Deformation Analysis of a Triangular Mild Steel Plate Using CST as Finite Element

Mr. Amit Karkamkar
*(Mechanical Engineering Final year Student / National Institute Of Technology Warangal, India)

ABSTRACT: *This research work aims to analyse isotropic triangular mild steel plate constrained by boundary condition using Finite Element Analysis with CST as master element. Static deformation analysis of a mild steel triangular plate fixed at one end under point loads at the free end has been performed using Constant Strain Triangle (CST) as finite element. FEA analysis techniques are used to find the deformation of the plate. Problem Modelling and simulation of the plate is done in ANSYS and the results obtained by the finite element analysis using Constant Strain Triangle (CST) are compared. Good agreement has been found out in the deformation results by both the methods. Simulation results are critically studied and salient conclusions have been drawn.*

Keywords: *ANSYS, Constant Strain Triangle (CST), Deformation, FEA, Mild steel plate*

I. INTRODUCTION

Finite element analysis (FEA) is a powerful computational technique used for solving engineering problems having complex geometries that are subjected to general boundary conditions. In the FE analysis technique, the system is divided into discrete finite number of parts or elements by expressing the unknown field variable in terms of the interpolation functions within each element. The ease of computation depends on the linearity and the nature of the interpolation functions. This paper focuses on the use of the three node triangular element as the finite element. Geometry and displacement variables follow isoparametric representation . It is also known as Constant strain triangle(CST) because the localized strain in the element remains constant as it depends only on the nodal displacements. The shape functions for this elements allow ease of calculations. The paper first relates the shape functions to the displacement variables and element stiffness matrix for the CST element has been found out. Later the deformation in the plate is studied by finding out the nodal displacements in given loading and boundary condition. Accuracy of the results have been found out by comparing the results with ANSYS analysis.

II. DESCRIPTION

A triangular mild steel plate is considered in this paper for analysis. Mild steel is the most common form of steel used in industries because of its low cost and good mechanical properties. Mild steel is a general term for a range of low carbon steel (about 0.3%). They have good strength and can be bent, worked or can be welded into an variety of shapes for uses from vehicles to building materials. A typical CAD model for such a plate is shown in FIG .1 The model is generated in the designing software package PRO-E.

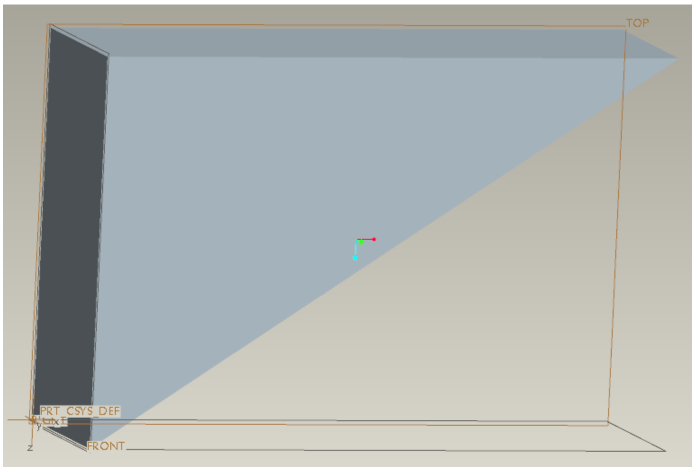

FIG. 1

For the deformation analysis a typical boundary condition is considered. One edge of the plate is fixed and point loads applied at the free end of the plate. The objective is to study the deformation in the plate under the influence of the loads that is the amount of deflection of the free end both in horizontal and vertical direction. For the analysis Finite element analysis method is utilized and Constant strain triangle is taken as the basic element type. For the purpose of analysis typical values of the point loads are assumed and the condition of plane stress condition is considered.

2.1) Constant Strain Triangle (CST) :

In the finite element method, the displacements at points inside the element need to be represented in terms of the nodal displacements of the elements. For the constant strain triangle, the shape functions or the interpolation functions are linear over the element. The independent shape functions are represented by ξ and η. CST will have three nodes at each of the vertices of the triangle and each having two degrees of freedom. Let us define the DOF in X positive X axis as q_{2i-1} and DOF in Y axis as q_{2i} . where, I is the node number. FIG. 2 shows the three nodes 1, 2 and 3. Thus q_1, q_2, q_3, q_4, q_5, q_6 represent the displacements at the three nodes.

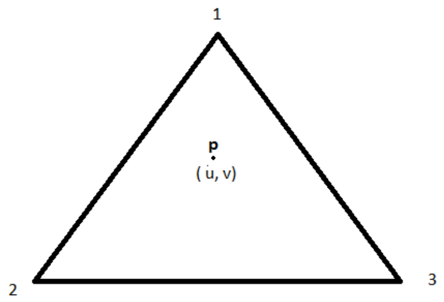

FIG. 2

The displacements inside the element are now written using the shape functions and the nodal values of the unknown displacement field.

$$u = (q_1 - q_5)\xi + (q_3 - q_5)\eta + q_5$$
$$v = (q_2 - q_6)\xi + (q_4 - q_6)\eta + q_6$$

For the triangular element, the coordinates x, y can also be represented in terms of nodal coordinates using the same shape functions. This is isoparametric representation. From reference [1], for the triangular element,

$$\zeta = Bq \quad \text{.......... (1)}$$

Where, ζ is the strain matrix and q is the displacement matrix. B is a (3 x 6) element strain - displacement matrix relating the three strains to the six nodal displacements and is given by

$$B = \frac{1}{\det J}\begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix} \quad \text{.......... (3)}$$

Where, $x_{ij} = x_i - x_j$ and $y_{ij} = y_i - y_j$

i and j represent the node numbers.

det J is the Jacobian of the transformation. Analysis shows that Value of det J is equal to twice the area of the triangular element [2].

det J = 2 x A_e .... (4)

Where A_e is the area of the triangular element.

Thus it can be seen from equation (1) that the strain in this triangular element remains constant throughout and that is why a lot of computational time is saved by using Constant strain triangle as model element when using Finite element method. This paper aims at verifying the computational accuracy of this method when compared to the ANSYS analysis method.

The element stiffness matrix k^e is given by

$$k^e = t_e A_e B^T DB$$

Where,

t_e = Thickness of the element

A_e = Area of the triangular element

D = Material property matrix

Value of D matrix can be taken depending on the plane stress or the plane strain condition. In this analysis plane stress condition is considered for the deformation of the mild steel plate under loading.

Now , by the finite element analysis method,

KQ = F ....(5)

where, K is the global stiffness matrix, Q is the displacement matrix, and F is the force vector.
Equation (3) is solved for the vector matrix Q which will give the values of displacement under the influence of the force vector F.

III. PROBLEM STATEMENT

FIG. 3 clearly elaborates the direction and the magnitude of the point loads acting on the plate . The plate is 20 mm in height and 30 mm in width and 10 mm in thickness. A point load of 100 N is acting in the negative Y direction at the free end of the plate and point load of 50 N in the positive X direction. The plate is fixed along the edge AB.

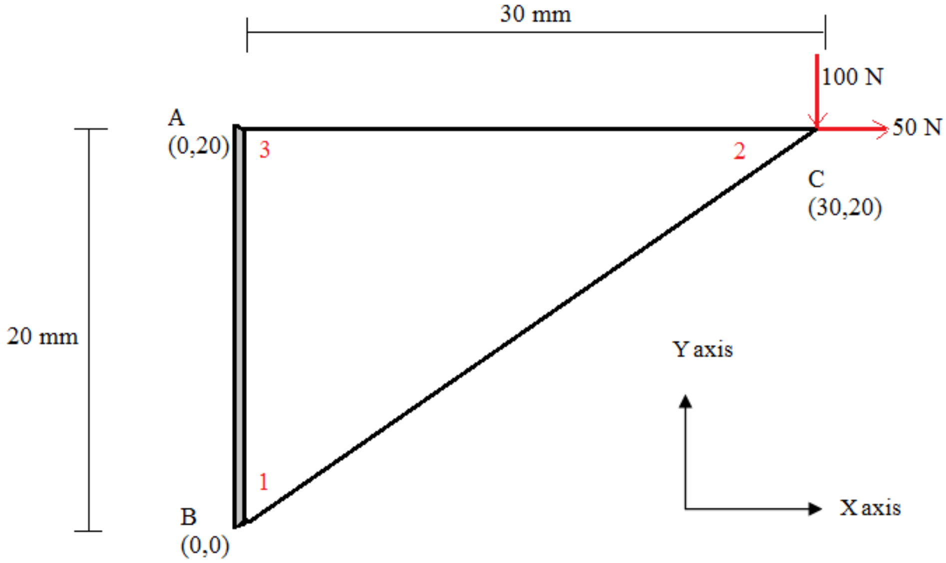

FIG. 3

Now this triangular plate is mapped to a isoparametric constant strain triangle. The problem is solved using FEA and one element. Thus the points 1, 2 and 3 represent the nodes of the constant strain triangular element. Coordinate system used indicating the positive directions is shown in the figure. Point B is considered as origin for the simplicity of analysis.

3.1) Material Properties of mild steel:

The material properties of mild steel are shown in the following TABLE 1.

TABLE 1

Material Used	Young's modulus (E)	Poisson's Ratio (ν)	Density (ρ)
Mild Steel	210 GPa	0.3	7850 kg/m^3

IV. CALCULATIONS

To study the deformation in the plate, under given force vectors and the dimensions of the plate , plane stress condition is assumed. Thus the value of matrix D is given as

$$D = \frac{E}{1-\nu^2}\begin{pmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{pmatrix} = \frac{210x10^3}{0.91}\begin{pmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & \frac{1-0.3}{2} \end{pmatrix} = 230769.23\begin{pmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{pmatrix}$$

Using eq. 4;

det J = 2 x A_e = 2 x $(\frac{1}{2}x20x30)$ = 600

Using eq. 3 , value of matrix B is found out.

$$B = \frac{1}{600}\begin{bmatrix} 0 & 0 & 20 & 0 & -20 & 0 \\ 0 & -30 & 0 & 0 & 0 & 30 \\ -30 & 0 & 0 & 20 & 30 & -20 \end{bmatrix}$$

$k^e = t_e A_e B^T DB$ = 10 x 300 x $B^T DB$

Using the program MATLAB above complex multiplication has been carried out and the value for the element stiffness matrix is

$$k^e = 1923.07\begin{bmatrix} 315 & 0 & 0 & -210 & -315 & 210 \\ 0 & 900 & -180 & 0 & 180 & -900 \\ 0 & -180 & 400 & 0 & -400 & 180 \\ -210 & 0 & 0 & 140 & 210 & -140 \\ -315 & 180 & -400 & 210 & 715 & -390 \\ 210 & -900 & 180 & -140 & -390 & 1040 \end{bmatrix}$$

In this case as shown in FIG 3; only one CST element is considered for analysis. Thus,

$k^e = k_g$, where k_g is the global stiffness matrix.

Global force vector will be given by, $F=\begin{Bmatrix} 0 \\ 0 \\ 50 \\ -100 \\ 0 \\ 0 \end{Bmatrix}$

Using equation (5);

$$1923.07\begin{bmatrix} 315 & 0 & 0 & -210 & -315 & 210 \\ 0 & 900 & -180 & 0 & 180 & -900 \\ 0 & -180 & 400 & 0 & -400 & 180 \\ -210 & 0 & 0 & 140 & 210 & -140 \\ -315 & 180 & -400 & 210 & 715 & -390 \\ 210 & -900 & 180 & -140 & -390 & 1040 \end{bmatrix}\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 50 \\ -100 \\ 0 \\ 0 \end{Bmatrix} \quad \ldots\ldots\ldots\ldots(6)$$

The edge AB of the mild steel plate is fixed as shown in FIG .3 Thus the DOFs q_1, q_2, q_5 and q_6 will be constrained to be zero. So now using the elimination method to solve the equation no. (6) striking off the rows and columns corresponding to the constrained Degrees of freedoms, we get,

$$1923.07\begin{bmatrix} 400 & 0 \\ 0 & 140 \end{bmatrix}\begin{Bmatrix} q_3 \\ q_4 \end{Bmatrix} = \begin{Bmatrix} 50 \\ -100 \end{Bmatrix}$$

Solving the above equation , we get,

q_3 = 6.5 x 10^{-5} mm ; q_4 = - 3.71 x 10^{-4} mm ……………………………………………………… . (7)

Let σ be the stress generated in the element.

σ = DBq , where q is the element nodal displacement vector.

$$\therefore \quad \sigma = \frac{230769.23}{600}\begin{pmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{pmatrix}\begin{bmatrix} 0 & 0 & 20 & 0 & -20 & 0 \\ 0 & -30 & 0 & 0 & 0 & 30 \\ -30 & 0 & 0 & 20 & 30 & -20 \end{bmatrix}\begin{Bmatrix} 0 \\ 0 \\ 6.5x10^{-5} \\ -3.71x10^{-4} \\ 0 \\ 0 \end{Bmatrix}$$

σ = $\{0.49, 0.15, -0.99\}^T$ MPa ……………………………………………………….. …………………………….. (8)

V. ANSYS Modeling

Commercially available software ANSYS 13.0 is used to model the problem and solve for deformation. ANSYS parametric design language (APDL) is utilized for this purpose. A model is generated in the work plane for the given dimension and thickness of the mild steel plate. After specifying the material properties and real constants as shown in TABLE 1, boundary condition is applied. That is both the nodes 1 and 3 are fixed thus constraining their displacement to zero. At the node 2, constant point load of 50 N in the positive X axis and 100 N in the negative Y axis direction is applied. The generated model is shown in FIG. 4. Deformation analysis finding the nodal displacements using the numeric method is done by the ANSYS postprocessor. The stress generated in the element is also found out. The displacement in the positive X direction of the node 2 is 0.65E-04 mm and in the negative Y direction 0.37E-03 mm. Using the command plot results- a deformed shape of the plate is generated by the ANSYS software. This is shown in the FIG 5

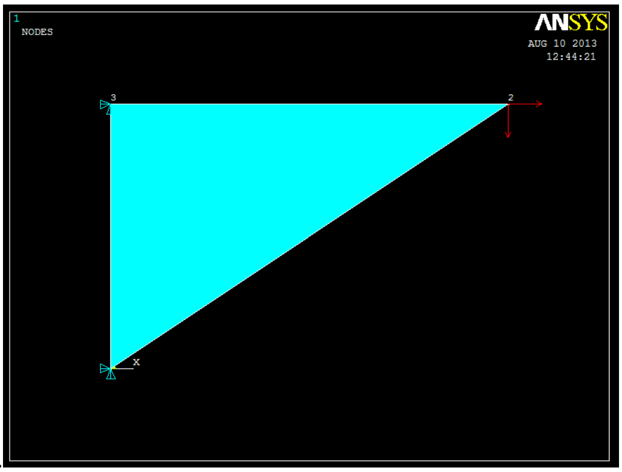

FIG. 4

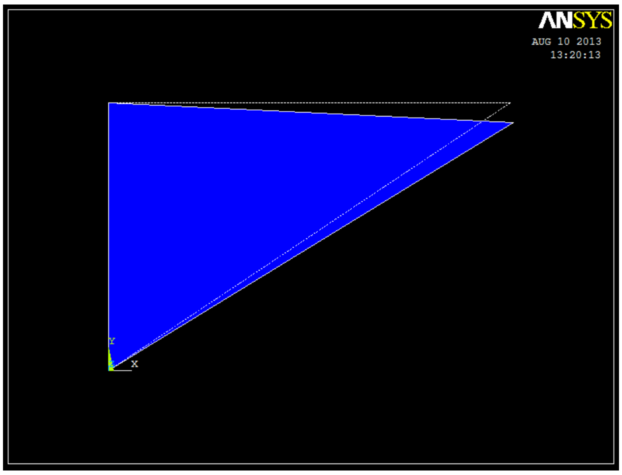

FIG 5

VI. Results & Discussions

From eq. (7) and (8) and the results obtained from ANSYS modeling, it can be conclusively averred that the results of the finite element analysis using Constant Strain Triangle and that of ANSYS show good agreement. The deformation of the plate subjected to given boundary condition and loading and the corresponding Von-Mises Stress generated were equal in both the methods.

VII. Conclusions

This paper mainly focused on finding the deformation of the triangular mild steel plate subjected to a boundary condition under the influence of point loads. The deformation analysis is first carried out by finite element method using the three node Constant Strain Triangle (CST) as a basic geometric shape. Later, for the same structure and load, boundary conditions, analysis has been performed using analysis software ANSYS (APDL). Finally, the results obtained from FEA and ANSYS are compared and they are closely converging. Thus use of Constant Strain Triangle as a master element for

FEA in complex stress and deformation analysis is proposed because of the following reasons:
1) The stress value in the basic modeling element is constant and depends on nodal displacements. This saves a lot of computational effort and time required for analysis. The shape functions are linear over the element. Isoparametric representation allows use of same shape functions for representing geometry and displacements inside the element. This approach lends simplicity of development and retains uniformity with other complex elements.
2) The results obtained by FEA using CST as modeling element are in good agreement with the solution.

REFERENCES

Books:
[1] A. D Belegundu , T.R Chandrupatla ,*Finite elements in Engineering* second edition (Prentice hall India).
[2] S. S Rao , *The Finite Element Method in engineering* fourth edition (Elsevier Science & Technology).
[3] E. Madenci , I. Guven , *The finite element method and applications in engineering using ANSYS* (Springer NY USA).
[4] Y. Kwon, *The Finite element method using MATLAB* (CRC press Washington DC).