International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 06 | June-2018 www.irjet.net p-ISSN: 2395-0072

Comparative Result of Displacement and Stress for Tapered Beam L/D=1.25 by FEM Program, ANSYS

Mrudul Burande[1],Dr. Tushar Shende[2]

[1]MTech Student, G. H. Raisoni Academy of Engg. & Tech, Nagpur India
[2]HOD of Civil Dept. G. H. Raisoni Academy of Engg & Tech, Nagpur, India

---

Abstract – *The analytical study of reinforced concrete simply supported deep beams subjected to two point loads was carried out using finite element method to study the behavior of deep beam by considering flexural stress variation at different section for various effective lengths to depth ratio and compared with Euler Bernoulli Theory. The effective span to depth ratios of the beam. There are many software packages available for the analysis of deep beams but we generally use SAP, ANSYS software. In finite element as triangular element, elements of serendipity family. In ANSYS it is modeled using Solid82 for 2D and Solid186 3D analysis. In this project, result of deflection, flexural stress shear of cantilever prismatic deep beams obtained using FEM program for isoparametric elements and ANSYS 2D and 3D analysis are compared*

Key Words: Deep beams, aspect ratio, convergence study, flexural stress, discritisation

1. INTRODUCTION

Deep beam are defined as the members loaded on the face and supported on the opposite face so that compression struts can develop between the loads and the supports. In IS-456(2000) clause 29, a simply supported beam is classified as deep when the ratio of its effective span L to overall depth D is less than 2. Continuous beams are considered as deep when the ratio L/D is less than 2.5. The effective span is defined as the center to center distance between the support or 1.15 times the clear span whichever is less.

There are many software packages available for the analysis of deep beams but we generally use SAP, ANSYS software. In available such as triangular elements of serendipity family. In ANSYS it is modelled using Solid82 for 2D and Solid 186 for 3D analysis.

In this project, results of deflection, flexural stress and shear stress of cantilever prismatic deep beams obtained using FEM program for Iso-parametric elements, and ANSYS 2D and 3D analysis are compared. A parametric study of deep rectangular beams for point load has been carried out.

2. FINITE ELEMENT METHOD

The finite element method (FEM) is a numerical analysis technique for obtaining approximate solution to a wide variety of problems in engineering and science. The FEM is ideally suited for problem involving complicated geometries, loading and boundary condition, for which analytical mathematical solution are not possible . This method is similar to matrix displacement method of structural analysis in which a skeletal structure is made up of one-dimensional members connected at joints (bars/truss, beams/frame type)

The basic concept of finite element is based on dividing the complete structure/domain/region into smaller pieces or sub-region known as element are interconnected at finite number of joints or nodes. The elements can be of any size depending on the problem. The dividing or splitting of the structure into number of element is called discretization.

2.1 ANALYTICAL RESULTS

The common type of isoparametric elements used for 2D elastic analysis re linear, parabolic (quadratic) and cubic elements with 4,8 and 12 nodes.

This we have

Table 1: Abscissa and weight coefficient

N	ξ_i	w_i
(1)	(2)	(3)
1	0.00000	2.00000
2	± 0.57735	1.00000
3	± 0.77459 0.00000	0.55555 0.88888
4	± 0.86113 ± 0.33998	0.34785 0.65214
5	± 0.90617 ± 0.53846 0.00000	0.23692 0.47862 0.56886

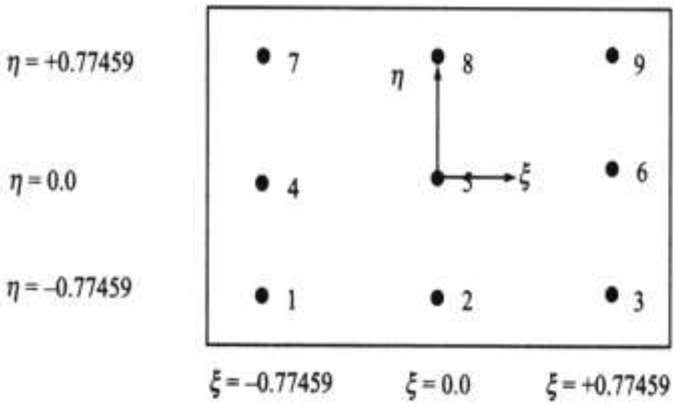


Figure no. 1

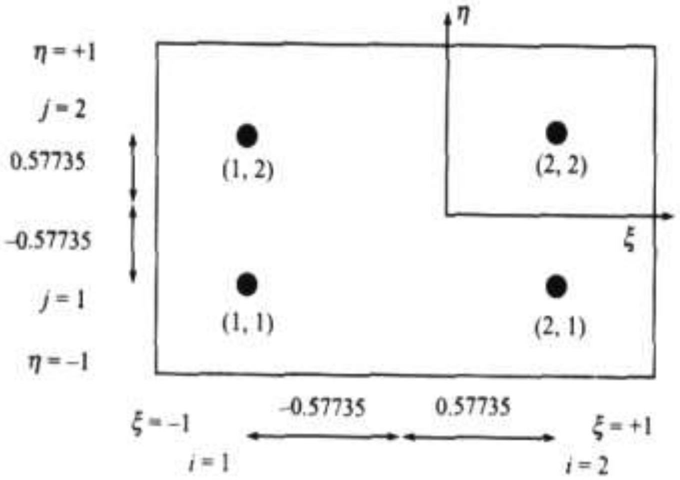


Figure no. 2

3 ANSYS:

ANSYS software is powerful and flexible general-purpose finite element analysis and computational fluid dynamics package used foe civil engineering. Mechanical engineering, electrical engineering, physic and chemistry simulation/ Simulation tools including ANSYS Mechanical APDL, ANSYS CFX and ANSYS FLUENT, can also solve mechanical problems, static/dynamic structural analysis, heat transfer and fluid problem as well as acoustic and electromagnetic problems.

ANSYS is a general purpose finite element modeling package foe numerically solving a wide variety of problems in engineering, in particular:

1. Static/dynamic structural analysis (both linear and non-linear)
2. Fluid analysis- Laminar and turbulent flow
3. Acoustic analysis
4. Electromagnetic analysis
5. Thermal analysis- conduction, convection, radiation
6. Transient thermal analysis
7. Spectrum analysis
8. Bucking Analysis
9. Harmonic analysis
10. Model analysis – Vibration Characteristics

PLANE82 is a higher order version of the 2-D , four-node element (PLANE42). It provides more accurate results for mixed (Quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The 8-nodes element have compatible displacement shapes. And are well suited to model curved boundaries.

3.1 PLANE82 INPUT SUMMARY

Nodes
I,J.K,L,M,N,O,P
Degree of Freedom
UX,UY
Real Consultants

Material Properties
EX, EY, EZ, PRXY, PRXY,PRYZ,(or NUXY, NUYZ, NUXZ),
ALPX, ALPY, ALPY (or CTEX, CTEY, CTEZ or THSX, THSY,THSZ), DENS, GXY, DAMP
Surface Loads
Pressures:
faces 1 (J-I), fce 2 (K-J), face 3(I-K), face 4 (I-L)
Body Loads
Temperatures-
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)
T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

3.2. PLANE82 OUTPUT DATA

The solution output associated with the element is in two forms:-

- Nodal displacement included in the overall nodal solution
- Additional element output

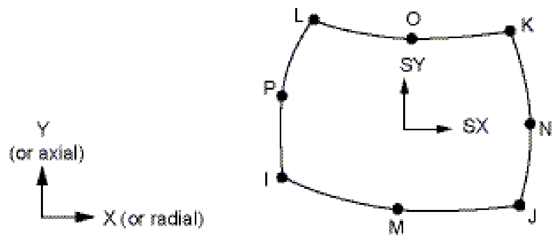


Figure no. 3

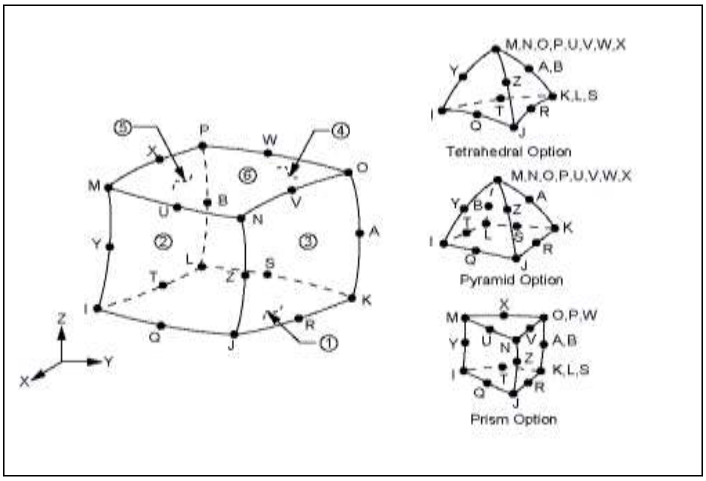
PLANE82 Stress output.

The element stress directions are parallel to the element coordinate system. Surface stresses are available on any face. Surface stresses on face IJ, for example, are defined parallel and perpendicular to the IJ line and along the Z axis for a plane analysis or in the hoop direction for an axisymmetric analysis.

3.3 SOLID186 Element Description

OLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.
SOLID186 is available in two forms:

- Homogenous Structural Solid (KEYOPT(3) = 0, the default) -- See "SOLID186 Homogenous Structural Solid Element Description ".
- Layered Structural Solid (KEYOPT(3) = 1) -- See "SOLID186 Layered Structural Solid Element Description".



3.4 SOLID186 Homogenous Structural Solid Input Summary

Nodes

I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z, A, B

Degrees of Freedom

UX, UY, UZ

Real Constants

None

Material Properties

EX, EY, EZ, ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX,THSY, THSZ),

PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ),

DENS, GXY, GYZ, GXZ, DAMP

Surface Loads

Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),

face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P)

Body Loads

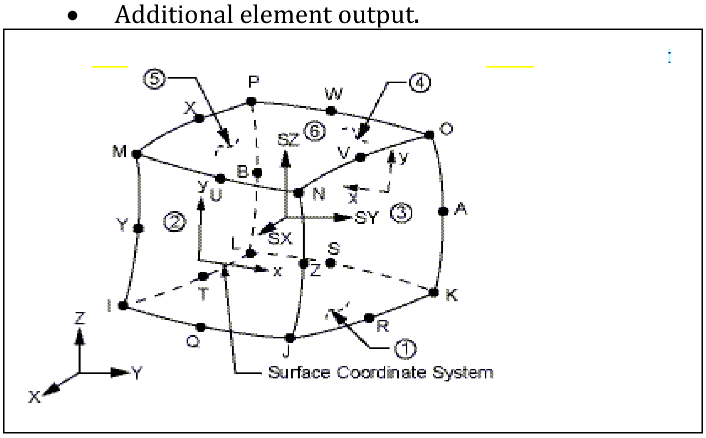
Temperatures --

T(I), T(J),T(K), T(L), T(M), T(N), T(O), T(P), T(Q), T(R), T(S), T(T), T(U), T(V), T(W), T(X), T(Y), T(Z), T(A), T(B)

3.5 SOLID186 Homogenous Structural Solid Output Data

The solution output associated with the element is in two forms:

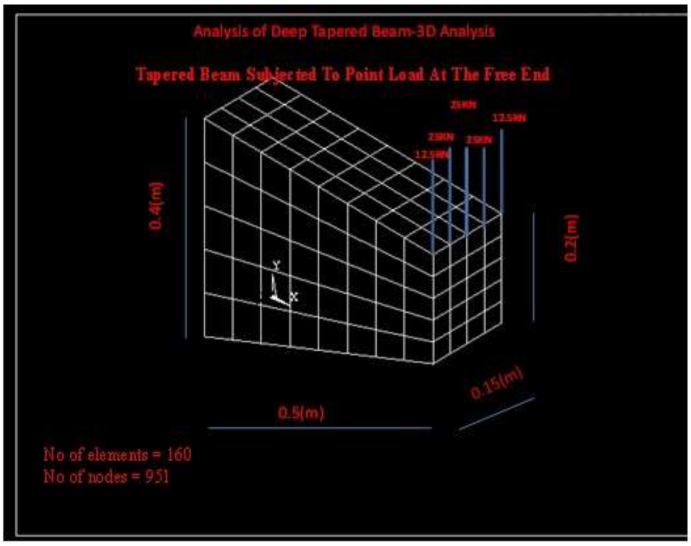
- Nodal displacements included in the overall nodal solution
- Additional element output.



4 RESULT

L/D	METHODS	DEFLECT-ION (MM)	σx (KN/ mm^2) (TENSILE)	σx (KN/ mm^2) (COMP)	τxz (KN/ mm^2)
1	2	3	4	5	6
1.25	FEM PROGRAM	0.5973	13.986	8.996	3.370
	ANSYS 2D	0.6372	14.212	9.426	3.606
	ANSYS 3D	0.52254	14.801	9.617	3.245

Analysis of Cantilever Tapered Deep Beam-3D Analysis



4 CONCLUSIONS

The beam of different L/D ratios are analyzed by FEM program and ANSYS 2D. Results obtained are compared with each other. From the results conclusions are made and are as follows.

Deflection values given by EBT are much less than the values given by FEM program and ANSYS in case of rectangular deep beam

- From the results given by FEM program, ANSYS it is found that the flexural stress distribution in deep beam is not linear as in case of slender beams, but EBT gives the flexural stress distribution as linear for deep beam.
- EBT gives the maximum shear stress at the center line of the beam, but in deep beams the maximum shear stress is below the center line of the beam. It is observed that the neutral axis moves downward
- It is observed that though there is a variation in the results of deflection obtained from FEM Program and ANSYS, the agreement is very close.

REFERENCES

1.Wai Chi Mun, AhemadRivai, OmarBapokutty"Effects of elements on linear elastic stress analysis: A Finite Element Approach". IJRET: International Journal of Reaserch in Engineering and TechnologyVolume:2 Issue:10-Oct 2013

2. Sudarshan D. and Patil S.S. (2013) "Analysis and Design of R. C. Deep Beams Using Code Provisions of Different Countries and Their Comparison" , , International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249-8958, Volume-2, Issue-

3. Godbole P. N. (2013) "Introduction to Finite Element Method", Page No. 23-184 I. K. Publishing House Pvt. Ltd. New Delhi Page 5. NiranjanB.R. and Patil S. S. (2012), "Analysis of Deep Beam By Finite Element Method" , IJMER VOL. 2, Issue 6.

5. Yuwaraj M. Ghugal And Rajneesh Sharma (2011) "A refined shear deformation theory for flexure of thick beams", Latin AmericanJournal of Solids and Strures Vo1. 8183-195.

6. Mahdy and O. Sh. Farhan, A. A. A1- Azzawi 1, A. H. (2010) "Finite element analysis of deep beams on nonlinear elastic foundations", Journal of the Serbian Society for Computational Mechanics, Vo1. 4, No. 2, pp. 13-42 8