Numerical Analysis of Laterally Loaded RC Stiffened-Plate Using ANSYS

Ravpreet Kaur[1], Manpreet Kaur[2]

[1]Research Scholar, Department of Civil Engineering, Guru Nanak Dev Engineering College, Ludhiana, Punjab, India
[2]Assistant Professor, Department of Civil Engineering, PAU Ludhiana, Punjab, India

***Abstract*—** *The most widely used structural element in civil engineering structures are reinforced concrete plates.. In the available literature, numbers of design guidelines are available for proportioning the plate systems and mostly, these guidelines have been suggested on the basis of the results of an extensive series of tests and the well established performance record of various plate systems. However, its behavior is not fully understood. In the present paper, an attempt has been made to study the effect of beam stiffness on the moment field and crack pattern appearing in the plate under a uniform lateral load. To validate the analytical results, these are compared with the experimental results available in the literature and are found to be in good agreement.*

***Keywords*— Numerical analysis, ansys, laterally loaded, stiffened.**

I. INTRODUCTION

A plate is a part of a large structure surrounded by adjacent girders and other plates, and its boundaries are typically supported with strong longitudinal and transverse girders. The applications of the plates in the structural system are usually restricted by the limited shear strength and hogging moment-capacity at the column supports. Usually, very thick plate sections are required in practice to provide an adequate strength against these force-resultants and, simultaneously, to satisfy the serviceability requirements of the applicable design code. However, the thickness of the plate system can be reduced by selecting the minimum possible thickness of the plate required to provide an adequate moment-capacity against the given load and by providing some internal flexural member; referred to as stiffening beams in the plate to increase the stiffness and the shear capacity of the plate system.

Plates stiffened by internal beam(s) along its one or two orthogonal directions are an efficient and economical system readily constructible in most of common materials. It can be built as a monolithic unit or as a composite system comprising a plate cast in concrete and the beams constructed as fabricated sections in steel. These are used almost in every type of structural system to build and/or enclose the space along with some other structural elements such as walls and columns. The current state-of-art available for the design of reinforced concrete slab does not satisfactorily address the problem of proportioning the RC stiffened plates. Little research work has been reported on reinforced concrete stiffened-plates (Singh 2012; Sapountzakis 2008; Singh et.al. 2010). However, a good number of research publications are available for steel stiffened-plates (Kumar et.al 2009; Brubak and Hellesland 2008; Castro et.al 2007; Kukreti and Cheraghi 1993; Sapountzakis and Katsikadelis 2002; Schubal et.al 1993; Manolakos and Mamalis 1988; Bedair 1997). In a decade 1960-80, most of the work (Sozen and Siess 1963; Park 1968; Gamble et al. 1969; Park 1971; Gamble 1972; Sozen et al. 1963) was carried out on the single and multi-panel slabs proportioned using some empirical methods and various researchers tried to check the performance of RC slabs at service and collapse load.

In this paper, results from a study carried out on RC stiffened plates are presented. Results of RC stiffened plates obtained from finite element based software ANSYS are compared and validated with experimental results available in published literature.

II. NUMERICAL MODELLING

Finite element based software was used to determine the non-linear response of RC stiffened plate. The plate has been assumed to be supported over the simple non-yielding edges on its outer four boundaries and some internal flexural member-called as stiffening-beams-was used to divide it into panels of equal lengths. Two-panel RC plates of size 2350 x 4000 x 75mm and three-panel of size 2350 x 5000 x 75mm were used for supporting uniform lateral load applied over the top face of the plates. A stiffening-beam size of 200x150mm and 200x235mm were used to check the influence of beam stiffness on the moment field. A concrete mix of M20 grade and reinforcing steel of fy415 grade was used in the plates. The rebars were placed with a bottom clear cover of 15 mm and a side clear cover of 25 mm.

Finite element analysis provides a solution to problems that would otherwise be difficult to obtain experimentally. Moreover, crack pattern, deflection profile at short term as well as long term can very easily be obtained using this method. The numerical analysis were performed with a software ANSYS. This software is a suite of powerful engineering simulation programs, based on finite element method, which can solve problems ranging from relatively simpler linear analyses to the most challenging non-linear simulations. The analysis of a structure with ANSYS is performed in three stages

a) *Pre-processing*–It is used to define and for creating a model, applying load and boundary constraints to it. Various steps involved in this stage are summarized below:

- Chose element type, real constants, and material models of the concrete model.
- Create the model.

b) *Analysis solver*–this part of the software mesh the model, allows user to refine the mesh, if required and solves the model. Various steps involved in this stage are summarized below:

- Model was meshed by mapped mesh.
- Applying boundary conditions.
- Solve.

c) *Post-processing* of results like deformations, preparing contours for stress and displacement field. It also provides various visualization tools for interpretation of results.

2.1 Pre-processing

SOLID65 element was chosen to represent the concrete in the stiffened-plate model. This element has eight nodes with three degrees of freedom at each node – translations in the nodal x-, y-, and z-directions. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. A schematic of the element is shown in Fig.1

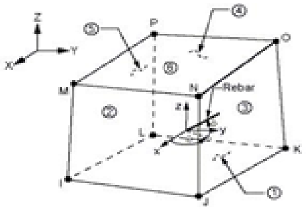

Fig.1:Solid65 Element

A Link8 element was used to model the steel reinforcement in the stiffened-plate. This element is a 3D spar element and it has two nodes with three degrees of freedom–translations in the nodal x-, y-and z-directions, being also capable of plastic deformation. This element is shown in Fig.2.

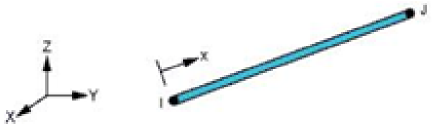

Fig.2:Link8 element.

Individual elements contain different real constants. Real Constant Set 1 is used for the Solid65element.Therefore; a value of zero was entered for all real constants. Real Constant Sets 2, 3 and 4 are defined for the Link8 element. Values for cross-sectional area and initial strain were entered. For each model *viz:* 2PSF, 2PSR, 2PNS, 3PSF, 3PSR, 2PNS, different real constant sets were defined as per the reinforcement detailing of samples of benchmark slabs and these are given in Tables 1 to 6.

TABLE.1: Real constant sets for 2PSF

Set No.	Element Type	Diameter of Bars, mm	Area, m^2
1	Solid65	0	All values given zero
2	Link8	10	0.0000785
3	Link8	16	0.00020096
4	Link8	12	0.00011304

TABLE.2: Real constant sets for 2PSR

Set No.	Element Type	Diameter of Bars, mm	Area, m^2
1	Solid65	0	All values given zero
2	Link8	8	0.00005024
3	Link8	20	0.000314
4	Link8	12	0.00011304

TABLE.3: Real constant set for 2PNS

Set No.	Element Type	Diameter of Bars, mm	Area, m^2
1	Solid65	0	All values given zero
2	Link8	8	0.00005024
3	Link8	16	0.00020096
4	Link8	12	0.00011304

TABLE.4: Real constant set for 3PSF

Set No.	Element Type	Diameter of Bars, mm	Area, m^2
1	Solid65	0	All values

			given zero
2	Link8	10	0.0000785
3	Link8	16	0.00020096
4	Link8	12	0.00011304

TABLE.5: Real constant set for 3PSR

Set No.	Element Type	Diameter of Bars, mm	Area, m^2
1	Solid65	0	All values given zero
2	Link8	6	0.00002826
3	Link8	20	0.000314
4	Link8	12	0.00011304

TABLE.6: Real constant set for 3PNS

Set No.	Element Type	Diameter of Bars, mm	Area, m^2
1	Solid65	0	All values given zero
2	Link8	6	0.00002826
3	Link8	16	0.00020096
4	Link8	12	0.00011304

The meshed models of these plates are shown in Figs. 3 to 8. A value of zero was entered for the initial strain because there is no initial stress in the reinforcement. The material properties for concrete and steel are defined through two material models

viz: material 1 for concrete and material 2 for steel under the linear isotropic material definition. The details of these material models are given below:

Material 1: M20 concrete.

1. Modulus of elasticity, E_c = 22,360 MPa
2. Poisson's ratio, ν = 0.12
3. Density of Concrete=24 KN/m^3

The constitutive relationship of concrete is shown in Fig. 3.

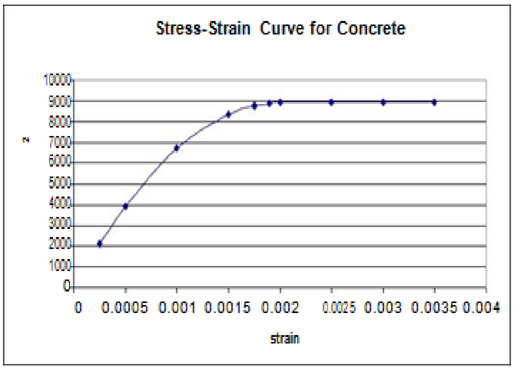

Fig.3: Stress Strain Curve of Concrete

Material 2: steel

Modulus of elasticity, Es = 2e+5 MPa

Poisson's ratio, ν = 0.33.

For Solid65 element, the mesh was set up such that square or rectangular elements were created. The volume mapped command was used to mesh the model so generated. This properly sets the width and length of elements in the rebars to be consistent with the elements and nodes in the concrete portions of the model. The meshed model was reinforced using line elements so that the nodes of the line elements come exactly over the node of the solid elements which are later merged so that both rebar elements and the concrete elements share the same nodes.

Displacement boundary conditions are needed to constrain the model to get a unique solution. The simple outer boundaries of the slab were simulated by keeping moment as zero along with vertical displacement. The symmetry boundary conditions were set first. Nodes

defining a vertical plane through the beam cross-section centroid define a plane of symmetry. To model the symmetry, nodes on this plane must be constrained in the perpendicular direction. Therefore the surfaces at the outer area are given displacement along y-direction, y; UY= 0.The outer areas given constraint in the UY- directions, applied as constant values of 0. Uniform force acting over the entire top face of the plate is applied. It increased in increments to collapse and the displacement field and the corresponding crack pattern was observed from the post processing.

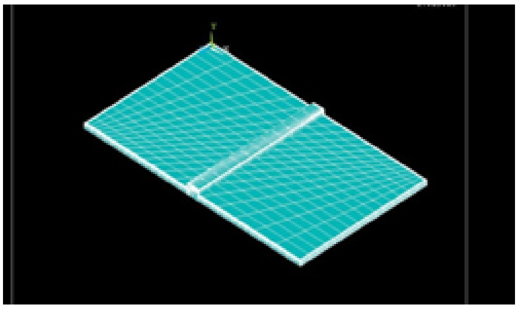

Fig.3: Meshed Model of 2PSF plate

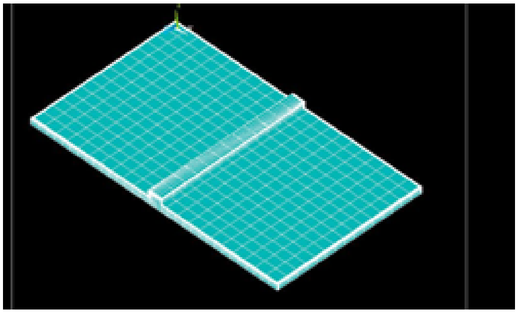

Fig.4: Meshed Model of 2PSR plate

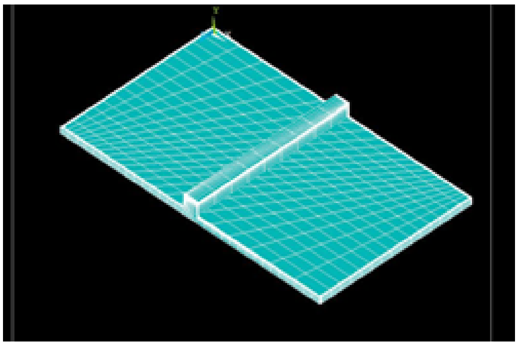

Fig. 5: Meshed Model of 2PNS plate

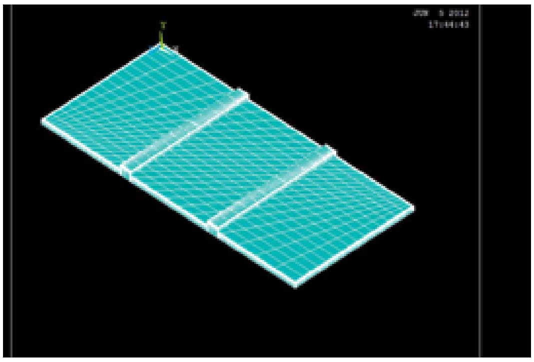

Fig. 6: Meshed Model of 3PSF plate

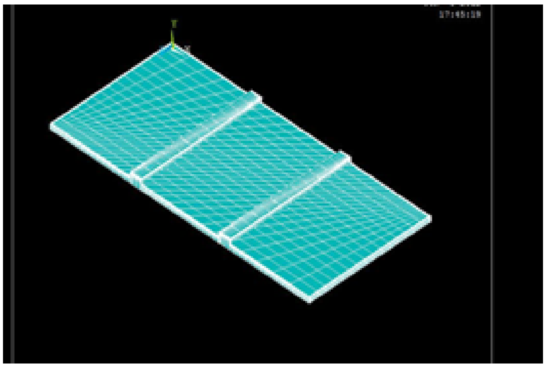

Fig. 7: Meshed Model of 3PSR plate

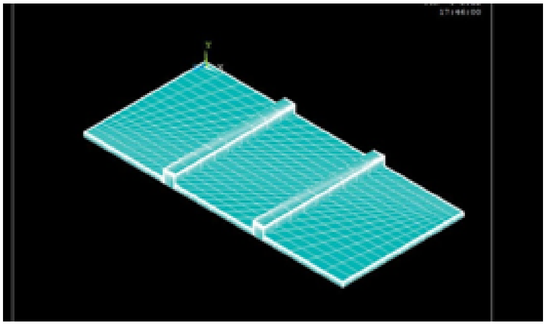

Fig. 8: Meshed Model of 3PNS plate

III. RESULTS AND DISCUSSIONS

RC rectangular stiffened-plate models shown in Fig 3 to 8 were analyzed under the static incremental loads using ANSYS software. The load-vertical displacement plots, load-slip plot of these plate models are shown in Fig. 9 to 26 along with their comparison with the experimental results. It must be noted that it is very difficult to observe and measure the slip along the plate and stiffening beams of the plate system. The crack pattern at 20kN/m^2 load level was shown in Fig. 27 to 30. This pattern was observed both at short term as well as long term loading. Crack widths in all plate samples are observed to be well within the limits prescribed by design guidelines. However, vertical deflection in plates 2PSF, 2PSR and 3PSF, 3PSR is found to exceed the allowable limits at long term loading but this value is well within safe limits for 2PNS and 3PNS plates. This trend is in conformity with the experimental results reported in [1]. One of the plus point of this study is that analyst is able to obtain the load-slip plots of all plate models. This shows the amount of horizontal displacement that occurs along the interface of plate and beams. These figures indicate that magnitude of slip reduces with the increase of beam stiffness. In case of beam with *span-depth* ratio more than 10, it increases elastically up to the design load; thereafter plate system shows a sudden increase of interfacial displacements [see Fig's 11, 14, 20, and 23]. At collapse, the value of the slip obtained from ANSYS is in good agreement with the experimental results [1]. However, this value for plates with internal beams having *span-depth* ratio less than 10 is very small in comparison to the former case and varies proportional to the load [see Fig's 17 and 26]. It indicates that plates with shallow beam would attract more interfacial shear in comparison to non-shallow beams thus requires a careful detailing of the shear reinforcement.

Figures 9 to 26 indicates that increase in the beam depth enhances the overall stiffness of the plate-system; thereby producing a comparatively lesser magnitude of the vertical deflection for an identical loading. All plate models exhibit an elastic behavior up to service load and thereafter, shows yielding along the failure lines.

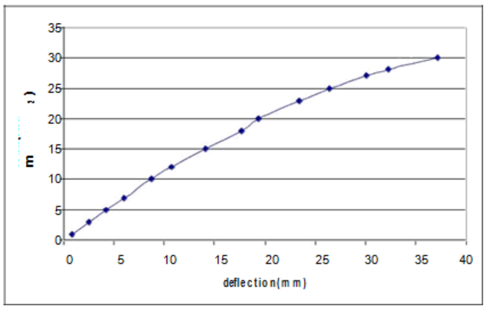

Fig. 9: Load-vertical deflection plot of a 2PSF plate (ANSYS)

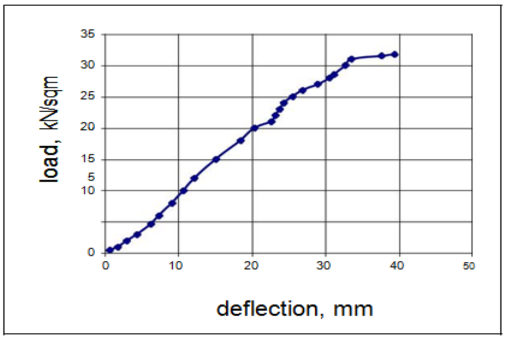

Fig.10: Load-vertical deflection profile of a 2PSF plate [1]

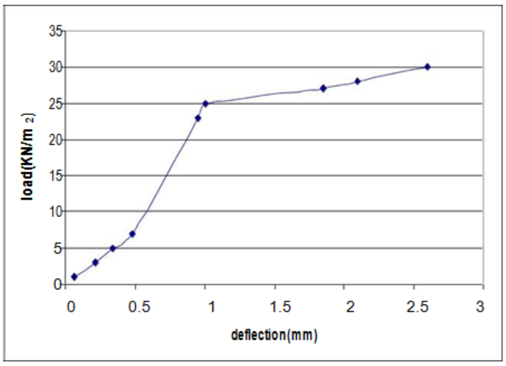

Fig. 11: Load-slip plot of a 2PSF plate (ANSYS)

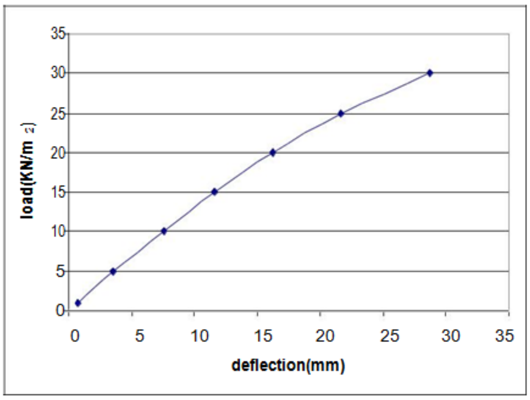

Fig. 12: Load- vertical deflection plot of a 2PSR slab (ANSYS)

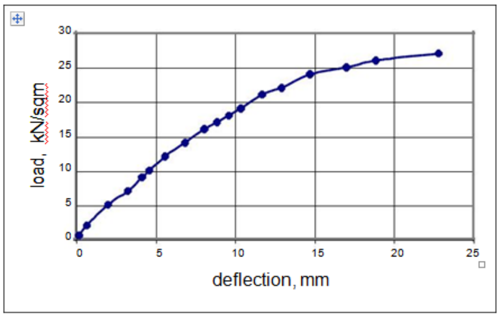

Fig.13: Load-vertical deflection plot of a 2PSR slab [1]

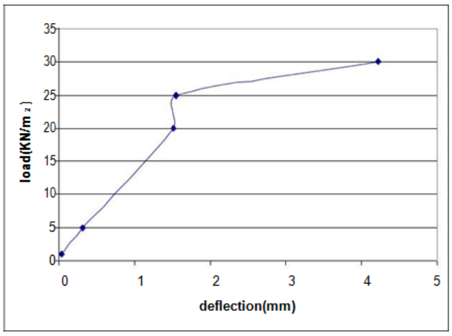

Fig.14: Load-slip plot of a 2PSR plate (ANSYS)

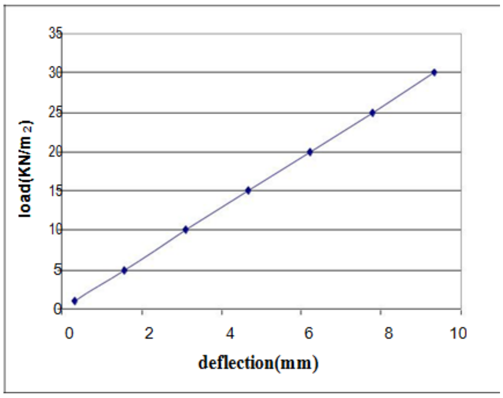

Fig. 15: Load-vertical deflection plot of a 2PNS plate (ANSYS)

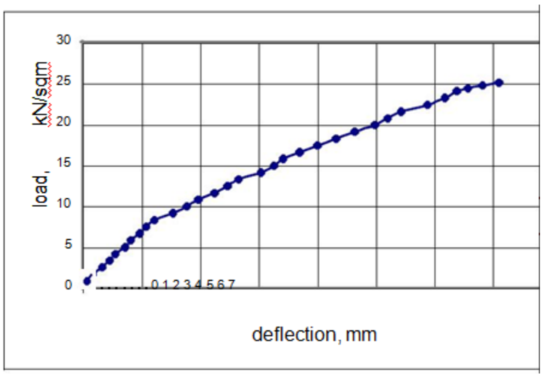

Fig. 16: Load-vertical deflection profile of a 2PNS plate [1]

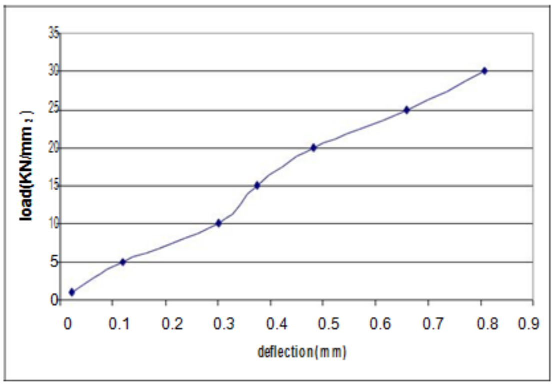

Fig. 17: Load-slip plot of a 2PNS plate (ANSYS)

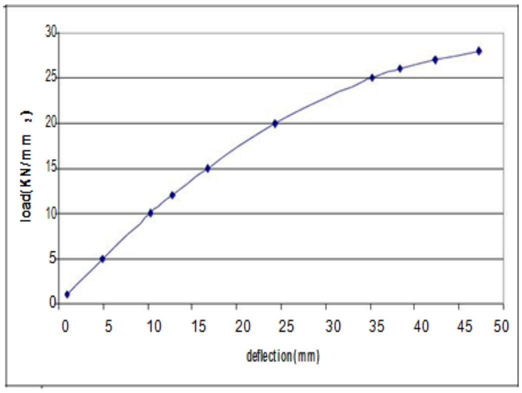

Fig. 18: Load-vertical deflection profile of a 3PSF plate (ANSYS)

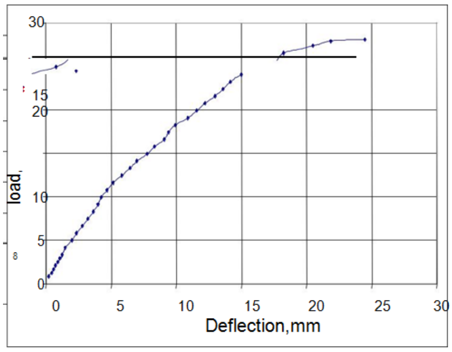

Fig.19: Load- vertical deflection profile of a 3PSF plate [1]

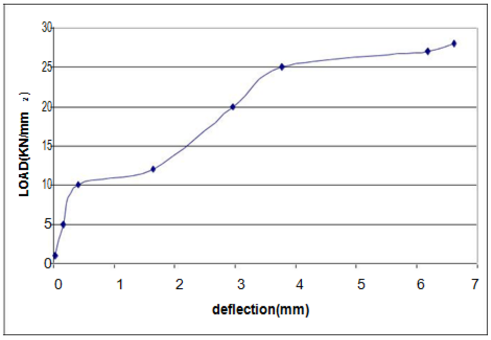

Fig. 20: Load- slip plot of a 3PSF plate (ANSYS)

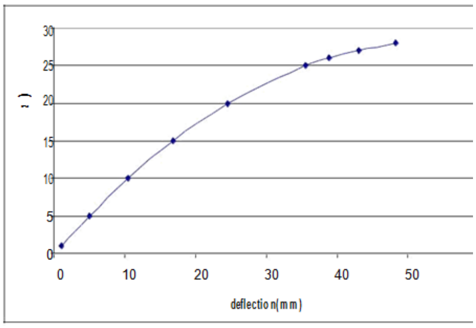

Fig. 21: Load-vertical deflection profile of a 3PSR plate (ANSYS)

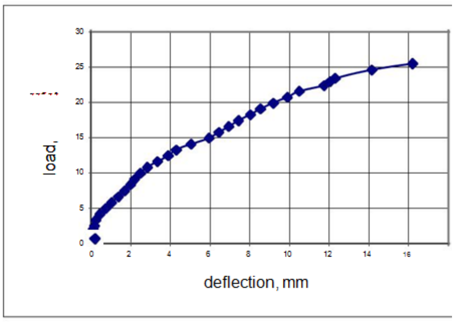

Fig. 22: Load-vertical deflection profile of a 3PSR plate [1]

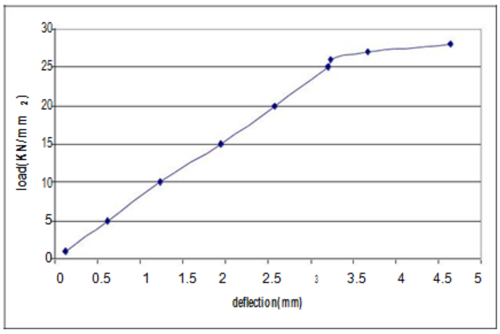

Fig.23: Load-slip plot of a 3PSR plate (ANSYS)

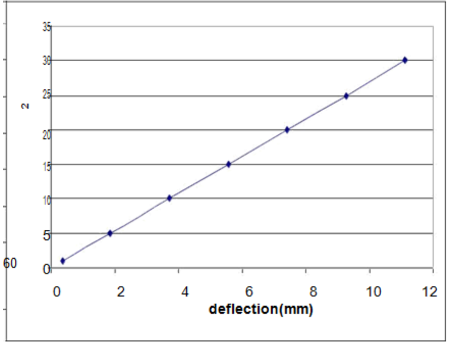

Fig. 24: Load-vertical deflection profile of a 3PNS plate (ANSYS)

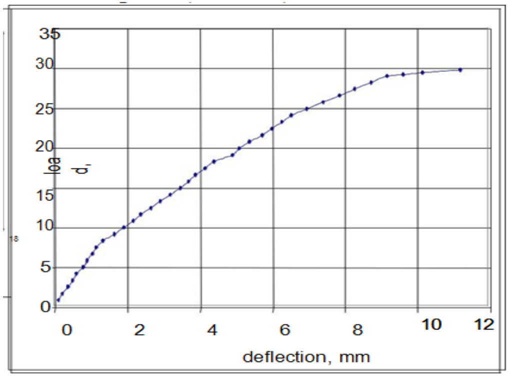

Fig.25: Load-vertical deflection profile of a 3PNS plate [1]

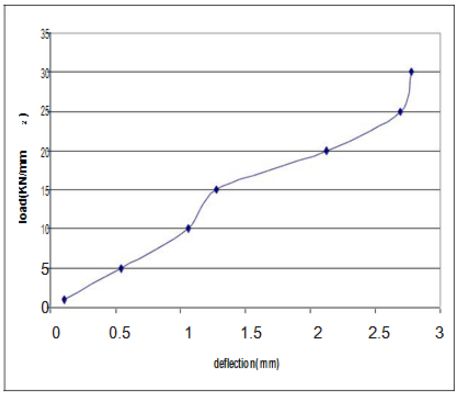

Fig.26: Load- slip plot of a 3PNS plate (ANSYS)

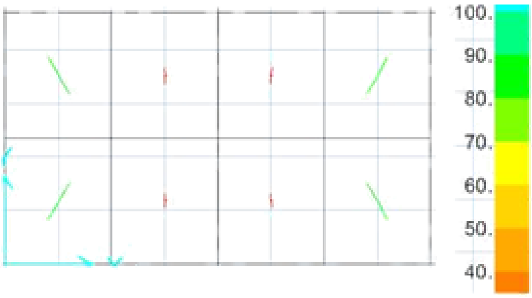

Fig.27: Typical crack patter for 2PSF plate at bottom face (mm)

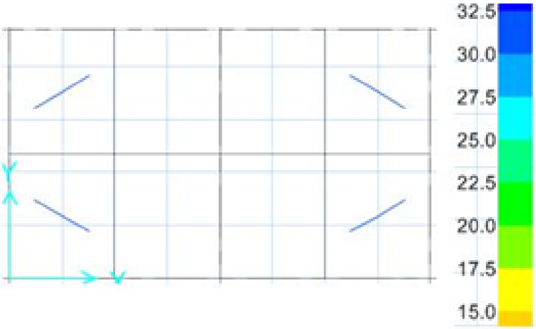

Fig.28: Typical crack patter for 2PSF plate at top face (mm)

Figures 26 to 30 indicate a typical crack pattern at bottom and top face of the stiffened plates. Stiffened-plates designed with beam having *span-depth* ratio more than 10 (*e.g.* 2PSF, 2PSR, 3PSF, 3PSR) exhibits a single panel behavior and allow the flexural crack to pass through the internal beams. No crack was observed at top face of the beams along its length in this case; whereas considerable tensile cracks were noticed along its length in case its span-depth ratio is kept less than 10 (*e.g.* 2PNS, 3PNS). These crack patterns are in good agreement with the experimental observations [1]. Crack width was found to be well within allowable limits prescribed by the serviceability limits of design code [13].

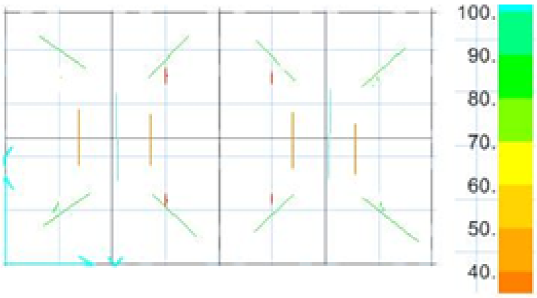

Fig.29: Typical crack patter for 2PNS plate/ at bottom face (mm)

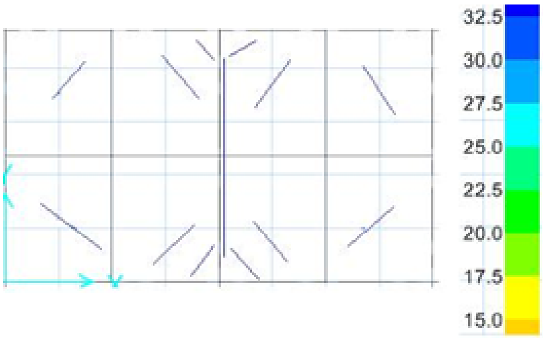

Fig.30: Typical crack patter for 2PNS plate at top face (mm)

IV. CONCLUSIONS

Finite element based software offers a economically feasible alternative to the conventional experimental investigations. It gives more insight to the problem under investigations and can be used to validate the design before actual construction. Following conclusions can be drawn from the present study.

1) The results from the finite element analysis are in good agreement with the experiment results under identical loading and boundary conditions.

2) Plates with shallow beam would attract more interfacial shear in comparison to non-shallow beams thus requires a careful detailing of the shear reinforcement.

3) Increase in the beam depth enhances the overall stiffness of the stiffened-plate system; thereby producing a comparatively lesser magnitude of the vertical deflection for an identical loading.

4) Crack pattern in all stiffened-plate models was observed to be in good agreement with experimental results.

REFERENCES

[1] Harvinder Singh,Maneek Kumar and Naveen Kwatra,Behavior of Shallow-Beam Supported Reinforced Concrete Rectangular Slabs:Analytical and Experimental Investigations,Advances in Structural Engineering,*Volume 13 No. 6*

[2] R. Srinivasan and K. Sathiya, Flexural Behaviour of Reinforced Concrete Beams Using Finite Element Analysis (ELASTIC ANALYSIS),Tomul LVI (LX), Fasc. 4, 2010

[3] Tavio, T and Tata, A, Predicting Nonlinear Behavior and Stress-Strain Relationship of Rectangular Confined Reinforced Concrete Columns with ANSYS, Civil Engineering Dimension, Vol. 11, No. 1, March 2009

[4] Zoran Bonić, Todor Vacev, Verka Prolović, Mathematical Modelling of Materially Nonlinear Problems in Structural Analysis, Architecture and Civil Engineering,Vol. 8, No 2, 2010, pp. 201 – 210

[5] Doc.Ing.Vladimir Ivančo,Nonlinear Finite Element Analysis,Script of Lectures,HS Wismar,June 2011

[6] Barzegar, Maddipudi, and Srinivas, “Three-Dimensional Modeling of Concrete Structures. I: Plain Concrete.” Journal of Structural Engineering, 123(10), 1339-1346, 1997.

[7] Bathe, K. J., Finite Element Procedures, Prentice-Hall, Inc., Upper Saddle River, New Jersey, 1996.

[8] F.A.Fathelbab ,M. S. Ramadan and A.Al-Tantawy,Finite Element Modelling of Strengthened Simple Beams using FRP Techniques: A parametric Study Concrete Research Letters,Vol. 2 (2) – June 2011

[9] Barzegar, F. and Maddipudi, S., “Generating Reinforcement in FE Modeling of Concrete Structures,” ASCE Journal of the Structural Division”Vol. 120, pp. 1656–1662, May 1994.

[10] Bashur,F. K.and Darwin,D., “Nonlinear Model for Reinforced Concrete Slabs,”ASCE Journal of the Structural Division” Vol. 104, pp. 157–170, January 1978.

[11] Borkowski, A.,“Optimization of Slab Reinforcement by Linear Programming,”Computer Methods in Applied Mechanics and Engineering, vol. 12, pp. 1–17, September 1977.

[12] Brotchie, J. F. and Russell, J. J., “Flat Plate Structures: Elastic-Plastic Analysis,”Journal of the American Concrete Institute” vol. 61, pp. 959–996, August 1964.

[13] Bureau of Indian Standards (BIS). (2000). “Plain and reinforced concrete—Code of practice.” IS 456 (4th rev.), New Delhi, India.