FLEXIBILITY ANALYSIS OF BEAM USING FINITE ELEMENT ANALYSIS IN MAT LAB

Vinay Prabhakar[1], Pradyumna Dashora[2]

[1]M.Tech. Student, Department of Civil Engineering, Pacific Academy of Higher Education And Research University, Udaipur, Rajasthan, India

[2]Assistant Professor, Department of Civil Engineering, Pacific Academy of Higher Education And Research University, Udaipur, Rajasthan, India

---***---

Abstract:-This paper presents the derivation and implementation of Analysis of beam using finite element method in Mat Lab. Now a day the digit computers have revolutiozed the engineering activities. Now a day's analysis of all the simplest and hardest problems or equations is solved with the help of computer programs, that's not only for calculation it also gives the results quickly. The hypothetical methodology is useful to understudies in the long haul as it gives them a more profound comprehension of the arithmetic behind the improvement of the limited component strategy.The aim of this project is to introduce the theory of the finite element method while keeping a balanced approach between its mathematical formulations, programming implementation. The computer implementation is carried out using MATLAB. Matlab is a general language exceptionally intended for managing lattices, making it specific reasonable for programming the limited component strategy.

1. INTRODUCTION

1.1 INTRODUCTION

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Just expressed, a limit esteem issue is a numerical issue where at least one ward factors should fulfill a differential condition wherever inside a known space of free factors and fulfill explicit conditions on the limit of the area. Limit esteem issues are likewise at times called field issues.

The field is the space of interest and frequently addresses an actual construction. The field factors are the ward factors of interest administered by the differential condition.

The limit conditions are the predetermined upsides of the field factors (or related factors, for example, subsidiaries) on the limits of the field.

The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation.

The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field.

Depending on the type of physical problem being analyzed, the field variables may include physical department, temperature, heat flux, and fluid velocity to name only a few.

1.2 SCOPE OF THE PROJECT

This project aims at applying numerical method. An application of numerical method with user friendly interface has been developed which requires input parameters from the user.

In this project work the numerical method concept is used for beam with various problems. In usual methods the calculations of flexural strength various project an attempt has been made for evaluating the flexural strength loading condition span and geometry.

The use of numerical method analysis of beam is section formulation is an innovative idea. This would produce results same as results produced by the conventional methods of formulation but which reduce time taken and less complicate.

1.3 STATEMENT OF PROBLEM

Here this paper aims at applying a numerical method concept is creating an intelligent system for find out flexural strength by finite element analysis section by varying geometry of span and loading condition.

Software with user friendly interface has been developed which requires input parameter from the user with or without technical knowledge about the beam.

In conventional methods the calculation of flexure involves detailed analysis. In this study the flexural strength has been evaluated under a given loading span and geometry without doing a detailed analysis and design to overcome the difficulties in the conventional design method.

1.4 OBJECTIVE OF THE PROJECT

1. To train and validate the numerical finite element analysis of beam is flexural.
2. To develop an intelligent approach for a system for analyzing the beam numerical method.

1.5 ADVANTAGES OF FEA

Advantages of FEA over most other numerical analysis methods

1. FEA is applicable to any field problem, such as heat transfer, stress analysis, magnetic fields, and so on.
2. There is no geometric restriction: It can be applied the body or region with any shape.
3. Boundary conditions and loading are not restricted (boundary conditions and loads may be applied to any portion of the body)
4. Material properties may be change from one element to another (even within an element and the material anisotropy is allowed.
5. Different elements (behavior and mathematical descriptions can be combined in a single FE model.
6. An FE structure closely resembles the actual body or region to be analyzed.

---

7   The approximation is easily improved by grading the mesh (mesh refinement).

In industry FEA is for the most part utilized in the investigation and streamlining stage to diminish the measure of model testing and to reproduce plans that are not reasonable for model testing. Programmatic experience permits numerous "consider the possibility that Scenarios to be tried rapidly and viably.

The model for the subsequent explanation is careful inserts, like a fake knee. Then again, different explanations behind inclination of the FEM are cost investment funds, time reserve funds, decreasing chance to showcase, making more dependable and better-quality plan.

2. LITERATURE REVIEW

2.1 GENERAL

The substantial pressure strain connection is gotten from comparable uniaxial relations in the tomahawks of orthotropic. The relationship examines among logical and test results and a few boundary studies are directed with the goal to build up the legitimacy of the proposed models and distinguish the meaning of different consequences for the nearby and worldwide reaction of supported substantial individuals. These examinations show that the impacts of strain hardening and bond-slip are vital and ought to consistently be remembered for limited component models of the reaction of built up substantial individuals. Then again, boundaries, like the rigidity of concrete and the worth of the broke shear steady, don't appear to influence the reaction of slim pillars in twisting. Catchphrases—stress; radiates, pieces, monotonic burdens built up concrete.

S.Rao. Gunakala Department of Mathematics and Statistics has distributed a diary on 8-october-2012 The upward avoidance of an essentially upheld and cinched pillar is considered under a uniform burden utilizing the limited component strategy. The issue is settled utilizing homogenous and non-homogenous limit conditions with different quantities of components. Cubic components are utilized as needed for progression. Diagrams are introduced and talked about for various burdens for each situation.

The demonstrating, reproduction and investigation of cantilever pillar are finished by utilizing ANSYS and MATLAB and hypothetically by limited component technique (FEM) for the assessment of regular recurrence and mode shape. By utilizing Lagrange's condition, the detailing of condition movement for the pillar is determined, through which firmness and mass framework is acquired. Eigen esteem approach is utilized for the computation of normal recurrence and mode shape. The recurrence reaction work (FRF) is acquired by utilizing MATLAB.

FERUM User's Guide the FERUM program was set up throughout the late spring of 1999 fundamentally for educational purposes. With time it has developed to likewise fill in as a proving ground for execution and representation of techniques and arrangement systems for the more thorough work on unwavering quality in Open Sees. An open-source improvement technique has been utilized, inviting different analysts to interface with and utilize the program. This has demonstrated fruitful. Huge commitments have been given by others. FERUM is currently additionally known in the primary

dependability local area after introductions on different gatherings and courses. Adaptation 3.0 of FERUM rented throughout the spring of 2001.

A significant with regards to this current client's Guide is that it doesn't contain a point by point depiction of the contribution to FERUM. That is given in the format input record (layout inputfile.m) gave the center FERUM bundle. FERUM (Finite component Reliability Using Matlab is set of capacities written in the MATLAB script language to perform limited component unwavering quality examination. FERUM can be viewed as a "tool compartment" in MATLAB That is, an assortment of subroutines. This idea is known from the MATLAB climate where different tool compartment bundles are monetarily accessible. FERUM is under consistent turn of events. Remarks and ideas from clients for sure impact how the program develops. As of now, the FERUM focal, where the program can be downloaded free of charge and where refreshed data is accessible, is situated at the web address Finite Element Analysis (FEA) is a technique for mathematical arrangement of field issues a field issue might be assurance of the temperature dispersion in a turbine cutting edge, or computation of the circulation of removals and weights on a helicopter rotor sharp edge.

A field issue is formed by differential conditions or by a basic articulation. Either portrayal might be utilized to plan limited components.

Why the Finite Element Method is important to take care of the designing issues? Insightful answers for the designing issues are conceivable provided that the calculation, stacking and limit states of the issue are basic. In any case it is important to utilize a rough mathematical arrangement like FEM.

The restricted part procedure is at first advanced to focus on the nerves in complex plane designs. Then, it is applied to various fields of continuum mechanics, for instance, heat move. **acwook Rhim and Sung W. Lee** has distributed a diary on "A Victoria Approach to Computational Modeling of Beams Undergoing Finite Rotations" The depiction of limited turns of bar like designs utilizing rotational boundaries isn't the most effective from a computational angle, in view of the non-vectorial nature of limited pivots in three-dimensional space.

In the current review, the traditional inflexible cross-segment supposition that is deserted and the movements of shaft chiefs in three-dimensional space is addressed utilizing vector amounts so the subsequent setup space of a pillar turns into a straight vector space. Issues concerning the appropriate constitutive relations are additionally examined.

Mathematical outcomes represent that the current methodology produces alluring properties for different non-direct issues including exceptionally huge turns contrasted with the customary rotational boundary approach.

The premium in productive and solid displaying of pillar like designs is shown by the huge group of writing committed to this subject. Numerous underlying individuals can be displayed as bars and their applications length a wide scope of designing fields.

An audit of work in every one of these stages is exceptionally essential to have a total comprehension of the course of development of this field. This paper endeavors to accomplish unequivocally this goal.

The investigation of enormous adequacy vibration of

essentially upheld bars can be followed to crafted by Krieger wherein the administering halfway differential conditions were diminished to normal differential conditions and the arrangement was gotten as far as elliptic capacities utilizing a one-term estimation.

Ray& Bert completed exploratory examinations to verity the insightful answers for the non straight vibrations of basically upheld bar and analyzed the arrangement plans like the Assumed Space Mode. Mathematical answers for chosen experiments are contrasted and the insightful outcomes accessible in writing for bar static avoidances, normal frequencies, and clasping loads. The adjusted twisting firmness relating to the current model aggresses intimately with an as of late detailed vivacious arrangement. The most extreme impact of surface energy impacts is noticed for cantilever radiates. The limited component plot gives an effective instrument to investigate, plan, and predicts the mechanical reaction of pillar components experienced in Nano electromechanical frameworks and other Nano scale gadgets.

Nanoscale shafts and plates are the vital parts of nano electromechanical framework (NEMS based sensor and actuator innovation 01]. Understanding the static and dynamic reactions of such shaft and plate structures is consequently of critical significance to the plan and assembling of NEMSs Mechanical and Civil Engineering.

During this brief period the limited component strategy was reached out to the arrangement of direct and nonlinear issues related with creep, steady development or removal, break shutting, heat move, stream of water in permeable media, soil union, unique reaction examination and PC helped learning of primary investigation. During the most recent six years of this period the fields of underlying investigation and continuum mechanics were brought together.

The PC programs created during this early period at Berkeley were uninhibitedly circulated overall permitting rehearsing designers to take care of numerous new issues in underlying mechanics. Subsequently, the examination was quickly moved to the designing calling. By and large the examination was utilized expertly preceding the distribution of a proper paper.

Decreased joining is demonstrated to be a proficient technique for curve pillar issues and chose diminished coordination is observed to be indistinguishable from the blended strategy. The meaning of the investigation is three crease. The blended technique and the decreased incorporation strategies join consistently at the ideal rate concerning the curve thickness boundary, so they are sans licking.

Second, blended strategy and diminished reconciliation keep the super intermingling properties of the standard technique. At last, this is the primary endeavor to explore the super intermingling of limited component techniques for curve shaft issues. We set up two sorts of super combination results: dislodging at the nodal focuses and slope at the Gauss focuses.

2.2 SUMMARY

After reviewing the literature have given a greater insight in understanding the phenomenon of design of finite element analysis. It also helps one to have information and perceive significance of the current status of the problem to be deal with.

3. FINITE ELEMENT FORMULATION
3.1 GENERAL

To develop the finite element equations, a two-node linear Galerkin's method. For Equation 7.4, The residual equations (in analogy with Equation 5.61) are expressed as:

$$\int_{x_1}^{x_2} \left[\frac{k_x d^2 T}{dx^2} + Q - \frac{hP(T - T_a)}{A} \right] N_i(X) A dx = 0, \qquad i = 1,2 \dots \dots (7.6)$$

$$A \int_{x_1}^{x_2} \frac{k_x d^2 T}{dx^2} N_i(X) dx - hP \int_{x_1}^{x_2} T(X) N_i(X) dx$$
$$+ A \int_{x_1}^{x_2} Q N_i(X) dx$$
$$+ hPT_a \int_{x_1}^{x_2} N_i(X) dx = 0, \qquad i = 1,2 \dots \dots . (7.7)$$

Integrating the first term by parts and rearranging:

$$K_2 A \int_{x_1}^{x_2} \frac{dN_i}{dx} \frac{dT}{dx} dx$$
$$+ hP \int_{x_1}^{x_2} T(X) N_i(X) dx = A \int_{x_1}^{x_2} Q N_i(X) dx$$
$$+ hPT_a \int_{x_1}^{x_2} N_i(X) dx$$
$$+ K_x A N(X) \frac{dT}{dx} \Big|_{x_1}^{x_2} i = 1,2 \dots \dots \dots \dots . (7.8)$$

Substituting for T(x) from Equation yields:

$$\int_{x_1}^{x_2} \frac{dN_i}{dx} \left[\frac{dN_1 dN_1}{dx} + \frac{dN_2 dN_1}{dx} \right] dx$$
$$+ hP \int_{x_1}^{x_2} N_i(X) dx + [N_i(X) T_1$$
$$+ N_2(X) T_2] dx$$
$$= A \int_{x_1}^{x_2} Q N_i(X) dx$$
$$+ hPT_a \int_{x_1}^{x_2} N_i(X) dx + L_x A N_i(X) \frac{dT}{dx} \Big|_{x_1}^{x_2} i$$
$$= 1,2 \dots (7.9)$$

The two equations represented by Equations 7.9 are conveniently combined into a matrix form by rewriting Equations 7.5 as:

$$T(X) = [N_1 + N_2] \begin{Bmatrix} T_1 \\ T_2 \end{Bmatrix} = [N][T] \dots \dots \dots \dots \dots \dots \dots . \quad (7.10)$$

And substituting to obtain:

$$K_2 A \int_{x_1}^{x_2} \left[\frac{dN^T}{dx}\right]\left[\frac{dN}{dx}\right](T)dx + hP \int_{x_1}^{x_2} [N]^T[N][T]dx$$
$$= A \int_{x_1}^{x_2} Q[N]^T dx$$
$$+ hPTa \int_{x_1}^{x_2} [N]^T dx$$
$$+ KxA[N]^T \frac{dT}{dx}\Big|_{x_1}^{x_2} \dots\dots (7.11)$$

Equation 7.11 is in the desired finite element form:

$$[K^e][T] = \{f^e Q\} + \{f^e h\} + \{f^e g\} \dots\dots\dots (7.12)$$

Where $[K^e]$ the conductance of matrix is is defined as:

$$[k^2] = kxA \int_{x_2}^{x_2} \left[\frac{dN^T}{dx}\right]\left[\frac{dN}{d}\right]dx = hP \int_{x_1}^{x_2}[N]^T[N]dx.. \quad (7.13)$$

The first integral is identical to that in Equation 5.66 representing the axial conduction effect, while the second integral accounts for convection.

Without loss of generally, we let xi s 0, x2 L so that interpolation function are:

$$N_1 = 1 - \frac{x}{L}$$
$$N_2 = \frac{x}{L} \qquad \dots\dots\dots\dots (7.14)$$

The result of the first integral are as given in Equation 5.68, so we need perform only the integrations indicated in the second term (Problem 7.2) to obtain

$$[k^e] = \frac{KxA}{A}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + \frac{hPL}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} = [K_e e][K_h e] \dots\dots (7.15)$$

Where $[k^e]$ and $[k_e e]$ represent the conductive and convective Portions of the matrix, respectively. Note particularly that both portions are symmetric.

The forcing function vector on the right-hand side Equations 7.12 include the internal heat generation and boundary flux terms, as in chapter 5.
These are given by:

$$\{f_Q e\} = A \begin{Bmatrix} \int_0^L QN_1 \\ 0 \\ \int_0^L QN_2 dx \\ 0 \end{Bmatrix} dx \dots\dots\dots\dots\dots\dots\dots\dots (7.16)$$

$$\{f_g e\} = kxA \begin{Bmatrix} \frac{dT}{dx}|0 \\ \frac{dT}{dx}|1 \end{Bmatrix} = A \begin{Bmatrix} qx = 0 \\ -qx = L \end{Bmatrix} A \begin{Bmatrix} q_1 \\ -q_2 \end{Bmatrix} \dots\dots (7.17)$$

Where q_1 and q_2 are the boundary flux values at nodes 1 and 2, respectively. In addition, the forcing function arising from convection is-

$$\{f_g e\} = hPTa \begin{Bmatrix} \int_0^L N_1 dx \\ 0 \\ \int_0^L N_2 dx \\ 0 \end{Bmatrix} = \frac{hPTal}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \dots\dots\dots\dots (7.18)$$

Where it is evident that the total element convection force is simply allocated equally to each node, like constant internal generation Q.
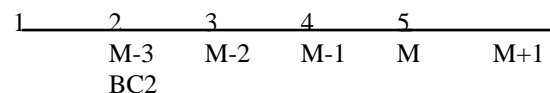
1	2	3	4	5
	M-3	M-2	M-1	M	M+1
	BC2				

Fig -1 Convection boundary condition at node M+1 of an M-Element, One-dimensional heat transfer finite element model.

3.2 METHODS OF ANALYSING BEAM
The solution methods can be classified in three categories.
1. Analytical Methods
2. Approximate Methods and
3. Numerical Methods

3.2.1 Analytical Methods:
They give shut structure careful answers for the numerical model of designing issues. They can be utilized provided that the calculation, stacking and limit states of the issue are straightforward. Joining strategies and other scientific arrangements techniques for differential conditions are the instances of the insightful techniques.

3.2.2 Approximate Methods:
They give shut structure surmised answers for the numerical model of designing issues. They can be utilized provided that the calculation, stacking and limit states of the issue are basic. Rit'z technique, Galerkin's Method, Collocation Methods, Least square strategy, Moment Method, Kantrovich's Method, Etc.

3.2.3 Numerical Methods:
They give discrete structure estimated answer for the numerical model of Engineering issue's. They can be utilized to tackle the issues with moderately complex math, stacking and limit conditions. Specifically limited components can address constructions of self-assertively complex math. Limited Difference Method, Finite Element Method, Boundary Element Method, Etc… …

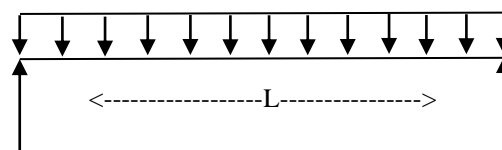## 3.3 ANALYSIS OF A SIMPLY SUPPORTED BEAM SUBJECTED TO UDL.

<-----------------L---------------->

Fig-2 Simply Supported Beam Subjected To UDL.

Let the beam is to consider as element 1.

      Let $V1$, $V2$ = Vertical deflection

      $\Theta1$, $\Theta2$ = Slope

      $F1$, $F2$ = Shear force

$M1$, $M2$ = Bending Moments
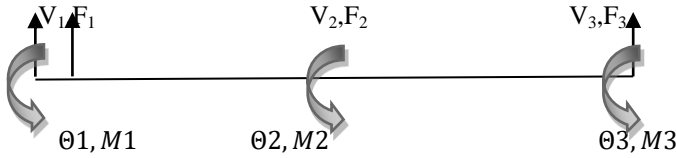
The various loading and deflection are given-

Fig-3 loading and deflection

Determination of nodal deflections-

Element-1-

It has nodal displacement –

    $V1$, $\Theta1$, $V2$, $\Theta2$

The element stiffness matrix for element is given by-

$$K = \frac{EI}{L1^3} \begin{array}{cccc} V1 & \Theta1 & V2 & \Theta2 \\ \begin{bmatrix} 12 & 6L1 & -12 & 6L1 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 \\ -12 & -6L1 & 12 & -6L1 \\ -6L1 & 2L1^2 & -6L1 & 4L1^2 \end{bmatrix} & \begin{array}{c} V1 \\ \Theta1 \\ V2 \\ \Theta2 \end{array} \end{array}$$

Element-2 –

The element stiffness matrix for element is given by-

$$K = \frac{EI}{L2^3} \begin{array}{cccc} V2 & \Theta2 & V3 & \Theta3 \\ \begin{bmatrix} 12 & 6L2 & -12 & 6L2 \\ 6L2 & 4L2^2 & -6L2 & 2L2^2 \\ -12 & -6L2 & 12 & -6L2 \\ -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} & \begin{array}{c} V2 \\ \Theta2 \\ V3 \\ \Theta3 \end{array} \end{array}$$

The global stiffness matrix can be written as –

$$K = \frac{EI}{L1^3} \begin{array}{cccccc} V1 & \Theta1 & V2 & \Theta2 & V3 & \Theta3 \\ \begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & 12 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} & \begin{array}{c} V1 \\ \Theta1 \\ V2 \\ \Theta2 \\ V3 \\ \Theta3 \end{array} \end{array}$$

The finite element equation is -

$$[F] = [K]\,[\Delta]$$

$$\begin{bmatrix} F1 \\ M1 \\ F2 \\ M2 \\ F3 \\ M3 \end{bmatrix} = \frac{EI}{L1^3} \begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & 12 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} \begin{bmatrix} V1 \\ \Theta1 \\ V2 \\ \Theta2 \\ V3 \\ \Theta3 \end{bmatrix}$$

The nodal conditions are –

$V1=0$, $M1$ & $M3=0$ (Simply Supported ends)

$V3=0$, $F1=-WL$, $F3=-WL$, $\Theta2=0$

$$\begin{bmatrix} W/2 \\ 0 \\ F2 \\ M2 \\ -W/2 \\ 0 \end{bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & 12 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} \begin{bmatrix} 0 \\ \Theta1 \\ V2 \\ 0 \\ 0 \\ \Theta3 \end{bmatrix}$$

$$\begin{bmatrix} W/2 \\ 0 \\ F2 \\ M2 \\ -W/2 \\ 0 \end{bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & 12 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} \begin{bmatrix} 0 \\ \Theta1 \\ V2 \\ 0 \\ 0 \\ \Theta3 \end{bmatrix}$$

$$K = \frac{EI}{L^3} \begin{bmatrix} 4L1^2 & -6L1 & 0 & 0 \\ -6L1 & 2L1 & -12 & 6L2 \\ 0 & -12 & 12 & -6L2 \\ 0 & -6L2 & -6L2 & 4L2^2 \end{bmatrix} \begin{bmatrix} \theta 1 \\ V2 \\ V3 \\ \theta \end{bmatrix}$$

$$\frac{EI}{L^3}(4L1^2\theta 1 - 6L1V2) = 0 \dots\dots\dots\dots\dots\dots [i]$$

$$\frac{EI}{L^3}(-6L1\theta 1 + 24V2 + 6L2\theta 3) = F2 \dots\dots\dots\dots [ii]$$

$$\frac{EI}{L^3}(-6L2V2 + 4L2^2\theta 3) = 0 \dots\dots\dots\dots [iii]$$

From equation – [i]

$$\theta 1 = \frac{6L1V2}{4L1^2}$$

Substitute θ1 in Equation [i], Find V2

By Solving this equations, Find θ3

Shear force and Bending Moments –

$$F2 = \frac{EI}{L^3}(-6L1\theta 1 + 24V2 - 12\theta 3) = F2 \dots\dots\dots [i]$$

$$M2 = \frac{E1}{L^3}(2L1^2\theta 1 + (-6L2)V2 + 12V3 - 6L2\theta 3)$$
$$= F2 \dots [ii]$$

Diagram:

SFD

BMD

Fig-4 SF, BF and Deflection Diagram

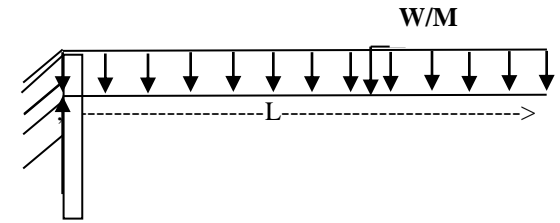## 3.4 ANALYSIS OF A CANTILEVER BEAM SUBJECTED TO UDL.

W/M

Fig-5 Cantilever Beam subjected to UDL.

Let the beam is to consider as element 1.
Let V1, V2 = Vertical deflection
θ1, θ2 = Slope
F1, F2 = Shear force
M1, M2 = Bending Moments
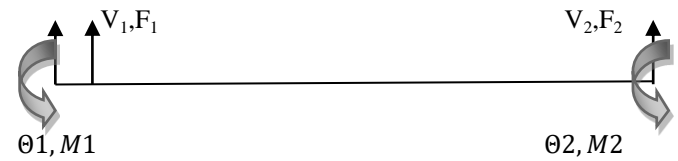
The various loading and deflection are given-

V_1, F_1 V_2, F_2

$\theta 1, M1$ $\theta 2, M2$

Fig-6 loading and deflection

Determination of nodal deflections-

Element-1

It has nodal displacement
V1, θ1, V2, θ2
The element stiffness matrix for element is given by-

$$K = \frac{EI}{L^3} \begin{array}{cccc} V1 & \theta 1 & V2 & \theta 2 \\ \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ -6L & 2L^2 & -6L & 4L^2 \end{bmatrix} & & & \begin{array}{c} V1 \\ \theta 1 \\ V2 \\ \theta \end{array} \end{array}$$
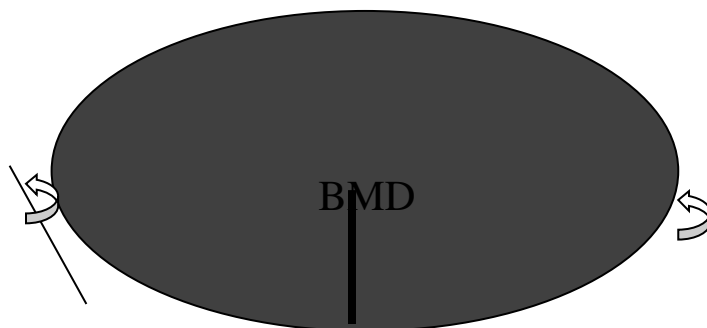
The global stiffness matrix can be written as

$$K = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ -6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

The finite element equation is-

$$[F] = [K][\Delta]$$

$$\begin{bmatrix} F1 \\ F2 \\ F3 \\ F4 \end{bmatrix} = \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ -6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} V1 \\ \theta1 \\ V2 \\ \theta2 \end{bmatrix}$$

The nodal conditions are-

V1 = O, θ = O, M2 = O

$$\begin{bmatrix} F1 \\ F2 \\ F3 \\ O \end{bmatrix} = EI/L^3 \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ -6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} O \\ O \\ V2 \\ \theta2 \end{bmatrix}$$

By neglecting 1st and 2nd rows and column-

$$\begin{bmatrix} F2 \\ O \end{bmatrix} = EI/L^3 \begin{bmatrix} 12 & 6L \\ -6L & 4L^2 \end{bmatrix} \begin{bmatrix} V2 \\ \theta1 \end{bmatrix}$$

F2 = EI/L^3(12V2 + 6L) ……………………….. [i]

EI/L^3(-6LV2 + 4L^2) = O ………………… [ii]

        F2 = -W*L

Equation – [i]

EI/L^3(12V2 + 6L) = -WL

12V2 +) = -WL^4/EI

$$\boxed{V2 = \frac{WL^4}{12EI} + 6L}$$

Equation – [ii]

-6LV2 + 4L^2 = O*L^3/EI

 4L^2 = 6LV2

$$\boxed{\theta = \frac{6LV}{4L^2}}$$

Shear force at node – [i]

F1 = EI/L^3 (−12V2 + 6L) = O

Bending moment at node – [i]

M1 = (-6LV2 +2L^2)EI/L^3

Diagram –
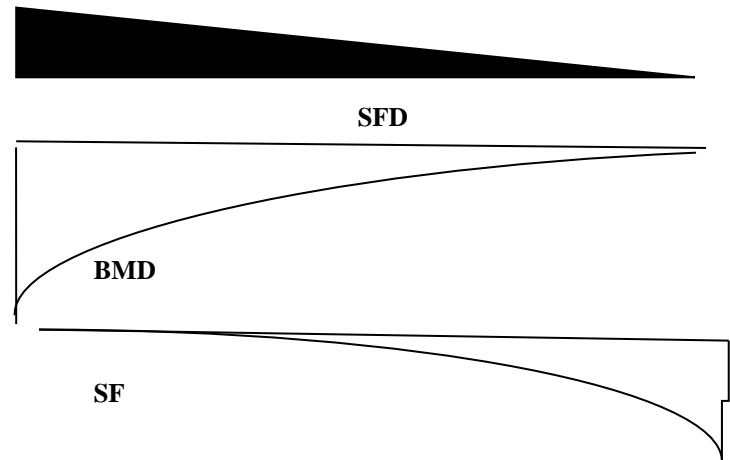
Fig-7 SF, BF and Deflection Diagram

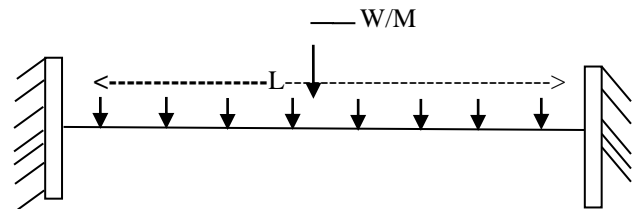## 2.5 ANALYSIS OF A FIXED BEAM SUBJECTED TO UDL.

Fag-8 Fixed Beam Subjected to Udl.

Let the beam is to consider as element 1.

Let V1, V2 = Vertical deflection

Θ1, Θ2 = Slope

  F1, F2 = Shear force

  M1, M2 = Bending Moments
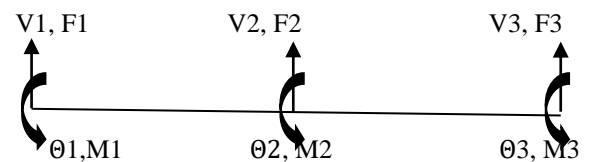
The various loading and deflection are given-

Fig-9 loading and deflection

Determination of nodal deflections-

Element-1

It has nodal displacement –

V1, Θ1, V2, Θ2

The element stiffness matrix for element [i] is given by-

$$K = \frac{EI}{L1^3}\begin{array}{cccc} V1 & \Theta1 & V2 & \Theta2 \\ \begin{bmatrix} 12 & 6L1 & -12 & 6L1 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 \\ -12 & -6L1 & 12 & -6L1 \\ -6L1 & 2L1^2 & -6L1 & 4L1^2 \end{bmatrix} & \begin{array}{c} V1 \\ \Theta1 \\ V2 \\ \Theta2 \end{array} \end{array}$$

Element-2

The element stiffness matrix for element [ii] is given by-

$$K = \frac{EI}{L2^3}\begin{array}{cccc} V2 & \Theta2 & V3 & \Theta3 \\ \begin{bmatrix} 12 & 6L2 & -12 & 6L2 \\ 6L2 & 4L2^2 & -6L2 & 2L2^2 \\ -12 & -6L2 & 12 & -6L2 \\ -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} & \begin{array}{c} V2 \\ \Theta2 \\ V3 \\ \Theta3 \end{array} \end{array}$$

The global stiffness matrix can be written as-

The finite element equation is-

  [F] = [K] [Δ]

The nodal conditions are-

V1 = O, M1 & M3 = IO (Simply Supported ends)

V3 = O, F1 = -WL, F3 = -WL, Θ2 = O

$$K = \frac{EI}{L1^3}\begin{array}{cccccc} V1 & \Theta1 & V2 & \Theta2 & V3 & \Theta3 \\ \begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L2 \\ 6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & -6L2 & 2L2^2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & 6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix} & \begin{array}{c} V1 \\ \Theta1 \\ V2 \\ \Theta2 \\ V3 \\ \Theta3 \end{array} \end{array}$$

$$\begin{bmatrix} F1 \\ M1 \\ F2 \\ M2 \\ F3 \\ M3 \end{bmatrix} = \frac{EI}{L1^3}\begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & -6L2 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & 6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix}\begin{bmatrix} V1 \\ \Theta1 \\ V2 \\ \Theta2 \\ V3 \\ \Theta3 \end{bmatrix}$$

$$\begin{bmatrix} W/2 \\ 0 \\ F2 \\ M2 \\ -W/2 \\ 0 \end{bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & 12 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix}\begin{bmatrix} 0 \\ \Theta1 \\ V2 \\ 0 \\ 0 \\ \Theta3 \end{bmatrix}$$

$$\begin{bmatrix} W/2 \\ 0 \\ F2 \\ M2 \\ -W/2 \\ 0 \end{bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L1 & -12 & 6L1 & 0 & 0 \\ 6L1 & 4L1^2 & -6L1 & 2L1^2 & 0 & 0 \\ -12 & -6L1 & 12+12 & -6L1+6L2 & -12 & 6L22 \\ -6L1 & 2L1^2 & -6L1+6L2 & 4L1^2+4L2^2 & 12 & -6L2 \\ 0 & 0 & -12 & -6L2 & 12 & -6L2 \\ 0 & 0 & -6L2 & 2L2^2 & -6L2 & 4L2^2 \end{bmatrix}\begin{bmatrix} 0 \\ \Theta1 \\ V2 \\ 0 \\ 0 \\ \Theta3 \end{bmatrix}$$

$$K = \frac{EI}{L^3}\begin{bmatrix} 4L1^2 & -6L1 & O & O \\ -6L1 & 2L1 & -12 & 6L2 \\ O & -12 & 12 & -6L2 \\ O & -6L2 & -6L2 & 4L2^2 \end{bmatrix}\begin{bmatrix} \Theta1 \\ V2 \\ V3 \\ \Theta \end{bmatrix}$$

$$\frac{EI}{L^3}(4L1^2\,\Theta1 - 6L1V2) = 0 \dots\dots\dots\dots\dots\dots\dots\dots\dots\dots.\ [\text{i}]$$

$$\frac{EI}{L^3}(-6L1\Theta1 + 24V2 + 6L2\Theta3) = F2 \dots\dots\dots\dots\dots\dots.\ [\text{ii}]$$

$$\frac{EI}{L^3}(-6L2V2 + 4L1^2\Theta3) = 0 \dots\dots\dots\dots\dots\dots\dots\dots.\ [\text{iii}]$$

From equation – [i]

$$\boxed{\Theta1 = \frac{6L1V2}{4L1^2}}$$

Substitute Θ1 in Equation [i]

Find V2

By Solving this equations,

Find Θ3

Shear force and Bending Moments –

$$F2 = \frac{EI}{L^3}(-6L1\Theta1 + 24V2 + 12\Theta3) = F2 \dots\dots\dots\dots\dots.\ [\text{i}]$$

$$M2 = \frac{EI}{L^3}(2L1^2\Theta1 + (-6L1 + 6L2)V2 + 12V3 - 6L2\Theta3)$$
$$= F2 \dots\ [\text{ii}]$$
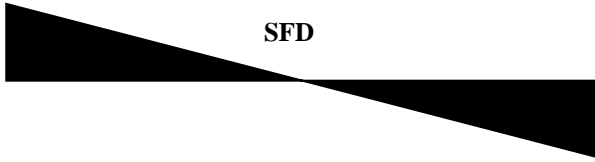
Diagram

SFD

Fig-10 SF, BF and Deflection Diagram

4. COMPUTER APPLICATION PROGRAMING

4.1 INTRODUCTION TO MATLAB

MATLAB incorporates numerical processing, perception, and an amazing language to give an adaptable climate to specialized figuring. The open design makes it simple to utilize MATLAB and its sidekick items to investigate information, make calculations and make custom devices that give early experiences and upper hands.

Known for its profoundly enhanced framework and vector computations, MATLAB offers a natural language for communicating issues and their answers both numerically and outwardly.

Typical uses include:

1 Numeric computation and algorithm development.

2 Symbolic computation (with the built-in Symbolic Math functions).

3 Modeling, simulation and prototyping.

4 Data analysis and signal processing.

5 Engineering graphics and scientific visualization.

In this chapter, we will present the MATLAB climate. We will figure out how to make, alter, save, run and troubleshoot M-records (ASCII documents with series of MATLAB proclamations). We will perceive how to make exhibits (grids and vectors, and investigate the underlying MATLAB direct polynomial math capacities for lattice and vector increase, speck and cross items, translate, determinants and inverses, and for the arrangement of straight conditions.

MATLAB depends on the language C, yet is by and large a lot simpler to utilize We will likewise perceive how to program rationale develops and circles in MATLAB, how to utilize subprograms and capacities, how to utilize remarks for clarifying the projects and tabs for simple meaningfulness, and how to print and plot illustrations both two and Three dimensional.

MATLAB's capacities for representative arithmetic are introduced. Utilization of these capacities to perform emblematic activities, to foster shut structure articulations for answers for logarithmic conditions, normal.

4.2 ELEMENTARY MATH BUILT-IN FUNCTION:

Table-1 Common Math function

Function	Description
Abs(x)	Computes the absolute value of x

Function	Description
Sqrt(x)	Computes the square root of x
Round(x)	Rounds x to the nearest integer.
Fix(x)	Rounds (or truncates) x to the nearest integer toward.
Floor(x)	Rounds x to the nearest integer toward $-\infty$.
Ceil(x)	Rounds x to the nearest integer toward ∞.
Sign(x)	Returns a value of -1 if x is less than 0, a value of 0 if x equals 0 and a value of 1 otherwise.
Rem(x,y)	Returns the remainder of x/y. for example, rem (25, 4) is 1, and rem (100, 21) is 16. This function is also called a modulus function.
Exp(x)	Computes e^x, where e is the base for natural logarithms, or Approximately 2.718282.
Log(x)	Computes in x, the natural logarithm of x to the base e.
Log10(x)	Computes log10 x, the common logarithm of x to the base 10.

MATLAB contains a number of functions for performing computations which require the use of logarithms, Elementary math functions and trigonometric math functions. List of these commonly used elementary.

Table-2 Exponential functions

Function	Description
Exp(x)	Exponential.
Log(x)	Natural logarithm.
Log10(x)	Base 10 logarithm.
Sqrt(x)	Square root.

4.2.1 VARIABLE NAMES

A variable is a name made of a letter or a mix of a few letters and digits. Variable names can be up to 63 (in MATLAB characters long (31 characters on MATLAB 6.0). MATLAB is case touchy. For example, X Xx, xX and xx are the names of four distinct factors. It ought to be noted here that not to utilize the names of an implicit capacities for a variable. For example, abstain from utilizing: sin, cos, exp, sqrt, and so on When a capacity name is utilized to characterize a variable, the capacity can't be utilized.

4.2.2 ARRAY

An exhibit is a rundown of numbers masterminded in lines as well as segments. A one-dimensional exhibit is a line or a segment of numbers and a two-dimensional cluster has a bunch of numbers orchestrated in lines and segments. An

---

exhibit activity is performed component by-component.

4.3.3 ROW VECTOR

A vector is a line or section of components. In succession vector, the components are entered with a space or a comma between the components inside the square sections.For example, x = [7 -1 2-5 8]

4.2.4 COLUMN VECTOR:

In a column vector, the elements are entered with a semicolon between the elements inside the square brackets. For example, x = [7; -1; 2; -5; 8].

4.2.5 MATRIX

A matrix is a two-dimensional array which has numbers in rows and columns. A matrix is entered row-wise with consecutive elements of a row separated by a space or a comma, and the rows separated by semicolons or carriage returns. The entire matrix is enclosed within square brackets. The elements of the matrix may be real numbers or complex numbers. For example, to enter the matrix,

The MATLAB input command is- A = [1 3 -4 0 -2 8]

5. TEST EXAMPLE

5.1 TEST EXAMPLE
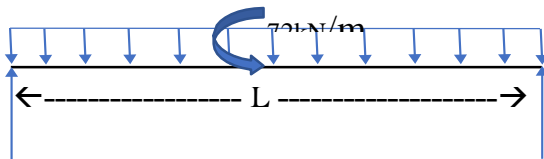
Fig-13 Simply Supported Beam Subjected UDL.

❖ Beam (900*150*150) is casted with reinforcement (4 no. of bars & 6 Stirrups) compressive strength Deflection results

5.2 MIX DESIGN FOR HIGH STRENGTH CONCRETE (M20)

1 . Stipulation for proportioning:

1. Grade Designation – M20 (1:1.5:3)

2. Type of cement – OPC 43 grade cement c .Maximum size of aggregate – 20mm
3. Maximum water-cement ratio – 0.50 e .Exposure conditions – Severe
4. Degree of supervision – Good

5. Type of aggregate – Course aggregate

5.3 TARGET STRENGTH FOR MIX PROPORTIONS.

Fck' = Fck + 1.65s= 20 + (1.65 * 5) = 28.25 N/mm2

5.4 SELECTION OF WATER CEMENT RATIO.

Maximum water cement ratio = 0.50 (From table 5; IS 456)

5.5 SELECTION OF WATER CONTENT.

Estimated content of water = 8.5L

5.6 CASTING AND CURING OF SIMPLY SUPPORTED BEAM:

Fig-14 Casting and Curing of Simply Supported Beam

Fig-15 Testing of Simply Supporting beam in Universal Testing Machine

5.9 DEFLECTION AND COMPRESSIVE STRENGTH

Table-3 Deflection and Compressive Strength

S.No.	Size	Ultimate Load	Deflection	Bending Moment	Compressive Strength
1.	(900*150* 150)mm	64.800 KN	10.85 mm	7.20kNm	2.880N/m m2

ANALYSIS USING STAAD PRO:-

Beams

Beam	Node A	Node B	Length (m)	Property	β (degrees)
1	1	2	0.180	1	0
2	2	3	0.180	1	0
3	3	4	0.180	1	0
4	4	5	0.180	1	0
5	5	6	0.180	1	0

Section Properties

Prop	Section	Area (in²)	I_{yy} (in⁴)	I_{zz} (in⁴)	J (in⁴)	Material
1	Rect 5.91x5.91	34.875	101.355	101.355	171.037	CONCRETE

Supports

Node	X (kip/in)	Y (kip/in)	Z (kip/in)	rX (kip·ft/deg)	rY (kip·ft/deg)	rZ (kip·ft/deg)
1	Fixed	Fixed	Fixed	-	-	-
6	Fixed	Fixed	Fixed	-	-	-

Combination Load Cases

Comb.	Combination L/C Name	Primary	Primary L/C Name	Factor
3	COMBINATION LOAD CASE 3	1	LL	1.00
		2	DL	1.00
4	COMBINATION LOAD CASE 4	1	LL	1.50
		2	DL	1.50

Beam Loads : 1 LL

Beam	Type	Direction	Fa	Da (m)	Fb	Db	Ecc. (m)
1	UNI lbf/ft	GY	-4.88E+3	-	-	-	-
2	UNI lbf/ft	GY	-4.88E+3	-	-	-	-
3	UNI lbf/ft	GY	-4.88E+3	-	-	-	-
4	UNI lbf/ft	GY	-4.88E+3	-	-	-	-
5	UNI lbf/ft	GY	-4.88E+3	-	-	-	-

Selfweight : 2 DL

Direction	Factor
Y	-1.000

Load 1

LOADINGS
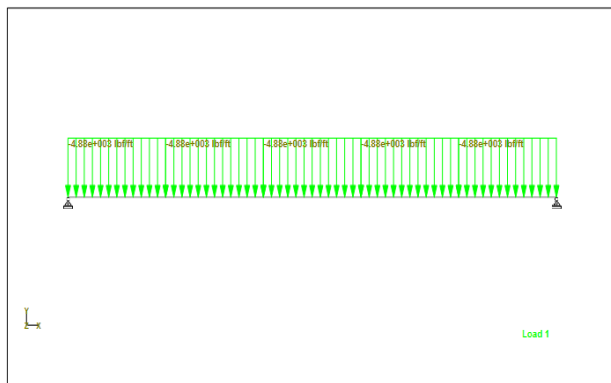
Node Displacements

Node	L/C	X (in)	Y (in)	Z (in)	Resultant (in)	rX (rad)	rY (rad)	rZ (rad)
1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
	2:DL	0.000	0.000	0.000	0.000	0.000	0.000	-0.000
	3:COMBINATIC	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
	4:COMBINATIC	0.000	0.000	0.000	0.000	0.000	0.000	-0.004
2	1:LL	0.000	-0.017	0.000	0.017	0.000	0.000	-0.002
	2:DL	0.000	-0.000	0.000	0.000	0.000	0.000	-0.000
	3:COMBINATIC	0.000	-0.017	0.000	0.017	0.000	0.000	-0.002
	4:COMBINATIC	0.000	-0.025	0.000	0.025	0.000	0.000	-0.003
3	1:LL	0.000	-0.026	0.000	0.026	0.000	0.000	-0.001
	2:DL	0.000	-0.000	0.000	0.000	0.000	0.000	-0.000
	3:COMBINATIC	0.000	-0.027	0.000	0.027	0.000	0.000	-0.001
	4:COMBINATIC	0.000	-0.040	0.000	0.040	0.000	0.000	-0.001
4	1:LL	0.000	-0.026	0.000	0.026	0.000	0.000	0.001
	2:DL	0.000	-0.000	0.000	0.000	0.000	0.000	0.000
	3:COMBINATIC	0.000	-0.027	0.000	0.027	0.000	0.000	0.001
	4:COMBINATIC	0.000	-0.040	0.000	0.040	0.000	0.000	0.001
5	1:LL	0.000	-0.017	0.000	0.017	0.000	0.000	0.002
	2:DL	0.000	-0.000	0.000	0.000	0.000	0.000	0.000
	3:COMBINATIC	0.000	-0.017	0.000	0.017	0.000	0.000	0.002
	4:COMBINATIC	0.000	-0.025	0.000	0.025	0.000	0.000	0.003
6	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	0.002
	2:DL	0.000	0.000	0.000	0.000	0.000	0.000	0.000
	3:COMBINATIC	0.000	0.000	0.000	0.000	0.000	0.000	0.002
	4:COMBINATIC	0.000	0.000	0.000	0.000	0.000	0.000	0.004

Node Displacement Summary

	Node	L/C	X (in)	Y (in)	Z (in)	Resultant (in)	rX (rad)	rY (rad)	rZ (rad)
Max X	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Min X	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Max Y	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Min Y	4	4:COMBINATIC	0.000	-0.040	0.000	0.040	0.000	0.000	0.001
Max Z	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Min Z	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Max rX	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Min rX	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Max rY	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Min rY	1	1:LL	0.000	0.000	0.000	0.000	0.000	0.000	-0.002
Max rZ	6	4:COMBINATIC	0.000	0.000	0.000	0.000	0.000	0.000	0.004
Min rZ	1	4:COMBINATIC	0.000	0.000	0.000	0.000	0.000	0.000	-0.004
Max Rst	4	4:COMBINATIC	0.000	-0.040	0.000	0.040	0.000	0.000	0.001

Beam End Displacement Summary

Displacements shown in italic indicate the presence of an offset

	Beam	Node	L/C	X (in)	Y (in)	Z (in)	Resultant (in)
Max X	1	1	1:LL	**0.000**	0.000	0.000	0.000
Min X	1	1	1:LL	**0.000**	0.000	0.000	0.000
Max Y	1	1	1:LL	0.000	**0.000**	0.000	0.000
Min Y	3	4	4:COMBINATI(	0.000	**-0.040**	0.000	0.040
Max Z	1	1	1:LL	0.000	0.000	**0.000**	0.000
Min Z	1	1	1:LL	0.000	0.000	**0.000**	0.000
Max Rst	3	4	4:COMBINATI(	0.000	-0.040	0.000	**0.040**

Beam Maximum Moments

Distances to maxima are given from beam end A.

Beam	Node A	Length (m)	L/C		d (m)	Max My (kip·in)	d (m)	Max Mz (kip·in)
1	1	0.180	1:LL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-40.835
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-0.305
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-41.140
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-61.710
2	2	0.180	1:LL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-61.253
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-0.457
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-61.710
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-92.565
3	3	0.180	1:LL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.090	-63.805
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.090	-0.476
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.090	-64.281
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.090	-96.422
4	4	0.180	1:LL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-61.253

Beam Maximum Moments Cont...

Beam	Node A	Length (m)	L/C		d (m)	Max My (kip·in)	d (m)	Max Mz (kip·in)
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-0.457
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-61.710
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-92.565
5	5	0.180	1:LL	Max -ve	0.000	0.000	0.180	-0.000
				Max +ve	0.000	0.000	0.000	-40.836
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-0.305
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-41.140
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.000	-61.710

Beam Maximum Shear Forces

Distances to maxima are given from beam end A.

Beam	Node A	Length (m)	L/C		d (m)	Max Fz (kN)	d (m)	Max Fy (kN)
1	1	0.180	1:LL	Max -ve	0.000	0.000	0.000	32.040
				Max +ve	0.000	0.000		
			2:DL	Max -ve	0.000	0.000	0.000	0.239
				Max +ve	0.000	0.000		
			3:COMBINATI(	Max -ve	0.000	0.000	0.000	32.279
				Max +ve	0.000	0.000		
			4:COMBINATI(	Max -ve	0.000	0.000	0.000	48.419
				Max +ve	0.000	0.000		
2	2	0.180	1:LL	Max -ve	0.000	0.000	0.000	19.224
				Max +ve	0.000	0.000		
			2:DL	Max -ve	0.000	0.000	0.000	0.143
				Max +ve	0.000	0.000		
			3:COMBINATI(	Max -ve	0.000	0.000	0.000	19.368
				Max +ve	0.000	0.000		
			4:COMBINATI(	Max -ve	0.000	0.000	0.000	29.051
				Max +ve	0.000	0.000		
3	3	0.180	1:LL	Max -ve	0.000	0.000	0.000	6.408
				Max +ve	0.000	0.000	0.180	-6.408
			2:DL	Max -ve	0.000	0.000	0.000	0.048
				Max +ve	0.000	0.000	0.180	-0.048
			3:COMBINATI(	Max -ve	0.000	0.000	0.000	6.456
				Max +ve	0.000	0.000	0.180	-6.456
			4:COMBINATI(	Max -ve	0.000	0.000	0.000	9.684
				Max +ve	0.000	0.000	0.180	-9.684
4	4	0.180	1:LL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-19.224
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-0.143
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-19.367
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-29.051
5	5	0.180	1:LL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-32.040
			2:DL	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-0.239
			3:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-32.279
			4:COMBINATI(	Max -ve	0.000	0.000		
				Max +ve	0.000	0.000	0.180	-48.419

6. RESULT AND DISCUSSIONS

The nodal Deflection result is to be compared with Test example 1, STAAD Pro and MAT Lab.
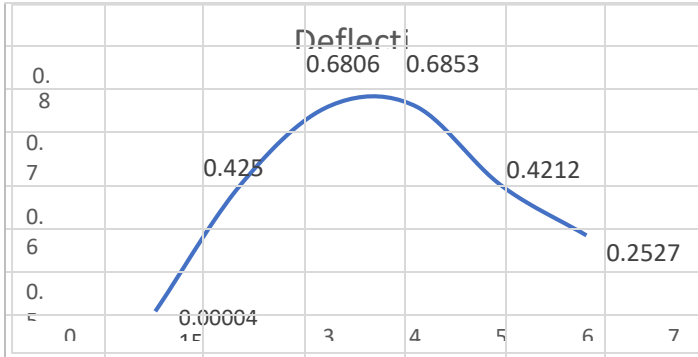
MAT Lab DEFLECTION GRAPH:

Fig-16 The graph represent the deflection for MAT Lab.
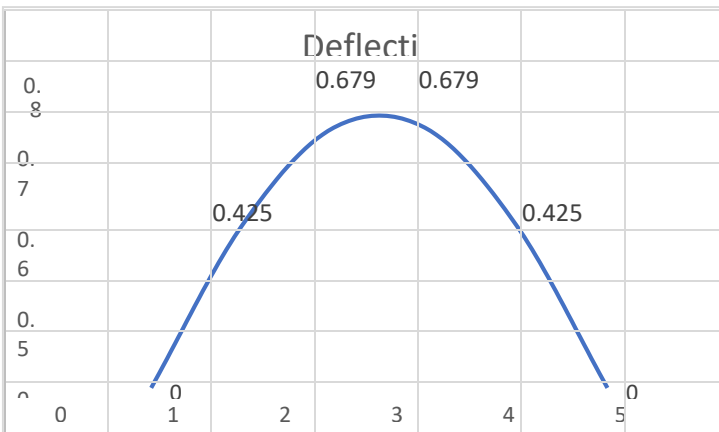
STAAD Pro. DEFLECTION GRAPH:

Fig-17 The graph represents the Deflection in STAAD pro.
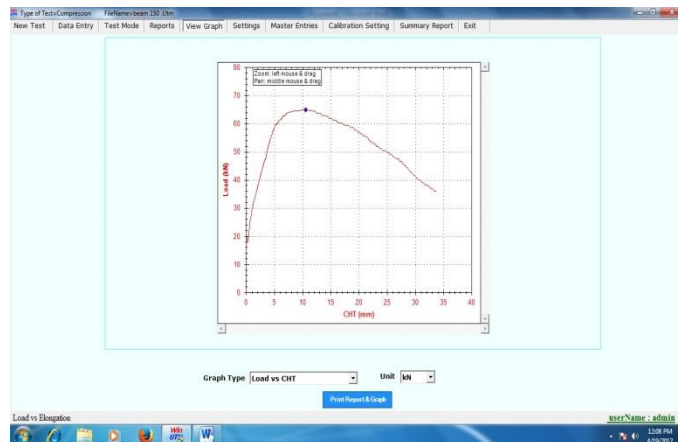
EXPERIMENTAL PART GRAPH:
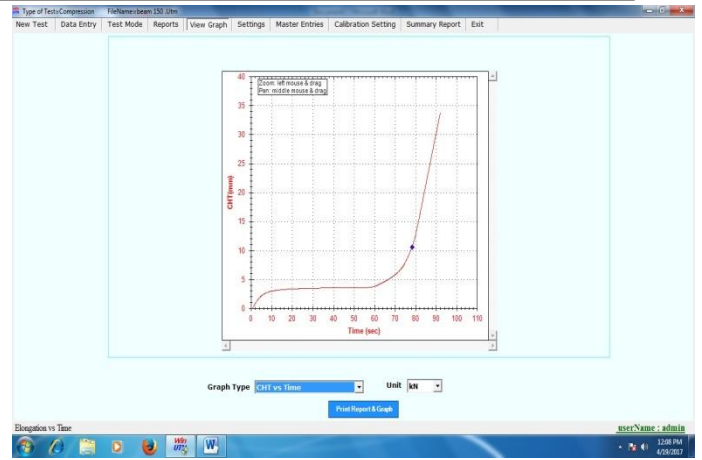

Fig-18 Graph represents Load Vs Time.

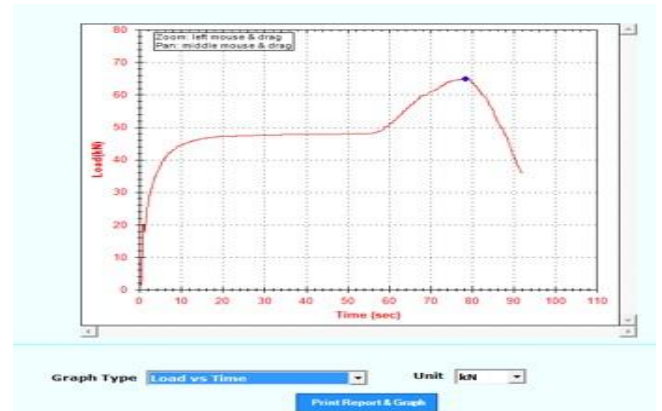**Fig-19 Graph represents Deflection Vs Time.**

Fig-20 Graph represent Load Vs Deflection

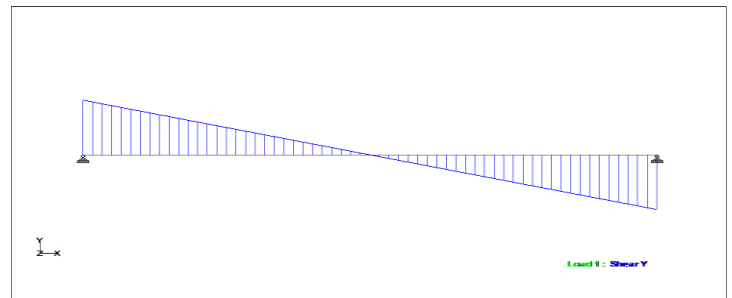### STAAD Pro. ANALYSED DIAGRAM:
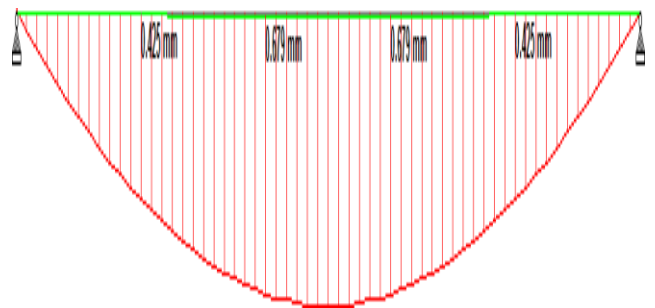
Fig-21 Deflection Digram

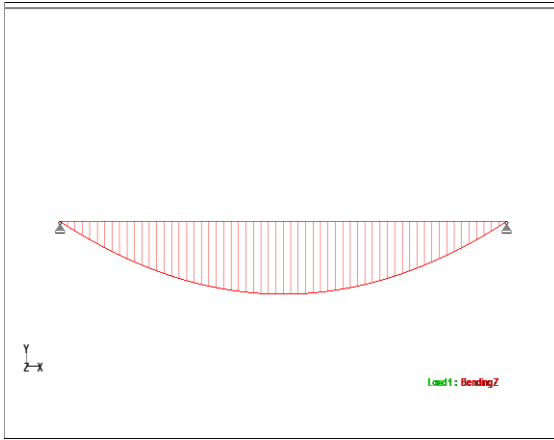**Fig-22 Shear force Diagram.**

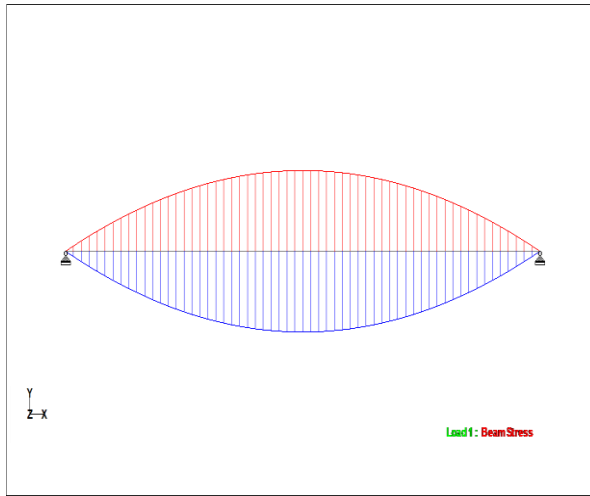**Fig-23 Bending Moment Diagram**

Fig-24 Beam Stress Diagram

6.2 COMPARISON OF DEFLECTIONS OF SIMPLY SUPPORTED BEAM

Table-5 Deflection of Test examples of MAT Lab, STAAD Pro and Experiment.

Sl.No	al Deflection Values In **Mat Lab** (mm)	dal Deflection Values In **Staad Pro.** (mm)	Max. Deflection Values In **Experimental Part** (mm)
1.	0.0000415	0	
2.	0.4250	0.425	
3.	0.6806	0.679	10.85
4.	0.6853	0.679	
5.	0.4212	0.425	
6.	0.2527	0	

7. CONCLUSIONS

At the point when pillar is defamed into five components which is to be broke down in STAAD Pro and MAT Lab programming. The outcome (for example - Deflection) is same.

The Advantages of this venture are, for example, efficient and improved outcome.

A test part is additionally led (a crystal and Beam with support bars) to check the avoidance result with programming done.

Subsequently the final product of MAT Lab Programming, STAAD Pro and Experimental part are same.

REFERENCES

1.   S. S. Bhavikatti, "Limited Element Analysis", New Age Publishers, 2007.

2.   C. S. Krishnamoorthy, "FiniteElemenAnalysis: Theory and Programming" ,Tata McGraw-Hill, 1995

3.David Hutton, "Basics of Finite Element Analysis", Tata McGraw Hill Publishing Company Limited, New Delhi, 2005.

4.Bathe, K.J., "Limited Element Procedures in Engineering Analysis", Prentice Hall Inc., 1996.

5.Zienkiewicz, O.C. also, Taylor, R.L "The Finite Element Method", McGraw Hill, 1987

6.Chandrupatla, R.T. and Belegundu, A.D., "Introduction to Finite Elements in Engineering", Prentice Hall of India, 1997

7. Moaveni, S., "Limited Element Analysis Theory and Application with ANSYS", Prentice Hall Inc., 1999

8.   S. Md. Jalaludeen, "Limited Element Analysis Anuradha Publications 2008

9. S. Senthil., "Introduction of Finite Element Analysis", Lakshmi distribution 200