

# Design and Analysis of Truss Elements

Sachin Madakatti, Sunilakumar Biradar

BLDEA's V.P Dr.PGH College of Engg & Tech, Vijayapur

**Abstract**— It is very Often that analysis of stresses in complex structures is Very tedious job for mechanical engineers. So various methods have been introduced like analytical method, by using software's, And by FEM approach. Among all the methods software based Analysis is easy because it encourages any types of problems and the results obtained are error free. MSC NASTRAN PATRAN is one among that software that has helped from aeronautical to design engineers in their work. Due to its well-structured program and user friendly interface Analysis of structure is very easy. Our vision is to understand the software by solving some complex Structures using this software and validating it with the tradition Methods like analytical method or by fem approach.

## I. INTRODUCTION

In day today life we come across of various many structures like loads, bridges, towers, overhead tank etc. but as an engineers can we think are these structures safe, if safe what is FOS? Or what is stress distribution, stress concentration etc.

These are the basic things that we need to know as a mechanical engineer. If those structures are not safe what is the modification on substitution for those. Our Project deals with such an effort or a small step to understand such concepts. We choose some day today examples of structures that are in existence to our study. Before moving deep further let's know what the basic to understand our report are.

### 1.1 Concept of Bridge:

**Bridge :** Bridge is a structure that connects between 2 points or destination constructed to avoid natural barriers like rivers valley mountain etc., From many years railway bridges are in use are generally constructed using truss elements.

**Types of Trusses:** There are different types of trusses few among them are

1. Allan Truss



Fig. 1.2(a)

2. Bailey Truss



Fig. 1.2(b)

3. Bowstring arch truss



Fig. 1.2(c)

### Advantages:

1. Being less in weight and simple in the design the load carrying capacity is very high.
2. Based on the types of loads and load conditions material of the rod can be changed.
3. Transportation of the raw materials is very cheap and easy.

4. High strength towers are having high resistance to wind and snow loads.
5. Easy assembly and dismantling.
6. Helps in Identifying Suitable quality material
7. Versatility
8. Modification & Repair
9. Recycling of components used
10. High Durability

**Various Loads:**

1. Dead Load: Dead loads can be defined as the loads that acting on a body at rest or is not a function of time
2. Live Load: Live loads can be defined as the loads that are acting on a body in motion or that are depending On time function.
3. Snow Load: Snow loads are the loads that are caused due to the perception of snow on the structure's, The loads are varying due to uneven deposition of the snow.
4. Wind Load: Wind loads are the types of loads that are exerted on the structures due to wind and breeze forces. Structures are generally susceptible to these kinds of loads. They act at arbitrary angles and they are dynamic in Nature.
5. Thermal Stress: Thermal stress can be defined as the stresses that are produced in the member due to constrain while thermally it is being expanded.

**1.2 Introduction to MD Nastran- Patran Software & FEM Approach:**

**Nastran:** Nastran is FEA program that was developed for NASA in 1960's in USA. It was initially used for stress analysis in space vehicles. It name taken by NASA Structure Analysis. It was released for public in late 1971 that helped to study the elastic structure.

**Patran:** Patran is the world's most widely used pre/post processing software for FEA that provides solid modeling, meshing, analysis setup & post-processing for multiple solvers.

**1.3 FEM Steps:**

The solution of a general continuum problem by the finite element method always follows an orderly step by step process. The step-by-step

procedure with the reference the static structural problem can be stated as follows:

1. Discretisation of the continuum: The First step in the finite element method is to divide the given continuum into smaller regions of finite dimensions called as "Finite elements". The original Continuum or Structure is then considered as an assemblage of these elements connected at the finite number of the joints called as "Nodes" or "Nodal Points".
2. Selection of approximating functions: Approximating functions are also known as the displacement function or interpolation model. Displacement function is the starting point of the mathematical analysis. The displacement function may be approximated in the form a linear function or a higher-order function. A convenient way to express it is by polynomial expression.
3. Formation of Element stiffness matrix: After continuum is discrete with desired element shapes, the individual element stiffness matrix is formulated. Basically it is a minimization procedure whatever may be approach adopted.
4. Formation of Overall Stiffness matrix: After the element stiffness matrices in global coordinates are formed, they are assembled to form overall stiffness matrix. The overall stiffness is symmetric and banded.
5. Formation of element loading matrix: The loading forms an essential parameter in any structural engineering problem. The Loading inside an element is transferred at the nodal points and consistent element matrix is formed.
6. Formation of Overall Loading Matrix: The element loading matrices are assembled to form the overall loading matrix. This matrix has one column per loading case.
7. Formation of Overall Equilibrium equation: Overall equilibrium equation is the systematic arrangement of overall stiffness matrix, overall load vector and overall displacement vector to get set of simultaneous equations. Overall equilibrium equation can be expressed as shown Below
8. Incorporation of boundary conditions: The boundary restraint conditions are to be

imposed in the stiffness matrix to avoid the condition of singularity. The solution cannot be obtained unless support conditions are included in the stiffness matrix.

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

Where,  $[K]$  is a overall or global stiffness matrix (square matrix)

$\{Q\}$  is a overall or global displacement vector (Column matrix)

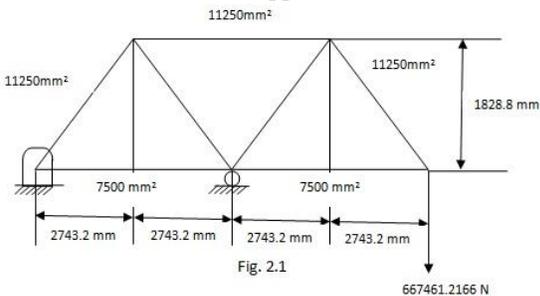
$\{F\}$  is a overall or global force vector (Column matrix)

9. Calculation of unknown nodal displacements: After incorporation of boundary conditions elimination method or penalty methods of handling boundary condition are used to calculate unknown nodal displacements from equilibrium equation or simultaneous equations.

10. Calculation of strain and stresses: Nodal displacements are utilized for calculation of strain and stresses using suitable equations.

II. ANALYSIS OF BRIDGE STRUCTURE

Find the deflection at the nodes, stresses in members & the reactions at the supports for the truss shown in Fig.3.1 when the 667461.2166 N loads are applied.



The figure 2.1 given above can be transformed in to FEA Model figure 2.2 can be solved using traditional FEM Method and then by software simulation Areas of the elements are as follows For the elements 1 & 4  $A=7500\text{mm}^2$  For the elements 2,3,5,6,7  $A= 11250\text{mm}^2$  From the given structure it can be seen that it is an 7 element structure consisting of 5 Nodes. So we can see that there is fixture at node1 and roller support at node 2. It means that in the node 1 all degree of freedom are restricted & At Node 2,

Only X direction displacement is allowed

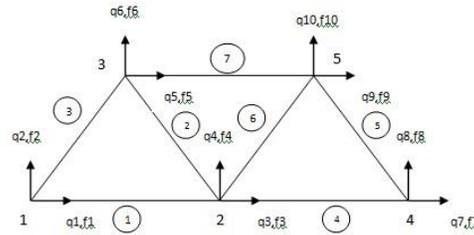


Table No.2 .A Nodal co-ordinate Data

Node	X	Y
1	0	0
2	5486.4	0
3	2743.2	1828.8
4	10972.8	0
5	8229.6	1828.8

Table No. 2. B Element Connectivity Data

Element	Initial node	Final node	Length of Element	Directions cosines	
				L	M
1	1	2	5486.4	1	0
2	2	3	3296.91	-0.8320	0.5547
3	1	3	3296.91	0.8320	0.5547
4	2	4	5486.4	1	0
5	5	4	3296.91	0.8320	-0.5547
6	2	5	3296.91	0.8320	0.5547
7	3	5	5486.4	1	0

The Element Stiffness Matrix for the Truss element is given by

$$[K_e] = \frac{AE}{L} \begin{bmatrix} I^2 & Lm & -I^2 & -lm \\ Lm & m^2 & -lm & -m^2 \\ -I^2 & -lm & I^2 & Lm \\ -lm & -m^2 & Lm & m^2 \end{bmatrix}$$

Global stiffness matrix

$$\{K_e\} = \{ke1\} + \{ke2\} + \{ke3\} + \{ke4\} + \{ke5\} + \{ke6\} + \{ke7\}$$

Applying the Equilibrium Condition

$$[K][Q] = [F]$$

Solving the matrix,

The Results obtained using FEM Approach are tabulated as shown below

Nodal (mm)	Displacement	Stress (N/mm <sup>2</sup> )	
At Node 1	0	At Node 1	42
At Node 2	0.0052	At Node 2	83
At Node 3	0.0052	At Node 3	148
At Node 4	39	At Node 4	120
At Node 5	1.56	At Node 5	140

### III. SOLUTION USING NASTRAN & PATRAN SOFTWARE

**Model:** It is the model taken for analysis. It has 7 elements and 5 nodes and at each node there is 2 degree of freedom

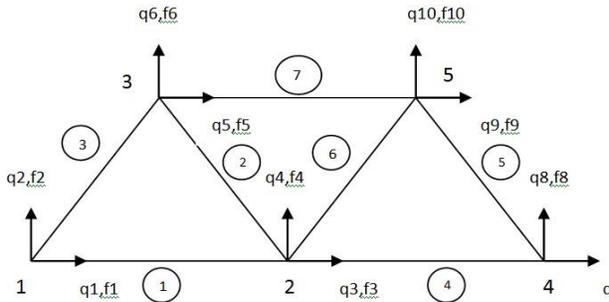


Fig 3(a) Model

Where, q represents the nodal displacement  
f represents the nodal force

**Material Data :** Here we given the material data's like Young's Modulus as 206842.7 N/mm<sup>2</sup> and Poisson's ratio as 0.3

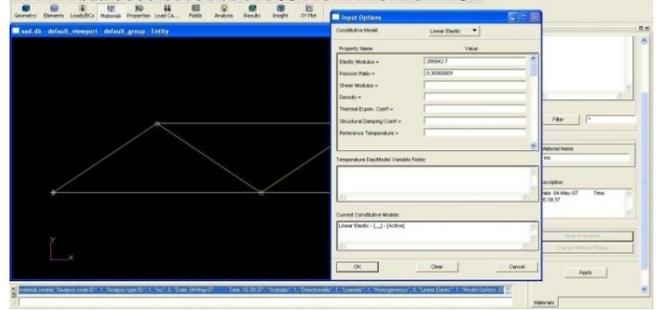


Fig 3 (b) Material Data

**Properties:** Here we given the properties like material of element as MS and the area for the elements 2,3,5,6 as 11250 mm<sup>2</sup> , for the elements 1,4,7 as 2743.2 mm<sup>2</sup>

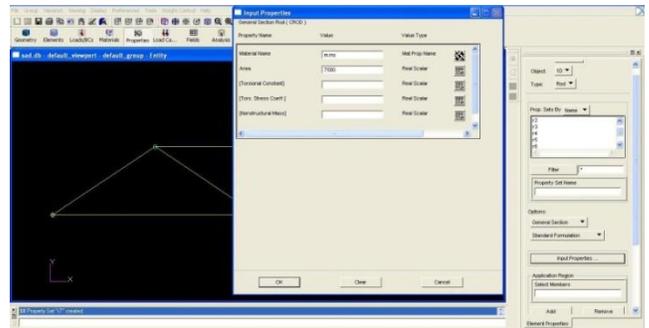


Fig 3 (c) Properties

**Loading conditions:** In this step we given the boundary conditions. The node 1 is fixed and at node 2 the displacement is provided along X axis, point load is applied at node 4

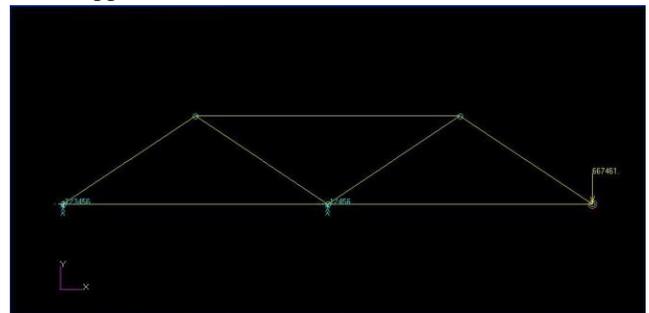


Fig 3 (d) Loading Conditions

**Displacement Result:** Finally we get the Maximum result as 39 mm and Minimum result as 0 mm

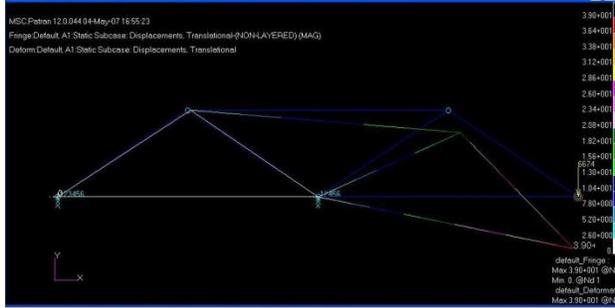


Fig 3 (e)

Displacement Result

**Stress Results:** We get the Maximum stress as 160 N/mm<sup>2</sup> and the Minimum stress as 53.5 N/mm<sup>2</sup>

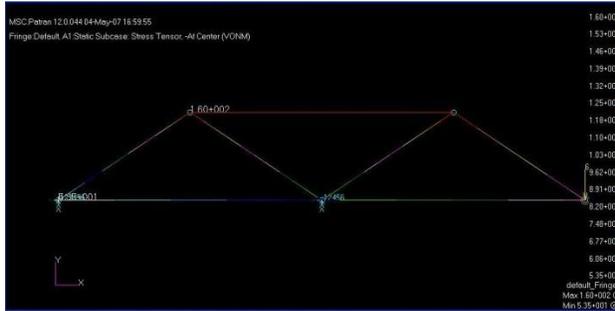


Fig 3 (f) Stress Result

#### IV. SOFTWARE RESULTS

Table No. 4.A Software Stress Strain Results

Items	Maximum	Minimum
Stress	160 Mpa	53.5 Mpa
Displacement	39 mm	0

#### V. RESULTS AND DISCUSSIONS

Table No. 5.A Stress Strain Results

Method	FEM Approach	Nastran & Patran Software
Stress	Maximum :148 Mpa	Maximum : 160 Mpa
	Minimum :42 Mpa	Minimum :53.5 Mpa
Strain	Maximum :39	Maximum :39 mm
	Minimum :0.052mm	Minimum :0 mm

#### VI. CONCLUSION

1. It is seen that the results obtained by FEA simulation is near to analytical solutions.
2. We can converge the simulation of FEA simulation by increasing that no of the elements.
3. Analytical method is observed to be tedious time consuming so the FEA software's are useful to save time.
4. By changing the material we can improve the stability of the structure.

#### REFERENCES

1. Wikipedia website [www.wikipedia.com](http://www.wikipedia.com)
2. Engineers garage website [www.engineersgarage.com](http://www.engineersgarage.com)
3. Design of Machine Elements text Book by V.B.Bhandhari
4. Finite Element Methods text Book by S B Halesh
5. Finite Element Methods text Book by H K Manohar
6. M C Graw's Access science Reference book
7. ASTM Standards & Engg Library
8. Scholar website [www.scholar.com](http://www.scholar.com)
9. Encyclopedia