IFERP
connecting engineers... developing research

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 8, December 2016

Static Analysis of Stepped Bar by Using Analytical Method, Finite Element Method and ANSYS Software

[1]Vikas S. Patil, [2]Vikram A. Patil, [3]Ravina R. Agarkar, [4]Moajij A. Shaikh, [5]Uttam Y. Siddha
[1][2][3][4] Student [5]Assistant Professor
[1][2][3][4][5]Department of Mechanical Engineering, Adarsh Institute of Technology & Research Centre Vita, India

Abstract:-- **The stepped bar is involved in many engineering application. These stepped bar systems suffer from the occurrence of deflection and stresses due to axial loading. These stresses and deflections have been examined to avoid possible resulting failure. This paper explains the application of finite element method for the analysis of a stepped bar subjected to an axial load. The element configurations that are studied range from one dimensional to three dimensional type and various mesh configurations. The Finite Element analysis results are compared with exact analytical solution and numerical solution of the stepped bar and this shows the elemental behavior of the stepped bar. The paper discusses the comparison of analytical exact solution, numerical solution and FEA results from ANSYS software.**

Index terms:-- **Finite Element Analysis (FEA), Finite Element Method(FEM), Numerical Method (NM)**

I. INTRODUCTION

FEM method is to solve the complicated problem and converted into whole region to the small region called as elements. The Finite Element Analysis (FEA) is a numerical method to solve the problems of engineering and mathematical physics. FEM is very useful to solve the complicated geometries where analytical solutions cannot be obtained. It is widely used in the industry for designs of buildings, airframes, electric motors, automobiles, machines etc.

In this paper we analyze the stepped bar analysis which is axial loading condition and then find out stresses, deformation and nodal displacements. Also analyze the stepped bar by using analytical method, finite element method and ANSYS software then compare the solutions of each node.

II. PROCEDURE OF ANALYTICAL SOLUTION FOR STEPPED BAR

A. *Define Problem*

The stepped bar is subjected to axial loading conditions with an force P. Analyze the axial loaded stepped bar shown in fig below. Predict the nodal displacement at U2 & U3.

A1 = 2400 mm2, A2 = 1200 mm2, A3 = 600 mm2.
L1 = 800 mm, L2 = 600 mm, L3 = 400 mm.
E1 = 83 × 103 MPa, E2 = 10 × 103 MPa, E3 = 200 × 103 Mpa

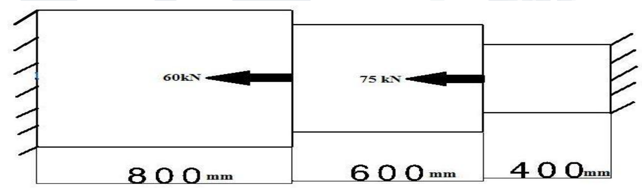

Fig. No. 1 – Stepped bar.

B. *Analytical Solution*

The force is applied on the stepped bar as shown in figure due to that the displacement or deformation of the nodes are occurs. These deformation can be calculated by using the following analytical formula,

$$dl = \frac{Pl}{AE}$$

Where,
P = Axial force in N.
l = Length of the step in mm.
A = Cross sectional area of bar in mm2.
E = Modulus of elasticity in N/mm2.

C. *Solution*

1. *Displacement of node 2*

$$dl_2 = \frac{Pl}{AE}$$

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 8, December 2016

$$dl_2 = -\frac{60 \times 10^3 \times 800}{2400 \times 0.83 \times 10^5}$$

$$dl_2 = -0.240mm$$

2. *Displacement of node 3*

$$dl_3 = \frac{Pl}{AE}$$

$$dl_3 = -\frac{75 \times 10^3 \times 400}{600 \times 2 \times 10^5}$$

$$dl_3 = -0.25mm$$

Node 1 & Node 4 are fixed so there is no any displacement i.e. dl1 = dl4 = 0.

III. PROCEDURE OF FINITE ELEMENT METHOD (FEM)

1. *Modelling*

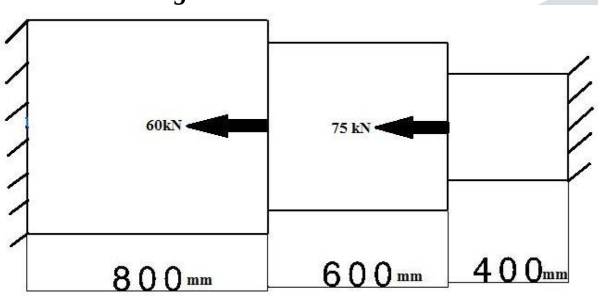

Fig. No. 2 – Stepped Bar.

or meshing of the problem is the transformation of infinite number of elements into the finite number of elements.

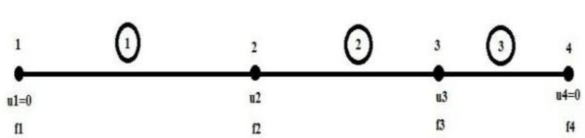

Fig. No. 3 – Meshing of Stepped Bar.

Total nodes = 4
Total elements = 3

1. *Connectivity Table*

Each node is connected to other elements the following table shows the node connectivity.

Table No. 1 - Connectivity Table.

Sr. No.	Element	Node
1.	1	1-2
2.	2	2-3
3.	3	3-4

2. *Stiffness matrix*

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

Where,
F = Force applied.
K = Stiffness.
U = Displacement.

4.1 Stiffness matrix for element 1

$$K_1 = \frac{A_1 E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_1 = \frac{2400 \times 0.83 \times 10^5}{800}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_1 = \begin{bmatrix} 2.49 & -2.49 \\ -2.49 & 2.49 \end{bmatrix} \times 10^5$$

4.2 Stiffness matrix for element 2

$$K_2 = \frac{A_2 E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_2 = \frac{1200 \times 0.1 \times 10^5}{600}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_2 = \begin{bmatrix} 0.2 & -0.2 \\ -0.2 & 0.2 \end{bmatrix} \times 10^5$$

4.2 Stiffness matrix for element 3

$$K_3 = \frac{A_3 E_3}{L_3}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

$$K_3 = \frac{600 \times 2 \times 10^5}{400}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

IFERP
connecting engineers... developing research

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Vol 1, Issue 8, December 2016

$$K_3 = \begin{bmatrix} 3 & -3 \\ -3 & 3 \end{bmatrix} \times 10^5$$

3. Assembly of stiffness matrix

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{bmatrix} 2.49 & -2.49 & 0 & 0 \\ -2.49 & 2.69 & -0.2 & 0 \\ 0 & -0.2 & 3.2 & -3 \\ 0 & 0 & -3 & -3 \end{bmatrix} \times 10^5 \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix}$$

4. Applying boundary conditions

$$\begin{Bmatrix} 0 \\ -60\times10^3 \\ -75\times10^3 \\ 0 \end{Bmatrix} = \begin{bmatrix} 2.49 & -2.49 & 0 & 0 \\ -2.49 & 2.69 & -0.2 & 0 \\ 0 & -0.2 & 3.2 & -3 \\ 0 & 0 & -3 & -3 \end{bmatrix} \times 10^5 \begin{Bmatrix} 0 \\ U_2 \\ U_3 \\ 0 \end{Bmatrix}$$

7. Solution

From above assembly matrix,

$$-2.49U_2 = 0$$

$$(2.69U_2 - 0.2U_3) \times 10^5 = -60 \times 10^3$$
$$(2.69U_2 - 0.2U_3) \times 100 = -60$$
$$269U_2 - 20U_3 = -60 ....(A)$$

$$(-0.2U_2 + 3.2U_3) \times 10^5 = -75 \times 10^3$$
$$(-0.2U_2 + 3.2U_3) \times 100 = -75$$
$$-20U_2 + 320U_3 = -75..........(B)$$

$$-3.2U_3 = 0$$

Equation A + 13.45 × Equation B we get,

$$269U_2 - 20U_3 = -60$$
$$+$$
$$-269U_2 + 4304U_3 = -1008.75$$

$$4284U_3 = -1068.75$$
$$U_3 = -0.2494mm$$

From equation 2 we get,

$$269U_2 = -64.98$$
$$U_2 = -0.2415mm$$

Since U_1 & U_4 are fixed there for $U_1 = U_4 = 0$.
analysis of stepped bar by using ansys software

Pre-Processing

Creating geometry, defining element type and constraints, defining material models, applying loads etc. those processes are carried out during preprocessing.

1. Modeling

The model is designed into the ANSYS software as per dimensions are given in the above problem.

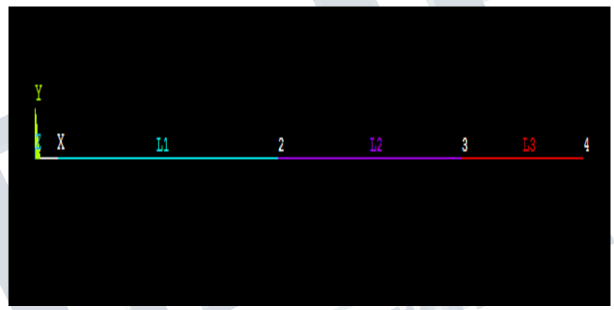

Fig. No. 4 – Modeling 1D of stepped bar.

2. Meshing / Discretization

Meshing is the conversion of the given infinite problem into the finite number of elements.

Here, Link element is used for the given 1D problem geometry and divided into 3 elements are shown in fig. below,

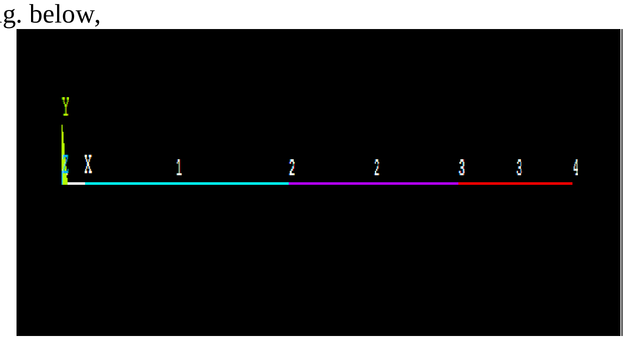

Fig. No. 5 – Meshing of Stepped Bar.

3.Constraints

The constraints are provided to the nodes 1 & 4 with all degree of freedom.

IFERP
connecting engineers... developing research
ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 8, December 2016

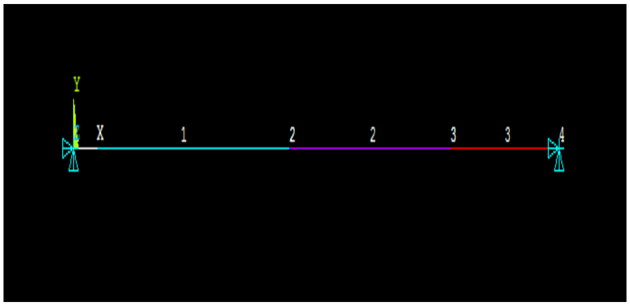

Fig. No. 6 – Constraints applied on stepped bar.

4.Loads & Displacement

As mentioned in given problem, The force is applied on nodes 2 & 3 having magnitude 60KN & 75KN respectively.

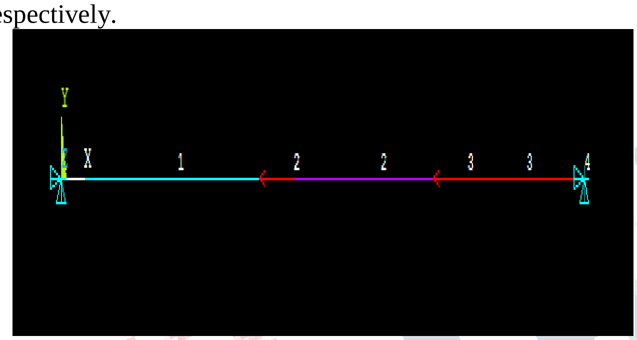

Fig. No. 7 – Loads & displacement applied on stepped bar.

Solution

The stiffness matrix is solved in the background of ANSYS and the solution is done. The results of deformation and nodal displacement are plotted in post processing.

Post Processing

Post processing means retrieving the results of an analysis. It is probably the most important step in the analysis, because it is useful to understand how the applied loads affect your design.

1.List Results

By applying compressive force the nodal displacement of node 2 & node 3 are listed below,

```
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL COORDINATE SYSTEM

  NODE      UX
     1   0.0000
     2 -0.24160
     3 -0.24947
     4   0.0000

MAXIMUM ABSOLUTE VALUES
NODE          3
VALUE  -0.24947
```

Fig. No. 8 – List results.

2.Plot Results

By applying force on the stepped bar the deformed size and shape of stepped bar are shown in below fig.

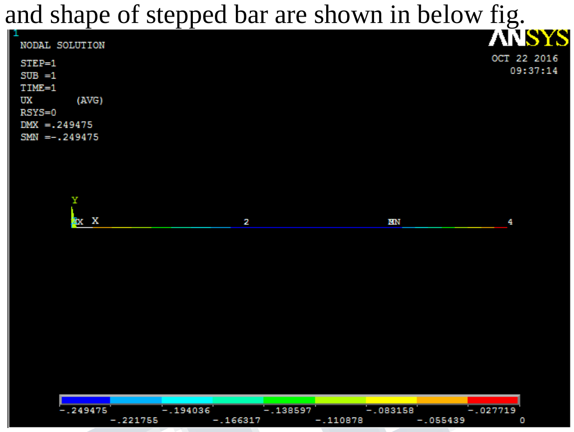

Fig. No. 9 – 1D Nodal displacement.

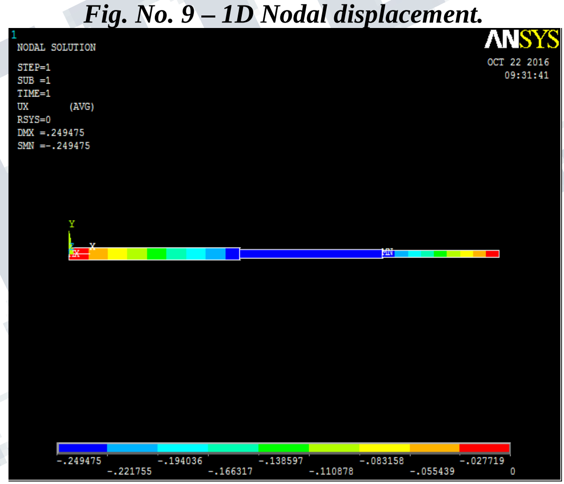

Fig. No. 10 – 2D Nodal displacement.

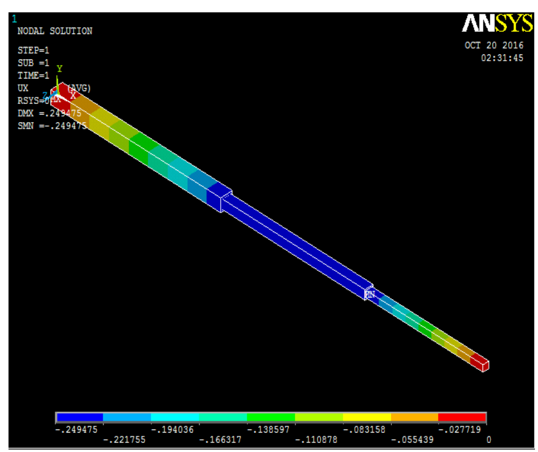

Fig. No. 11 – 3D Deformed size and shape.

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 8, December 2016

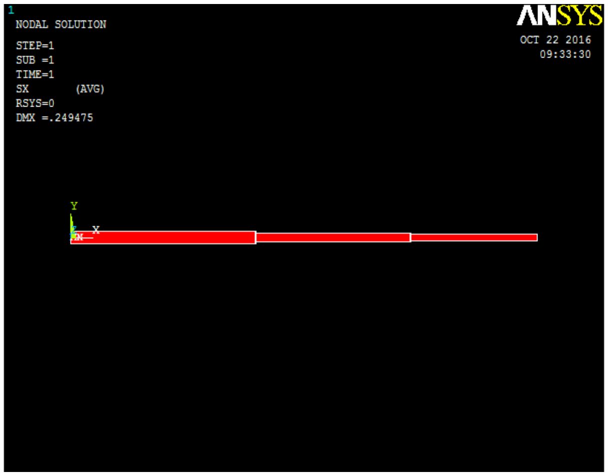

Fig. No. 12 – Equivalent stress.

IV. RESULTS

In this way we can solve the given problem by using analytical method, finite element method and ANSYS software. The results are obtained during analysis of stepped bar are listed in below table and compare with each other.

Table No. 1 – Comparison Table.

Node No.	Analytical Method (mm)	Finite Element Method (mm)	ANSYS Software (mm)
1.	0	0	0
2.	- 0.240	- 0.2415	- 0.24160
3.	- 0.25	- 0.2494	- 0.24947
4.	0	0	0

V. CONCLUSION

The results obtained from Analytical method, Finite Element Method (FEM) and ANSYS software were matched. The results achieved by ANSYS are very close to the exact values with an error around 1% which was accepted. This process of solving the problems would result in saving a lot of time, and avoiding common errors that usually occur in manual calculations.

REFERENCES

[1] Uttam Y. Siddha, Nawaj A. and Girish C. Mekalke, “Elemental Behaviour In Statics Analysis For A Simply Supported Beam by Using Fem Software”, International Journal of Science, Engineering and Technology Research (IJSETR), Volume 2, pp. 2187-2191, December 2013.

[2] Gopichand Allaka, Manikanta Kotti, B.Srikanth, Sk Rashed, And D Neehaar, “Modal Analysis Of A Stepped Bar Using Matlab & Ansys”, International Journal of Engineering Research and Applications (IJERA), Volume 3, pp. 504-509, March-April 2013.

[3] Siddha Uttam Y. & Kumbhar Samir B., “Natural Frequency Analysis of Automotive Seating System by using FEM Software”, International Journal on Mechanical Engineering and Robotics (IJMER), Volume 1, pp. 93-98, 2013.

[4] Introduction to Finite Elements in Engineering, 3rd Edition by Tirupathi R. Chandrupatla & Ashok D. Belegundu, Pearson Education.

[5] Gokhale, Nitin S, and Sanjay S Deshpande." Practical Finite Element Analysis.” FiniteTo Infinite, pune, 2008.