

STRESS – STRAIN FIELD ANALYSIS OF MILD STEEL COMPONENT USING FINITE ELEMENT METHOD UNDER UNIAXIAL LOADING

Vipul Gurumukhi¹, Tinkul kumar², Tushar chauhan³, Abhijeet Singh shishodia⁴

^{1,2,3,4}(Student, Dept. of Mechanical Engineering, NIET College, Gr. Noida, U.P., INDIA)

Abstract - Normally, when the force is applied on the component there will be some stress and strain generated in the component. So this stress, strain and displacement in the body have been analyzed with the help of MATLAB and software. Finite element method has been taken as the tool for mathematical modelling and analysis. As this method is a numerical method for solving problems of engineering and mathematical physics. To solve the problem, there is generation of mesh, it subdivides a large problem into smaller, simpler parts that are called finite elements. So, this method is helpful in calculating the comparatively exact values in the component. The main objective of this research is to investigate the mild steel component when different forces applied and what will be the results obtained. The investigation has been carried on MATLAB developed by MathWorks ltd. and FUSION 360 student version software.

Key Words: MATLAB, Autodesk FUSION360, tensile force, mild steel component, finite element method, stress-strain field.

1. INTRODUCTION

Finite element method is a numerical analysis technique and used to find variables. These field variables may be vector quantities like displacement, stress etc. The finite element method (FEM) is a powerful and versatile tool for practicing engineers that can be used to solve a wide variety of important engineering problems. Thus, it can be easily observed that FEM is a commonly used technique/method for stress concentration analysis. In the recent years, stress and strain concentration in materials has achieved the special attention of several researchers. **Donald W. Mueller, Jr[1]**: study about how finite element analysis is very helpful in computer languages like MATLAB. This paper tells about how anyone can use the knowledge of FEM for structural and design analysis. This paper emphasizes on important finite element concepts, such as the stiffness matrix, element and global coordinates, force equilibrium, and constraints. The paper gives the idea of using FEM with MATLAB. The ability of MATLAB to manipulate matrices and solve matrix equations makes the computer solution concise and easy to follow. **John R. Baker et al[2]**: had done computer-based engineering analysis that have become more powerful and user friendly in recent years. FEA is now considered as standard tool for engineers. So this paper outlines that how FEA can be used in various languages and software to get accurate results in lesser time. **C.J. Lissenden et al.[3]**: provided outlines that how much FEA is useful among

undergraduates. And also this paper gives the idea how FEM can be used in computer languages. As FEM can be applied in many problems like statics, dynamics, thermal, structural, nonstructural and many more. This paper is very much helpful in making FEM based projects. **Marshall F. Coyle[4]**: observed that Finite element analysis (FEA) is a powerful analytical tool used to evaluate structural, dynamic, thermal, fluid, and electrical and mechanical engineering problems. Engineering students can now solve complex problems that would not be feasible or practical to solve by hand. This paper also outline that by using different computer languages, it can easily solve complex problems. **Dr. Qin Ma et al.[5]**: briefly described about the deformation concepts. The paper outlines about how Finite Element Method is a Useful Modern Engineering Tool to Enhance Learning of Deformation Concepts. **Pikle Nilesh chandra et al.[6]**: gives the overview of solving mechanical 1D bar problem by Finite Element Method (FEM). This method include two steps, first we have to generate stiffness matrix and after generating stiffness matrix solve the system of linear equations by suitable method either by direct method or by iterative method.

Any analysis to be performed by using finite element method can be divided into following steps:

1.1 Finite Element Analysis

Solving FEM problem divided into following steps:

1. Divide problem into number of finite elements.
2. Generate local stiffness matrix.
3. Assemble to form the linear systems of equations of the form $Ku = f$.
4. Solve this linear system of equations by preconditioned conjugate gradient method.

Where, (K) is stiffness matrix, (u) is a load vector, (L) is total length of bar and (f) is a applied force in the referred figure. One end of bar is fixed and force (f) is applied at other end of bar. Bar is divided into (n) number of finite elements area of cross sections (A) at each element is given. According to the hook's law stiffness (k) is given by

$$K = \frac{AE}{L}$$

Where, E is modulus of elasticity.
Problem solving by FEM:

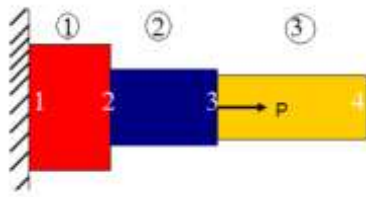
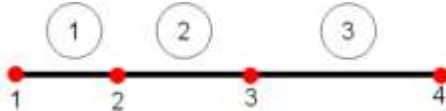


Figure: 1 1D problem of FEM

Above shown system can also be represents by line segment as shown below:



Here in 1D every node is allowed to move only in one direction, hence each node as one degree of freedom. In the present case the model as four nodes it means four degree of freedom. Let Q_1, Q_2, Q_3 and Q_4 be the nodal displacements at node 1 to node 4 respectively, similarly F_1, F_2, F_3, F_4 be the nodal force vector from node 1 to node 4 as shown. When these parameters are represented for a entire structure use capitals which is called global numbering and for representing individual elements use small letters that is called local numbering as shown.

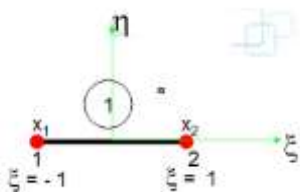


This local and global numbering correspondence is established using element connectivity element as shown:

Elements	Nodes		
e	1	2	Local
1	1	2	Global
2	2	3	
3	3	4	

Element Connectivity table

Now let's consider a single element in a natural coordinate system that varies as shown in figure given below, x_1 be the x coordinate of node 1 and x_2 be the x coordinate of node 2 as shown below.



Let us assume a polynomial

$$X = a_0 + a_1(z)$$

Now, $x = x_1$

$$X = x_2$$

After applying these conditions and solving for constants we have:

$$x_1 = a_0 - a_1$$

$$x_2 = a_0 + a_1$$

$$a_0 = (x_1 + x_2) / 2$$

$$a_1 = (x_2 - x_1) / 2$$

Substituting these constants in above equation we get:

$$X = a_0 + a_1 \xi$$

$$X = \frac{x_1 + x_2}{2} + \frac{x_2 - x_1}{2} \xi$$

$$X = \frac{1 - \xi}{2} x_1 + \frac{1 + \xi}{2} x_2$$

$$X = N_1 x_1 + N_2 x_2$$

$$N_1 = \frac{1 - \xi}{2} \quad N_2 = \frac{1 + \xi}{2}$$

Where, N_1 and N_2 are called shape functions also called as interpolation functions. These shape functions can also be derived using nodal displacements say q_1 and q_2 which are nodal displacements at node 1 and node 2 respectively, now assuming the displacement function and following the same procedure as that of nodal coordinate we get:

$$U = \alpha_0 + \alpha_1 \xi$$

$$U = \frac{1 - \xi}{2} q_1 + \frac{1 + \xi}{2} q_2$$

$$U = N_1 q_1 + N_2 q_2$$

$$= [N_1 \quad N_2] \begin{bmatrix} q_1 \\ q_2 \end{bmatrix}$$

$U = Nq$ Where N is the shape function matrix and (q) is displacement matrix. Once the displacement is known its derivative gives strain and corresponding stress can be determined as follows. Also with the help of potential energy approach we have the equation as:

Now total potential energy

$$\pi = \sum \pi_e = \frac{1}{2} Q^T \left(\int B^T E B A L \right) Q - \sum Q_i^T F_i$$

$$\Pi = \frac{1}{2} Q^T K Q - Q^T F$$

To extremise the potential energy

$$\frac{d\pi}{dQ^T} = 0 = KQ - F$$

The next step in FEM is finding out stiffness matrix from the above equation we have the value of K as:

$$K = \int B^T E B \, dv \quad \text{where } B = \frac{1}{L} [-1 \quad 1]$$

For an element

$$K = \int_e B^T E B A \, dx$$

But

$$\frac{dx}{d\xi} = L/2$$

Therefore, now substituting the limits as -1 to +1 because the value of zeta varies between -1 & 1 we have

$$K = \int_{-1}^{+1} B^T E B A \frac{L}{2} d\xi$$

Integration of above equations gives K which is given as

$$K = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Next step is assembly and the size of the assembly matrix is given by number of nodes X degrees of freedom, for the present example that has four nodes and one degree of freedom at each node hence size of the assembly matrix is 4 X 4. At first determine the stiffness matrix of each element say k₁, k₂ and k₃ as:

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

Similarly determine k₂ and k₃

$$K_2 = \begin{bmatrix} \frac{A_2 E_2}{L_2} & -\frac{A_2 E_2}{L_2} \\ -\frac{A_2 E_2}{L_2} & \frac{A_2 E_2}{L_2} \end{bmatrix} \quad K_3 = \begin{bmatrix} \frac{A_3 E_3}{L_3} & -\frac{A_3 E_3}{L_3} \\ -\frac{A_3 E_3}{L_3} & \frac{A_3 E_3}{L_3} \end{bmatrix}$$

Now, the global stiffness matrix for the bar element is:

$$\begin{bmatrix} 1 & 2 & 3 & 4 \\ \frac{A_1 E_1}{L_1} & -\frac{A_1 E_1}{L_1} & 0 & 0 \\ -\frac{A_1 E_1}{L_1} & \frac{A_1 E_1}{L_1} + \frac{A_2 E_2}{L_2} & -\frac{A_2 E_2}{L_2} & 0 \\ 0 & -\frac{A_2 E_2}{L_2} & \frac{A_2 E_2}{L_2} + \frac{A_3 E_3}{L_3} & -\frac{A_3 E_3}{L_3} \\ 0 & 0 & -\frac{A_3 E_3}{L_3} & \frac{A_3 E_3}{L_3} \end{bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix}$$

Next step is applying the boundary conditions for a given system. We have the equation of equilibrium KQ=F

K = global stiffness matrix

Q = displacement matrix

F = global force vector

Next step is solving the above matrix with generation of equations to determine the displacements which can be solved either by:

- Elimination method
- Penalty approach method

Last step is the presentation of results, finding the parameters like displacements, stresses and other required parameters.

The whole process of the Finite Element Analysis in any commercial software can be described in terms of three stages:

1. Pre Process:

In pre-processing we will firstly define what types of finite analysis will carry out on the system. After that we will define that what type of element which will be selected for find the behaviour of system. After defining the type of element we will define the material properties by given the value of E(Young Modulus of elasticity), ρ(density of the material), ν (poisson's ratio) for the material. After that we will model the entire system by making the node and build the element by assign the nodal connectivity

2. Process:

In this stage the process will be carried out by the computer to conduct the numerical analysis. Software package will solve the boundary value problem. Result of the analysis will be originated in this step.

3. Post Process:

In Post-Processing we will see the result of the field variable (displacement) in all over domain. The value of field variable may be found out in terms of nodal solution or can be find out in terms of the element solution. Stress and Strain can also be carried out in the post processing. The result can also be obtained as the contour plots of the displacement, stress or strain. All these results will available in post-processing stage. These results will help in the design of the mechanical component.

2. RESULT AND ANALYSIS

The component shown below in figure 2 is made of mild steel which has the young modulus of 220000Mpa. The arrow shown in figure tells that the tensile force has been applied on it. In this mechanical component we have taken three elements and four nodes (shown with blue dots) in figure 2 below, and now, what will be the displacement, strain and stress generated in the component has been derived with the help of MATLAB, FUSION360, and also done analytically.

The investigation derives the following conclusion:

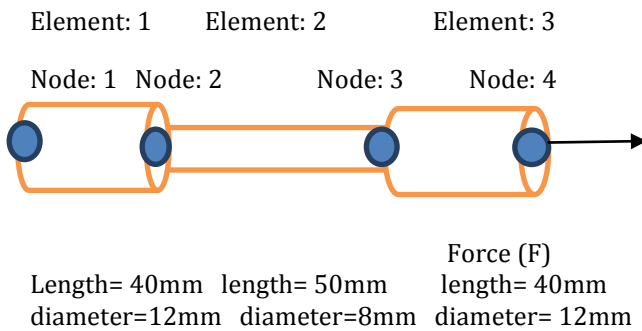


Figure-2 Mild Steel Specimen

2.1 Uniaxial tensile loading condition

2.1.1 Result obtained for 1000N:

When we take the force applied on the component is 1000N, then the generation of displacement, stress induced in body and the strain are shown with help of graph in figures 3, 4, 5 respectively. As we can see that the lines of analytically solved and the results given by MATLAB are approximately similar. So, we are getting approximately similar results through this. On the left side of figure 3 the values given are of MATLAB and on right side the values are of analytically solved values. Similar, in figure 4 and 5 that is of stress generation and strain the values are shown in figures respectively. Now, if we compare our results with Fusion software as shown by red lines it gives only maximum and minimum values of stress, strain and displacement.

The figure 6 represents the maximum and minimum values of stress and displacement. As it is shown in figure itself on the top left corner, there is one color strip which represents the minimum value and maximum values and according to that the color distribution is done in the component. The red part in the figure 6 represents that the maximum amount of stress and displacement of part is generated when we apply the desired load and blue color in this figure represents that the minimum amount of stress and displacement is generated.

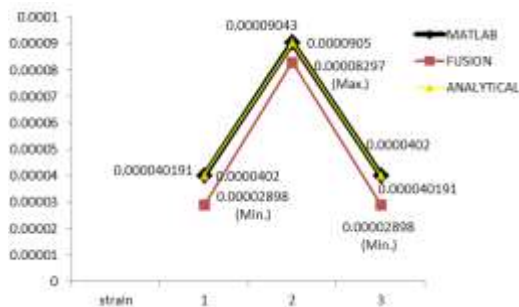


Figure-3 Strain field in mild steel component

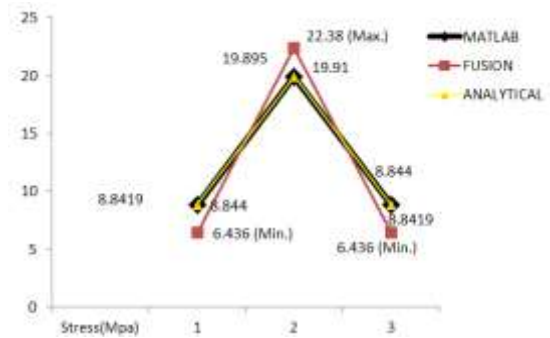


Figure: 4 Stress field in mild steel component

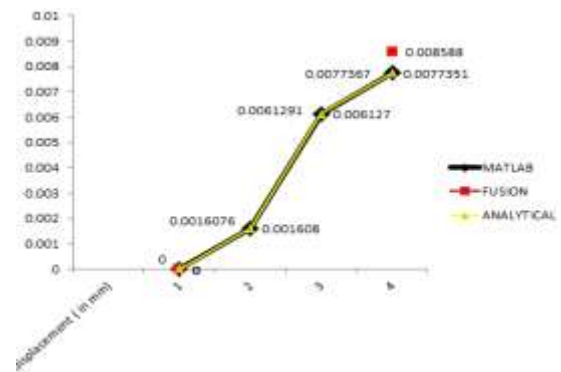


Figure: 5 Displacement (in mm)

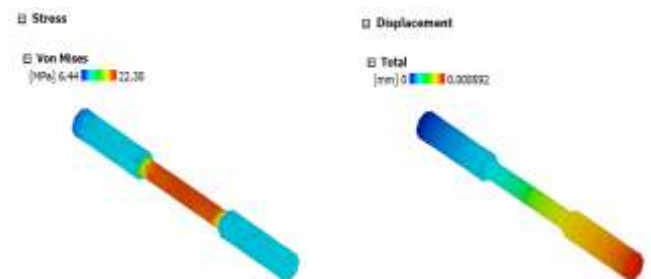


Figure: 6 Max. and Min. values of stress and displacement on FUSION software

2.1.2 Result for 3000N:

When the load applied is 3000N (tensile), the results are:

When the load applied on the component is 3000N then, the strain, stress generation and the displacement in the body is shown by the graph in figures 7, 8, 9 respectively. Since the lines of analytical solved (yellow line) and the MATLAB results (black line) are overlapping because the results come are approximately same. In figure 7 the values written on left side of black line are of MATLAB and on the right side are of analytically solved values. And at last, the FUSION360 values are shown by red lines in all the three figures are the minimum and maximum values generated in the component.

Here we can observe that only three elements and four nodes are taken for solving the given problem in MATLAB

and analytically. Therefore, the results are approximately similar. As we increase the no. of elements and no. of nodes in given specimen as shown in figure 2 , the coding done in MATLAB will become complex but on the other hand, we will get more accuracy in our results.

Here also in figure 10 the value generated by software has been shown. The red color shows maximum amount of stress is generated and blue color for lesser amount of stress generation. The color change in the component changes according to the color shown in top left corner strip. And on that strip, maximum and minimum values are also written on it.

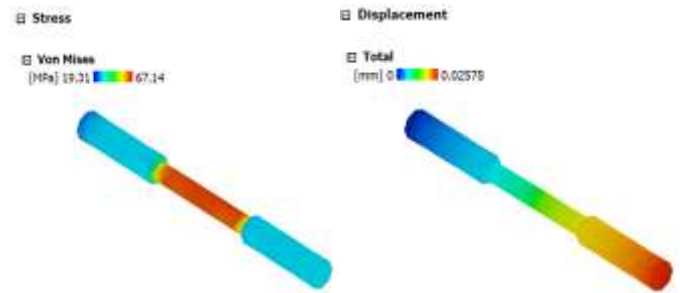


Fig: 10 Stress and displacement values on FUSION software

2.1.3 Result obtained from 9000N load:

Now, the third results are obtained when we apply the force of 9000N (tensile) on the component as shown in above figure 1. The results obtained on this are given below. As we can see that the same thing happens again and again. The lines of MATLAB and analytically solved are approximately similar in the figures 10, 11, 12. Similar here also the values of obtained from FUSION360 are maximum and minimum values of the component, as shown in figure 14.

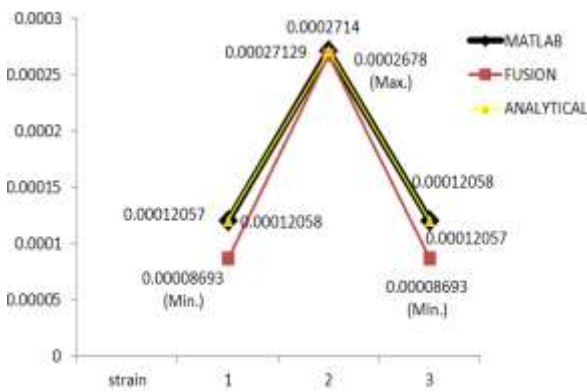


Figure: 7 Strain field in mild steel component

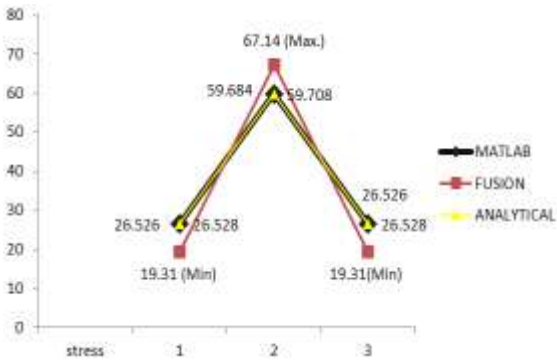


Fig:8 Stress (in Mpa) field in mild steel specimen

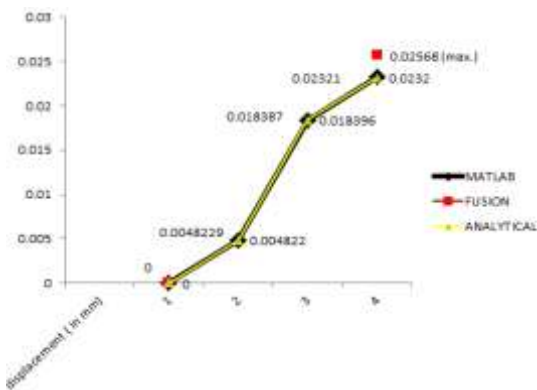


Fig: 9 Displacement (in mm) in mild steel specimen

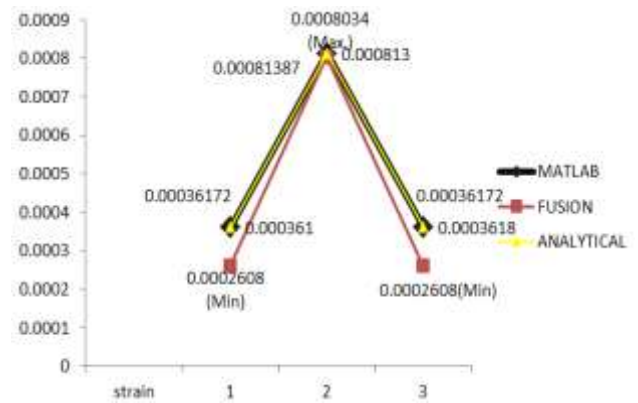


Figure: 11 Strain field in mild steel specimen

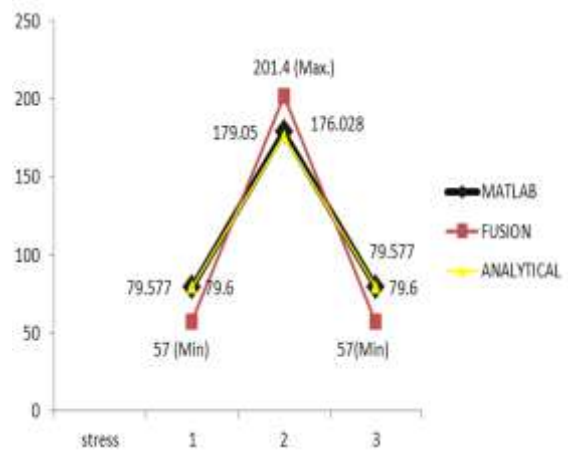


Figure: 12 Stress (in MPa) field in mild steel specimen

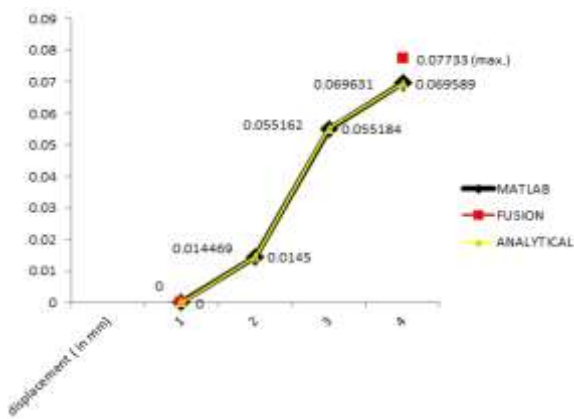


Figure: 13 Displacement (in mm)

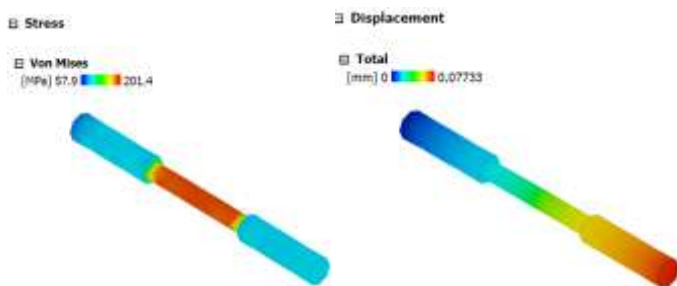


Figure: 14 MAX. and Min. values of stress and displacement on FUSION software

3. CONCLUSION AND FUTURE SCOPE

In this paper the behaviour of stress generation, strain in the body and displacement has been evaluated under the different uniaxial tensile load. This paper facilitate the choice of selection for adopting appropriate medium to analyse stress strain field. The result obtain with the aid of MATLAB is almost lie in the vicinity of analytic calculated results. As the complexity of physical model increases due to geometrical consideration, analysis become more time taking and tedious. Thus, comparison conducted that the number of nodes increase per element as adopted in software approach (fusion360). Accuracy become higher which provide the convergence plot on the basis of number of nodes per element.

4. REFERENCES

1. Donald W. Mueller, Jr, AN "Introduction To The Finite Element Method Using MATLAB", International Journal of Mechanical Engineering Education.
2. J. R. Backer, V. R. Capece and R. J. Lee, 'Integration Of Finite Element Software In Undergraduate Engineering Courses', ASEE Annual Conference Proceedings, Session 1520 (2001).

3. C. J. Lissenden, G. S. Wagle and N. J. Salamon, 'Applications Of Finite Element Analysis For Undergraduates', ASEE Annual Conference Proceedings, Session 3568 (2002).
4. M. F. Coyle and C. G. Keel, 'Teaching Finite Element Analysis To Second Year Students', ASEE Annual Conference Proceedings, Session 2793 (2001).
5. Dr. Qin Ma, Prof. Louie L Yaw "Finite Element Method As A Useful Modern Engineering Tool To Enhance Learning Of Deformation Concepts", 122nd ASEE annual conference and exposition paper id #13579, (2015).
6. Pikle Nileshchandra , Umesh B. Chavan, "Study Of 1d Bar Problem By Finite Element Method On Parallel Architectures", International Journal of Science and Research (IJSR), ISSN (Online): 2319-7064.
7. J. Rencis, *et al.*, 'Learning modules for finite element method on the world wide web', ASEE Annual Conference Proceedings, Session 2520 (1999).
8. D. L. Logan, A First Course in the Finite Element Method (PWS-Kent, Boston, 1986).
9. Y. W. Kwon and H. Y. Bang, The Finite Element Method Using MATLAB (CRC Press, Boca Raton, FL, 2000).
10. Przemieniecki, J.S., *Theory of Matrix Structural Analysis*, Dover Publications, Inc., New York, 1985.
11. Turner, M.J., R.W. Clough, H.C. Martin, and L.J. Topp: "Stiffness and Deflection Analysis of Complex Structures", *Journal of Aeronautical Sciences*, 23 : 805-823, 854 (1956).
12. Building Better Products with Finite Element Analysis, Vince Adams and Abraham Askenazi, OnWord Press, 1999. ISBN 1-56690-160X.
13. Finite Element Applications with Microcomputers, J. Frank Potts and J. Walter Oler, Prentice-Hall, Inc., 1989. ISBN 0-13-317439-5.
14. Miner, S. & Link, R., A Project-Based Introduction to the Finite Element Method. *Computers in Education Journal*, Vol. 10, No. 3, 2000.
15. http://wwwmath.cudenver.edu/~lfranca/links/fem_people.html, Leo Franca, accessed February 26, 2001