FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE RECTANGULAR AND TRAPEZOIDAL SLAB

Sanchita U Sawant[1], Nisha P Naik[1]

1. Goa College of Engineering, Farmagudi, Goa, India

ABSTRACT. In this paper, a finite element analysis of a rectangular and trapezoidal slab is carried out. Finite element method is a method for diving up a very complicated problem into small elements that can be solved in relation to each other. The study is conducted for a better understanding of the structural behaviour of a slab of a different shape. The finite element analysis was performed using software ANSYS under static load. In ANSYS software, the SOLID65 material is used to define concrete while link180 to define steel. The correlation studies between manual and analytical result were conducted with an objective to establish the validity of the proposed model. For validation purpose a simply supported discontinuous rectangular slab was modelled, the results obtained from analysis were verified using manual calculation. The limit State of serviceability conditions for deflection satisfies the limiting values obtained from software results. A similar analysis is performed on a trapezoidal slab, deflection and stresses are found out. The parameters like Von Mises stress, Maximum Principal Stresses, Total deformations, Maximum Shear stress were analysed in this study which cannot be obtained from the experiment. The main aim of this study is to develop a safe and economical finite element model which is very cost effective that can form an alternative to full-scale laboratory testing.

Keywords: Finite element analysis, Deflection, Rectangular slab, Stresses, Trapezoidal slab

Sanchita U Sawant is a M.E. Structural student, Department of Civil Engineering. Goa College of Engineering, Farmagudi, Ponda-Goa, 403401.
Telephone: 9403371528/7775090997 Email: sawantsanchita004@gmail.com

Dr Nisha P Naik is a Associate Professor of Civil Engineering, Department of Civil engineering, Goa College of Engineering, Farmagudi, Ponda –Goa, 403401. Telephone: 9552482752 Email: nisha@gec.ac.in.

INTRODUCTION

The finite element analysis (FEM) is a numerical method which uses discretization i.e. divides the complex structure into small domain called element and element consists of nodes. This numerical method formulates the differential equations into a system of algebraic equations. It gives approximate values of the desired element to represent its behaviour whose exact solution cannot be obtained in many situations. The accuracy of finite element analysis can be improved by increasing the number of elements that implicate more nodes to define unknown functions. Civil engineers use this method for analysis of beams, space frames, plates, shells, folded plates, foundations, rock mechanics problems. Clough was the first to use the term ‘finite element’.

The slab is a flat horizontal surface supported by beam and column. In this investigation rectangle slab and the trapezoidal slab is used for finite element analysis. The main aim of this paper is to study the structural behaviour of slab and creation of safe and economical structure. No guidelines are given in codes for deflection control and stress in the trapezoidal slab.

Mohamed et al. [1] analysed reinforced concrete deep beams with web openings using finite element software ‘ABAQUS’. A parametric study was carried out by the varying dimension of web opening. A simply supported beam with 3 and 4-point loading with and without web opening was analysed. The model has been validated using deep beam studied by Hong et al for evaluation of shear stress. The crack pattern was studied at failure modes. The comparative study of experiment and model shows the satisfactory result.

Hassan et al. [2] carried out finite element analysis of reinforced concrete beams with opening strengthened using FRP. Using ANSYS V12 fifty-seven beams were modelled under static load and compared with experimental beams done by Ibrahim who took into the considerate effect of UDL. A solid element, SOLID65 and SOLID45 which has eight nodes with three transitional degrees of freedom at each node were modelled for concrete and steel plates at support respectively. 3D spar link 8 element technique was followed for modelling. This solid element was capable of plastic deformation, cracking and crushing. Mesh of size 10×10×10mm was used. The failure occurred in beam due to tensile stresses near openings when analysis results were compared, it shows that strengthening helped the beam in empowering amount of stiffness which was lost due to openings.

Lantsoght et al. [3] analysed the distribution of peak shear stress infinite element models of reinforced concrete slabs. The shear stress was determined using spreadsheet base method and linear finite element model. To determine the peak linear distribution over width the linear finite element model was compared to an experiment. Levels of assessment were used for analysis of 600 RC slab badges in the Netherlands to determine critical shear. Results obtained from comparing experiment and FEM shows that distribution width 4di should be used. In the case study, it was found out that Level of Assessment works.

Kataoka et al. [4] analysed Nonlinear FE analysis of slab beam-column connection in precast concrete structures. Finite element model was verified using the experimental result. FE 3D nonlinear model was constructed using software Midas FX+ and also Diana software was used for FEA. For concrete fixed crack model was used and for reinforcement, the metal model was used. The structural interface was used for analysis. For mesh two types of finite elements were used, plane stress for concrete while interfacing element for joints between column and Beams. The numerical model showed displacement, strains and stress satisfactorily as compared to the experimental model. It can be concluded that 3D nonlinear finite element model offers full-scale alternate for laboratory testing. Laboratory testing is time-consuming and requires a high cost.

RESEARCH STUDY

Design of the Model

The finite element analysis was carried out on the rectangular and trapezoidal slab. The simply supported rectangular slab of dimension 3.6 X 2.4m, discontinuous at all edges were considered as shown in figure 1. The design of slab is done manually as per guidelines of IS 456: 2000. The four beams of dimensions 230 X 400mm were also designed. The safety against vertical deflection, shear stress and failure mode were tested. The grade of concrete used is M20 and Fe 415 steel was used. Similarly, trapezoidal slab discontinuous on all edges were analysed, dimensions are shown in figure 2.

Figure 1 Rectangular slab

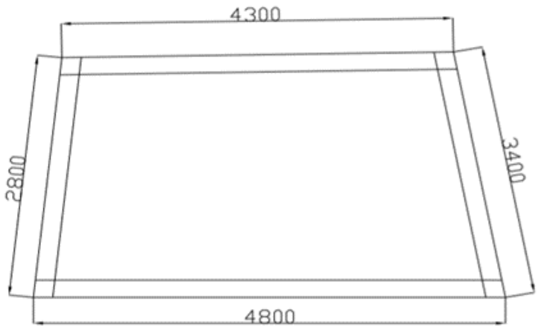

Figure 2 Trapezoidal slab

Finite Element Analysis of Model

The slab was modelled in ANSYS software. The properties were assigned to concrete and steel. The density of concrete is taken as 2400kg/m^3. The Young's modulus and Poisson's ratio is taken as 3e5Pa and 0.2 respectively. The density of steel is taken as 7850kg/m^3. The Young's Modulus, Poisson's Ratio and Tensile Yield strength of steel is taken as 2e11Pa, 0.3 and 4.15e8Pa respectively. The models of slab in ANSYS are shown in figure 3 and 4.

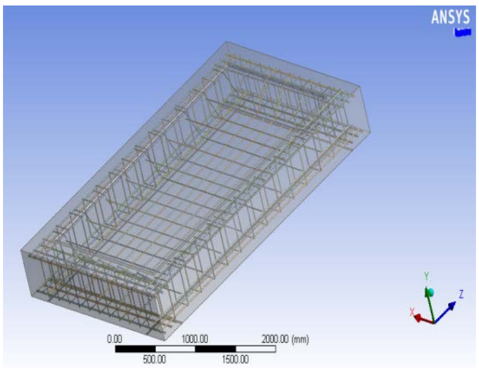

Figure 3 Geometry of rectangular slab

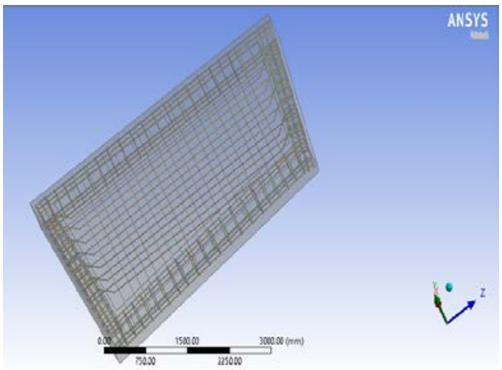

Figure 4 Geometry of trapezoidal slab

For slab models, the meshing of size 100mm was done shown in figure 5, 6, 7 & 8. For concrete and steel to behave as one body the No separation type of contact is assigned. The Nodal displacements were given. The UDL of magnitude 5N/mm2 was applied on slabs. The support conditions were given at the ends and analysis were done.

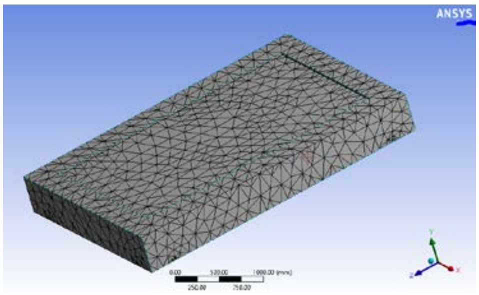

Figure 5 Mesh for concrete

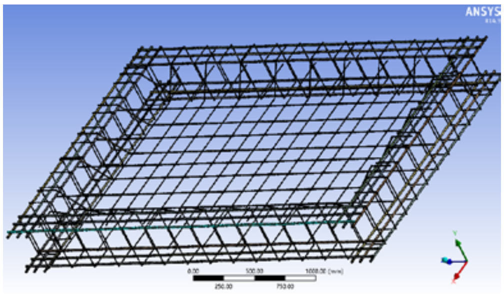

Figure 6 Mesh for reinforcement

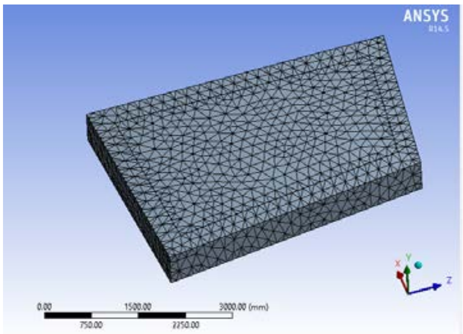

Figure 7 Mesh for concrete

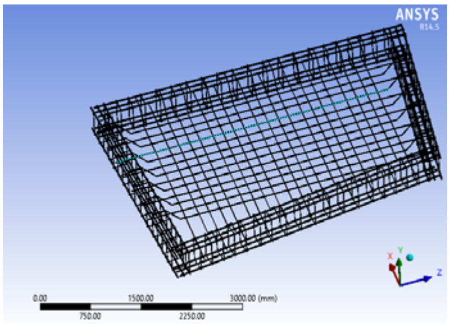

Figure 8 Mesh for reinforcement

RESULTS AND DISCUSSION

The permissible deflection of the slab which is allowed is 9.6mm, the deflection value obtained from ANSYS analysis is 3.67mm shown in figure 9. Therefore, for modelled slab deflection control is satisfied. The Maximum (L/d) ratio allowed is 30 while actual (L/D) ratio getting is 18.46. Hence the model is safe against deflection. The (L/D) ratio of the trapezoidal slab is 28.8. The maximum deflection obtained from ANSYS is 0.288mm shown in figure 11. The stresses in slab are shown in figure 10 & 12.

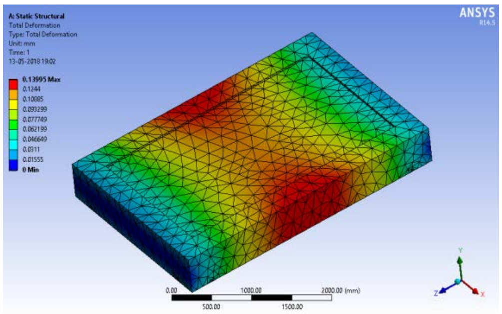

Figure 9 Deformation in rectangular slab

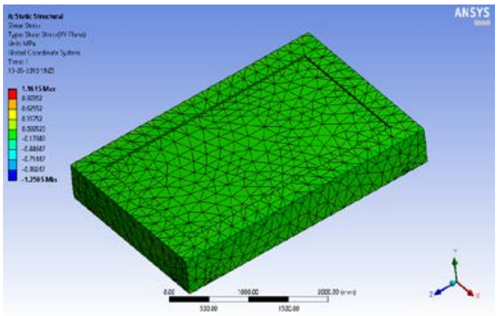

Figure 10 Stress in a rectangular slab

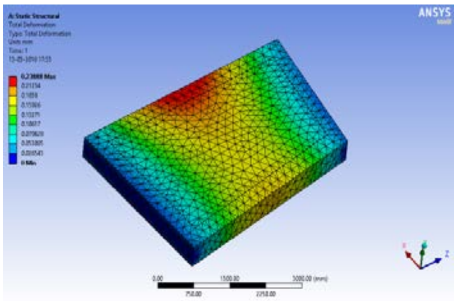

Figure 11 Deformation in trapezoidal slab

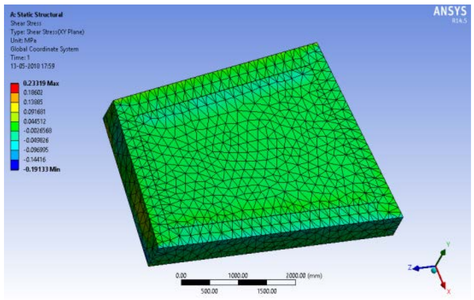

Figure 12 Stress in the trapezoidal slab

CONCLUSION

The main aim of this paper is to develop safe and reliable finite element model of the reinforced concrete element. This paper deals with finite element analysis of rectangular and trapezoidal slab. From above results, it can be concluded that this method can be adapted to develop finite element model and analysis of different type of slabs with a different shape for which no specific codes are available. The accuracy of results obtained can be increased by using more refined mesh. This analysis gives values of different parameters like maximum shear stresses, maximum equivalent stresses, maximum deflection, maximum principal stress, strain energy. This method is cheap and reliable compared to the experimental method of constructing the model.

REFERENCES

1. ANSYS ® Workbench ™ 2.0 Framework, Version 14.5.0.
2. MOHAMED, R.A., SHOUKRY M.S., AND SAEED J.M. Prediction of the behaviour of RC deep beams with web openings using the finite element method. Journal Alexendria Engineering, 2014. 53.329-339.
3. HASSAN N.Z., SHERIF A.G. AND ZAMARANRY A. H. Finite element analysis of reinforced concrete beams with opening strengthened using FRP. Ain Shams Engineering Journal. 2015.
4. LANTSOGHT E.O.L., BOER A.D., VEEN C.V. D. Distribution of peak shear stress infinite element models of reinforced concrete slabs. Engineering structures, 2017. 148 571-583.
5. KATAOKA M.N., FERREIRA M.A., DEBS A. L. H.D.C.E. Nonlinear FE analysis of slab beam-column connection in precast concrete structures.” Engineering structures 2017, 143 306-315.
6. SHAH V. L., KARVE S. R., Illustrated Design of Reinforced Concrete Buildings, Sixth Edition, Structures Publication, 2010.
7. BARI M. W., RAHMAN M. M., ALI M. M. Y., AND AWALL M. R. Finite Element Analysis of Slab and a Comparative Study with Others Analytical Solution. 7th ICCT 2004, M (24), pp265-272.
8. SHENBAGAM. N. K., PREETHA. V. AND JAYARAMAN A. Finite Element Analysis of Reinforced Concrete Beams. IJIRAE, 2014,Vol 1, No 8.