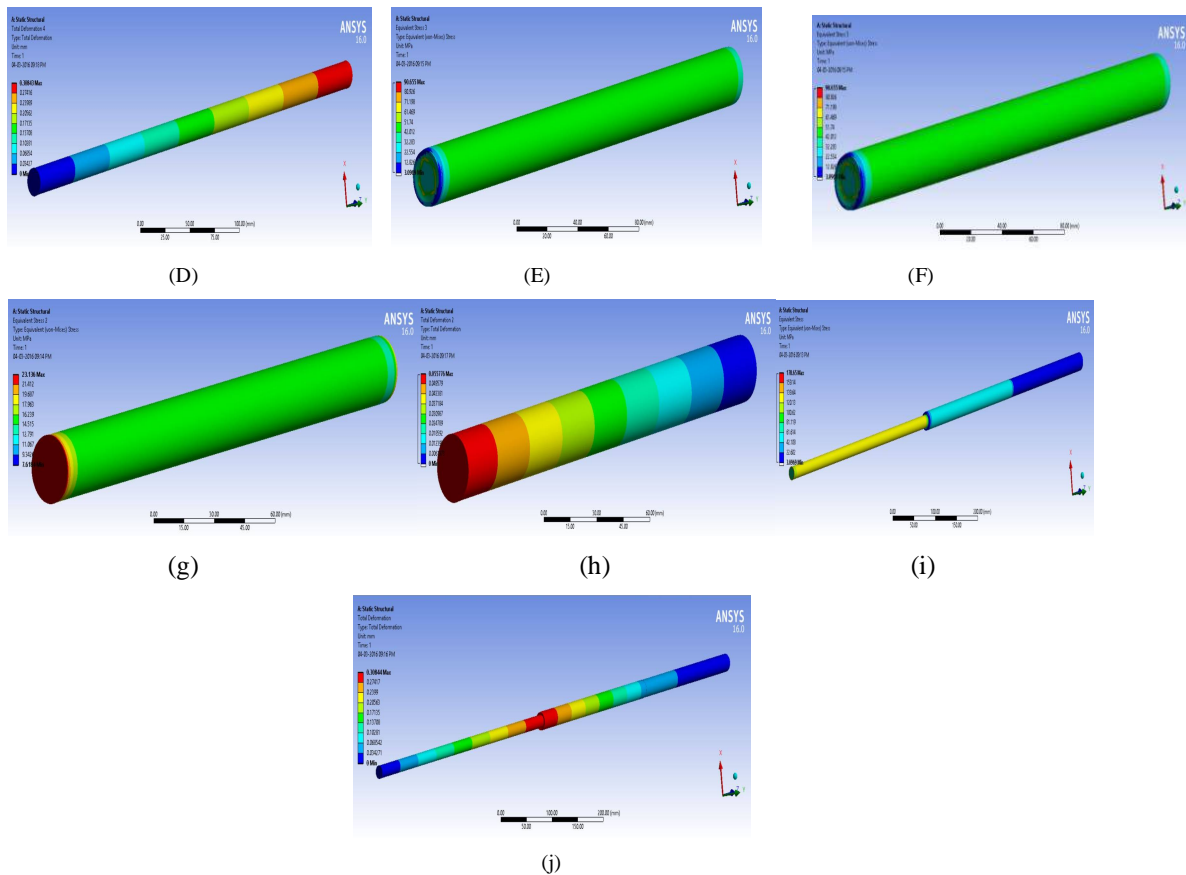


Fig. 2. Geometry of Composite Stepped Bar (a) Boundary Conditions of Stepped Bar. (b) Meshing of Composite Stepped Bar (C) Equivalent Stress in Steel Section. (D)Total Deformation in Steel Section (E) Equivalent Stress in Aluminium Section.(F)Total Deformation in Aluminium Section (g) Equivalent Stress in Brass Section.(h) Total Deformation in Brass Section(i) Total Equivalent Stresses in Composite Stepped Bar.(j)

Table : Comparative Results Composite Stepped Bar

Parameter	FEM	MATLAB	ANSYS
Displacement	$U_2=0.3074$ mm $U_3=0.055$ mm	$U_2=0.3074$ mm $U_3=0.055$ mm	$U_2=0.3084$ mm $U_3=0.055$ mm
Stress at element 1	122.96 N/mm ²	122.96 N/mm ²	178.62N/mm ²
Stress at element 2	-44.17 N/mm ²	-44.17 N/mm ²	90.65 N/mm ²
Stress at element 3	-15.24 N/mm ²	-15.24 N/mm ²	23.13 N/mm ²

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Special Issue 3, March 2018

III. CONCLUSION

This project describe that FEM, MATLAB and ANSYS are the efficient tool for analysing the mechanical components. We can solve the FEM problems for simpler objects very efficiently but as geometry gets complex it is difficult to find FEM solution manually. The MATLAB code is written for such complex geometry for different input and boundary conditions. The results obtained from MATLAB and solving the problem using FEM were nearly same but result obtained by ANSYS workbench are very close to the value obtained by FEM and MATLAB which is acceptable.

This process of solving the problems would result in saving a lot of time, and avoiding common errors that usually occur in manual calculations. However, writing MATLAB codes for solving higher order equations is difficult

REFERENCES

1. Gopichand Allaka, Manikanta Kati, B. Srikanth, Sk Rashed & D Neehaar, "Modal Analysis Of A Stepped Bar Using Matlab & Ansys", *International Journal of Engineering Research and Applications (IJERA)* ISSN: 2248-9622, Vol. 3, Issue 2, pp. 504-509. (March -April 2013),
2. Tirupathi R. Chandrupatla, "Introduction to Finite Elements in Engineering", 3rd edition, pp. 1-50.
3. MATLAB Guide to Finite Elements. An Interactive Approach, by Peter I. Kattan, pp. 11-44, 109-136.
4. Engineering programming in MATLAB: A Primer, by Mark A. Austin, Institute for System Research, University of Maryland, College Park, Maryland 20742, U.S.A. (February, 2000), pp. 41-64.
5. Gulab Pamnani, Dageshwar Singh Rajput, Nikhil Tiwari, Amit Gajendra, "Beam Analysis in Matlab (For simply supported & Cantilever; For Point Load & UDL)", *Advance Physics Letter*, ISSN:2349-1108, Vol. 1, Issue 2, pp. 27-36. (2014),
6. L. A. Dobrzanski, A. Pusz, A. J. Nowak, M. Gorniak, ("Application of FEM for solving various issues in material engineering", *JAMME, Journal of Achievements in Materials and Manufacturing Engineering*, Vol. 42, Issues 1-2, pp. 134-141. Sep.-Oct. 2010),
7. Rishi Raj*, Prabhat Kumar Sinha, Earnest Vinay Prakash "Modelling, Simulation and Analysis of Cantilever Beam of Different Material by Finite Element Method, ANSYS & MATLAB", *International Journal of Engineering Research and General Science* Volume 3, Issue 3, pp. 89-100(May-June, 2015),
8. Rishi Raj*, Prabhat Kumar Sinha, Earnest Vinay Prakash "Modelling, Simulation and Analysis of Cantilever Beam of Different Material by Finite Element Method, ANSYS & MATLAB", *International Journal of Engineering Research and General Science* Volume 3, Issue 3, pp. 89-100. (May-June, 2015),