World Academy of Science, Engineering and Technology
International Journal of Civil and Environmental Engineering
Vol:12, No:12, 2018

Study on Two Way Reinforced Concrete Slab Using ANSYS with Different Boundary Conditions and Loading

A. Gherbi, L. Dahmani, A. Boudjemia

Abstract**—**This paper presents the Finite Element Method (FEM) for analyzing the failure pattern of rectangular slab with various edge conditions. Non-Linear static analysis is carried out using ANSYS 15 Software. Using SOLID65 solid elements, the compressive crushing of concrete is facilitated using plasticity algorithm, while the concrete cracking in tension zone is accommodated by the nonlinear material model. Smeared reinforcement is used and introduced as a percentage of steel embedded in concrete slab. The behavior of the analyzed concrete slab has been observed in terms of the crack pattern and displacement for various loading and boundary conditions. The finite element results are also compared with the experimental data. One of the other objectives of the present study is to show how similar the crack path found by ANSYS program to those observed for the yield line analysis. The smeared reinforcement method is found to be more practical especially for the layered elements like concrete slabs. The value of this method is that it does not require explicit modeling of the rebar, and thus a much coarser mesh can be defined.

Keywords**—**ANSYS, cracking pattern, displacements, RC Slab, smeared reinforcement.

I. INTRODUCTION

TRADITIONALLY the reinforced concrete structures were designed using empirical methods based on experience or conducting experimental investigations on real structures. While this method yields a high degree of accuracy, it is always very expensive and time-consuming. With the introduction of advanced computers, Finite Element Analysis became a popular tool to analyze and design complicated structures. In this study, the finite element analysis software ANSYS was employed to model the two-way reinforced concrete slab in order to determine the failure pattern and load displacement behavior when subjected to different boundary conditions and loading.

II. FINITE ELEMENT MODELING

A rectangular reinforced concrete slab is discretized into quadrilateral brick elements. The nonlinear analysis is conducted using Ansys commercial finite element program [1], [2]. The smeared reinforcements are used at the bottom as a tensile reinforcement.

A. Physical Model

The geometry of the full concrete slab of size 2x3 m is shown in Fig. 1. The slab has been designed for service load of 18 kN/m^2 which is broken into load steps in order to capture the ultimate response of the specimen. The slab thickness is 200 mm. Concrete cover 25 mm is used, and reinforcement adopted is 8 mm diameter bar @ 250mm c/c.

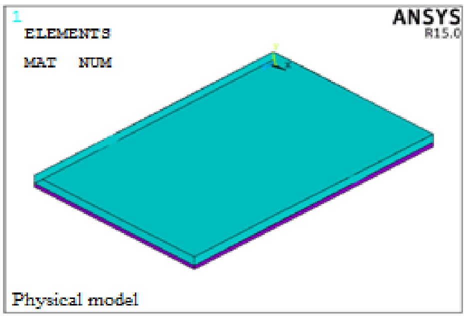


Fig. 1 Physical model

B. Reinforced Concrete Model

An eight-node solid element (SOLID65) was used to model the concrete [6], [7]. The solid element has eight nodes with three degrees of freedom at each node Fig. 2 (a) – translations in the nodal *x*, *y*, and *z* directions. Eight Gaussian integration points are used to recover nodal displacements and stresses Fig. 2 (b). Fig. 2 below shows the solid 65 element used by ANSYS in order to capture the cracking and crushing state in addition to the plastic deformation of the concrete slab.

C. Steel Reinforcements

The reinforcements in concrete slab can be modeled by two methods. A discrete method where the reinforcing is simulated as strut or beam elements connected to the solid elements. This method is more suitable for simple concrete models like beams (Fig. 2 (a)). A smeared method (used in this paper) where the reinforcements are introduced as a volume ratio which is defined as the rebar volume divided by the total element volume. The rebar element effectively sits on top of the existing concrete elements, and thus uses the same nodes as the underlying concrete elements (Fig. 2 (b)). Cracks can also be idealized into either the discrete type or the smeared type.

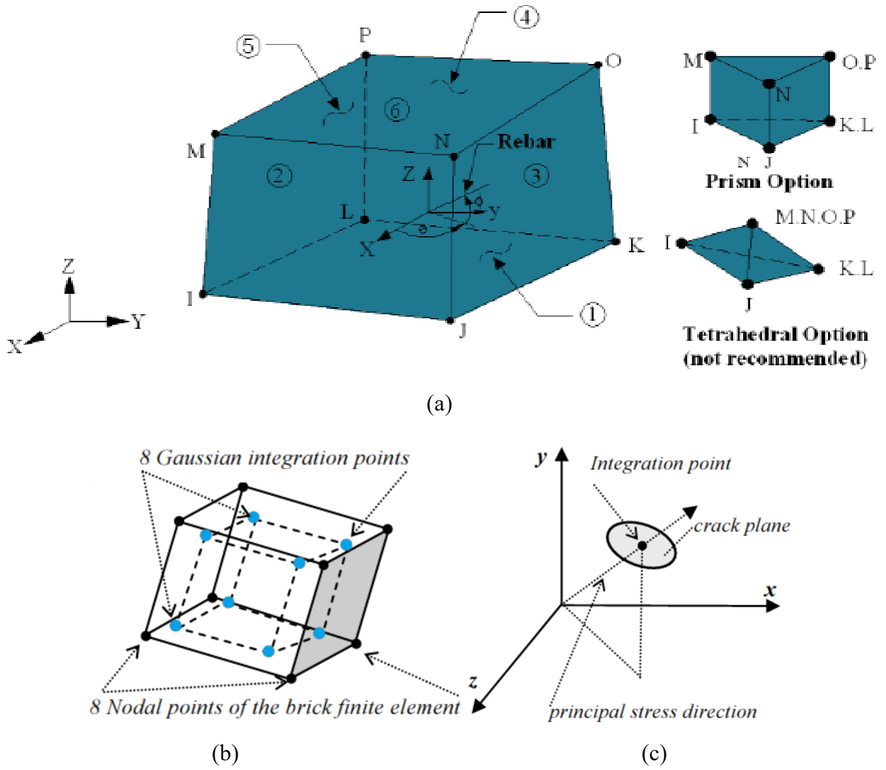
D. Finite Element Discretization

A solid concrete slab model shown in Fig. 1 is descretized with a 3D finite element model as shown in Fig. 4. The hole model is meshed at once with an hexahedral shaped elements along with a smartsizing control featured by Ansys. The stress

---

Ahmed Gherbi is with the University of Mouloud Mammeri of Tizi-Ouzou, Algeria (e-mail: lahlou_d@yahoo.fr).

and strains are then calculated after applying the load and boundary conditions to the finite element model [9]. A slab is composed of two regions; a concrete element without reinforcement (Fig. 5) and a concrete element with a smeared reinforcement (Fig. 6).



(a)

(b) (c)

Fig. 2 SOLID65: 3D reinforced concrete solid (ANSYS 15)

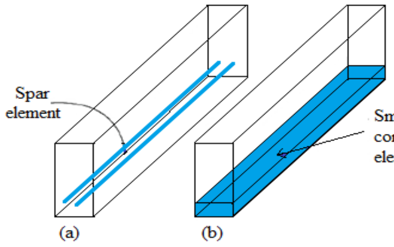


Fig. 3 Discrete vs. smeared element for concrete reinforcing

Two real constant sets for SOLID65 element are created. First one does not have volume ratio (the volume ratio is defined as the rebar volume divided by the total element volume) for the reinforcement, the second one does. Then, two separate volumes adjacent to each other and one under the other are created (Fig. 1). The glue operation used by ANSYS redefines the input volumes so that they share areas along their common boundaries. Before meshing, we choose the first real constant set for the upper one and the second set for the other as mesh attributes (Fig. 4).

III. NONLINEAR SOLUTION STRATEGY

Nonlinear solution technique and overall nonlinear solution strategy to be adopted are very important for nonlinear pre and post-yielding analyses of concrete members. The load is applied gradually by dividing it into a series of increments and adjusting the stiffness matrix at the end of each increment. The ANSYS program [1], [2], [4] uses Newton-Raphson equilibrium iterations for updating the model stiffness. Newton-Raphson equilibrium iterations provide convergence at the end of each load increment within prescribed tolerance limit. In this study, Full Newton-Raphson option with Sparse Direct Solver was used for speed and robustness of the solution.

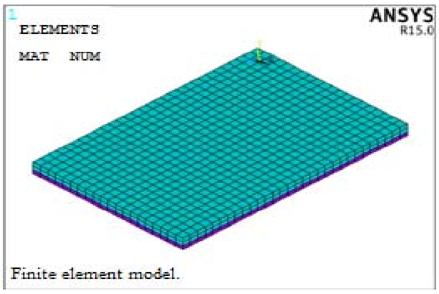


Fig. 4 Finite element model

IV. FAILURE CRITERIA FOR CONCRETE

ANSYS non-linear concrete model is based on William Warnke failure criteria. Two input strength parameters are needed to define a failure surface for the concrete. Once the failure is surpassed, concrete cracks if any principal stresses are tensile while crushing occurs if all the principal stresses are compressive [3], [4], [6], [9].

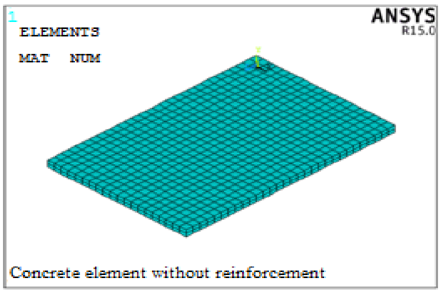


Fig. 5 Concrete element without reinforcement

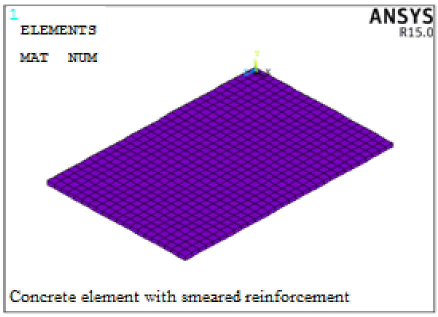


Fig. 6 Concrete element with smeared reinforcement

V. CRACKING ANALYSIS

The William and Warnke failure criterion [5]-[7] under multiaxial stress state is adopted to assess the initiation of failure and identify the corresponding failure modes (including cracking and crushing) at the centroid of a concrete element or one of its integration points. The criterion is expressed uniformly as:

$$\frac{F}{f_c'} - S \geq 0 \qquad (1)$$

where F is a function of the principal stresses, S is the spatial failure surface expressed in terms of the principal stresses and the material properties of concrete, and f_c' is the maximal compressive strength of concrete. Only if the criterion is satisfied, the failure of concrete element or that of its integration points will be assumed to occur [4], [6], [8].

The cracking state of the reinforced concrete slab with different boundaries conditions at different load steps is illustrated in Figs. 8-11 below. It can be observed that more and more dense micro cracks uniformly distributed over the slab supports propagate with the increasing of the load.

The ANSYS program records a crack pattern at each applied load step. In general, flexural cracks occur early at mid-span. When applied loads increase, vertical flexural cracks spread horizontally from the mid-span to the support. At a higher applied load, diagonal tensile cracks appear. Increasing applied loads induces additional diagonal and flexural cracks. Finally, compressive cracks appear at nearly the last applied load steps. The integration point cracking and crushing of the concrete in ANSYS program are represented by circles and octahedron outlines respectively with different colors. A red outline for the first crack, a green outline for the second crack and the blue outline for the third crack (Fig. 7).

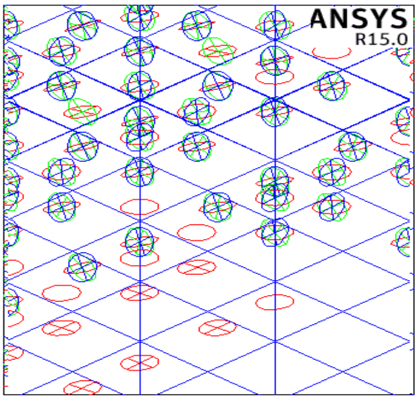


Fig. 7 Cracking and crushing of reinforced concrete

VI. YIELD LINES IN A CONCRETE SLABS

The yield line theory is the plastic analysis method at the ultimate load level. The positive and negative yield lines formed at failure are shown in Fig. 8 for the case of two way reinforced concrete slab with fixed boundary conditions. These lines represent also the position of the maximum positive and negative bending moments for the bottom and the top of slab respectively. At the failure stage, the reinforcing bars have yielded allowing to the cracks to form along the yield lines [7], [10].

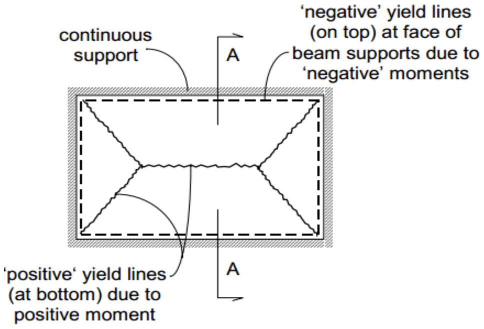
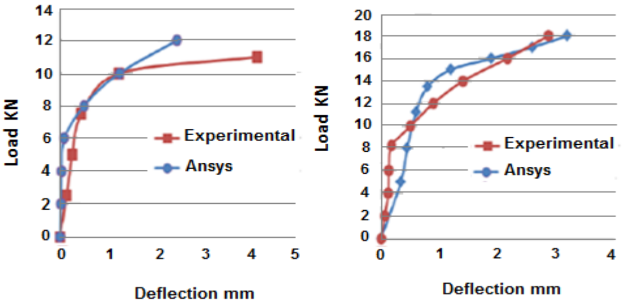


Fig. 8 Yield line patterns in all fixed reinforced concrete slab

VII. RESULT AND DISCUSSION

The validation of the FE models was conducted by comparing the load carrying capacity and load-deflection response of the experimental results with the FE model for the different case studies. Deflections are measured at mid span at the center of the bottom face of the slab. Fig. 6 shows the load-deflection plots of reinforced concrete slab specimens for a simply supported Fig. 9 (a) and fixed sides Fig. 9 (b). In general, the load-deflection plots for the slabs from the finite element analyses agree well with the experimental data.



(a) All sides simply supported (b) All sides fixed

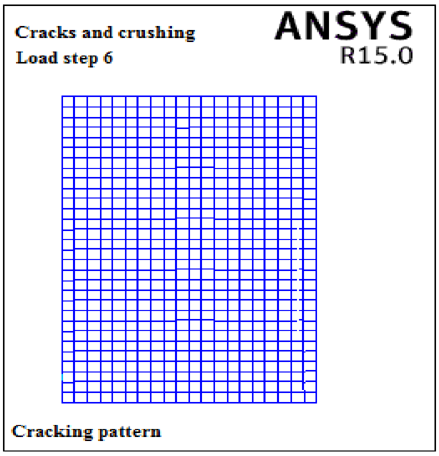
Fig. 9 Load–deflection response of the concrete slab

The crack pattern found by ANSYS program (Figs. 10 and 12) has confirmed Johansen's hypothesis [7] that the location of appropriate yield lines (Figs. 11 and 13) in a two-way reinforced concrete slab follows exactly the same pattern as the crack propagation. The crak formation in the slab starts to develop after oneset of yielding of the reinforcements. The yield line pattern caused by the crack formation reaches its maximum length at failure.

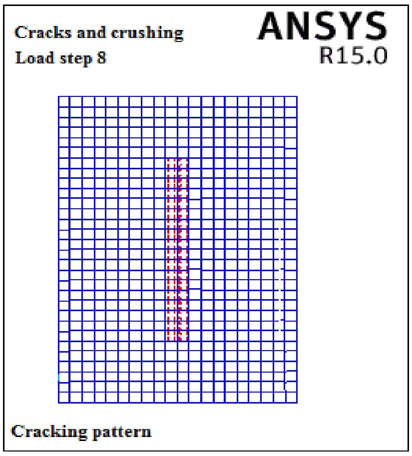
A. Case 1: All Sides Simply Supported

At load step 6 (Fig. 10 (a)), the total uniformly distributed load acting on the slab is 6 kN/m^2. Until this loading, slab behaves elastically. The deformation is small, and up to this point, the Hooke's law is valid. The slab reaches its ultimate collapse load in between 10-12 kN/m^2 and the transverse deflection suddenly increases as the load step increases from 10 to 12. The collapse of slab can also be confirmed by the study of cracking pattern which has been generated in the highly stressed elements just as the ultimate load has been reached.

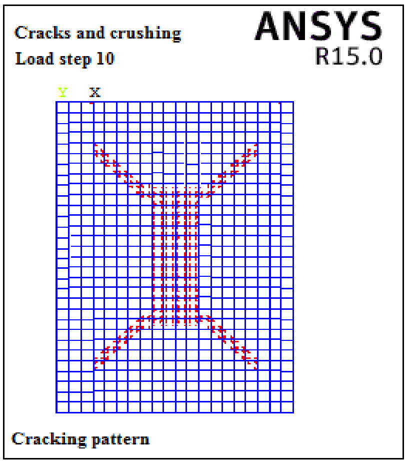
Figs. 10 (a)-(d) are enlisted below showing the difference in the crack patterns from load step 6 to 12. It is conspicuous in the figure that a complete fracture has occurred at load step 12 as the cracks (explicitly representing the yield lines) have reached to the boundaries of the slab. Figs. 11 (a)-(c) represent the corresponding yield line pattern given from the literature [7], [10].



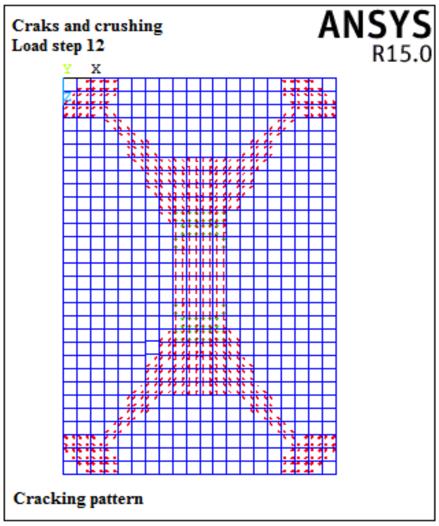
(a) No cracks



(b) First cracking at the bottom face

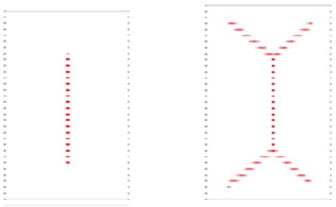


(c) Further cracking

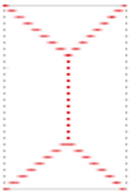


(d) Cracking at failure

Fig. 10 Cracking patterns by ANSYS

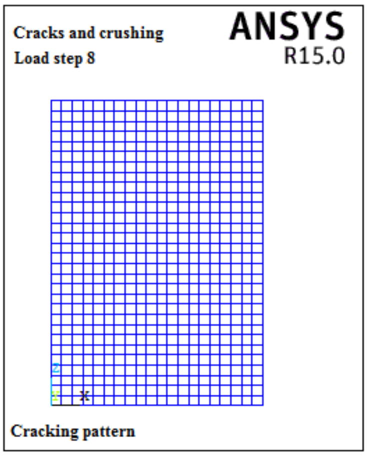


(a) First yielding (b) Further development of yield lines

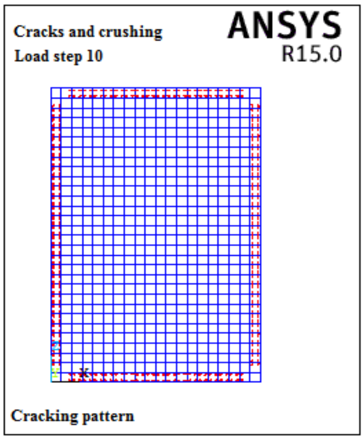


(c) Collapse mechanics formed

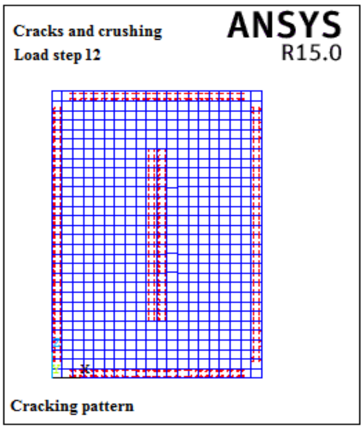
Fig. 11 Yield lines patterns for a simply supported slab [7]



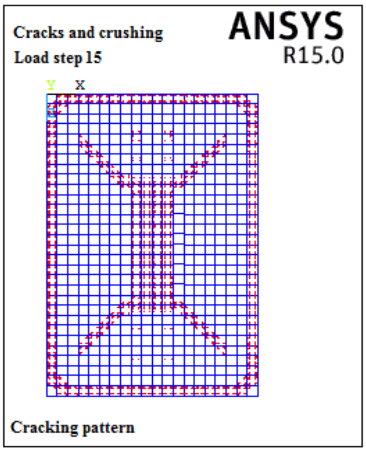
(a) No cracks



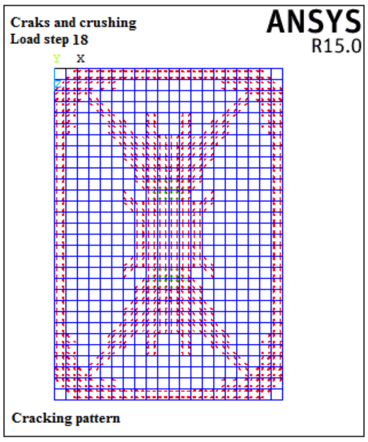
(b) First cracking at the top face



(c) Further cracking (top and bottom face)



(d) More cracks formation



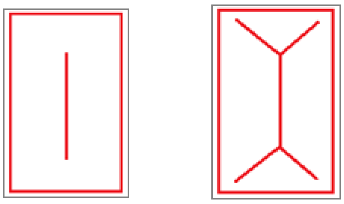
(e) Cracking at failure

Fig. 12 Cracking patterns by ANSYS

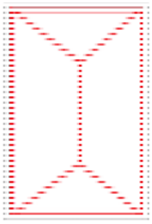
B. Case 2: All Sides Fixed

The sequential failure patterns in the case of all sides fixed slab are shown in Figs. 12 (a)-(e) from the load step 8 to 18. The failure pattern occurs at the highly stressed region (center of slab) and at the fixing support. The failure pattern follows the same pattern as in the simply supported slab with additional cracks at edge of the fixed slab [10].

Figs. 13 (a)-(c) represent the corresponding yield line pattern given from the literature [7], [10].



(a) First yielding (b) Further development of yield lines



(c) Collapse mechanics formed

Fig. 13 Yield lines patterns for a fixed slab [7]

VIII. CONCLUSION

Finite element model of 3x2 m ordinarily reinforced concrete slab, is developed using commercial general-purpose finite element analysis program ANSYS15.0. The concrete model has accurately captured the nonlinear flexural response of the concrete slab up to failure.

The following conclusions can be stated based on the evaluation of the analyses of the reinforced concrete slab:

- The propagation of cracks in the slab (Figs. 10 and 12) confirmed the sequence of yield line formation [7], [10] as the load was increased towards the ultimate collapse load.
- Flexural failure of the reinforced concrete slab is adequately modeled using a finite element package ANSYS, and the load applied at failure is very close to the experimental results (Fig. 8).

The failure model of concrete [5], [6] adopted by the commercial code ANSYS with the smeared reinforcement approach is adequate to determine the nonlinear behavior of reinforced concrete structures. Using Finite Element Modeling enables us to predict the results of experimental work before starting, which leads to minimize the cost of laboratory work.

REFERENCES

[1] ANSYS 15.0 Manual Set, ANSYS Inc., Canonsburg, PA, 2014.
[2] ANSYS Theory Reference, Seventh Edition, Swanson Analysis Systems, 2014.
[3] D. Kachlakev and T. Miller, “FE Modeling of Reinforced Concrete Structures Strengthened with FRP Lamiates”, Final Report SPR 316, Oregon State University, 2001.
[4] Kitjapat Phuvoravan and Elisa D. Sotelino, “Non Linear Finite Element for Reinforced Concrete Slabs”, Journal of Structural Engineering, vol. 131, No.4, 2005, pp.643-649.
[5] K. J. Willam, University of Colorado (Private communication), 1982.
[6] K. J. William and E. D. Warnke, “Constitutive model for the triaxial behavior of concrete,” in: Proc. of the Int. Association for Bridge and Structural Engineering, ISMES, Bergamo, 1975, Vol. 19, p. 174.
[7] Johansen, K.W. (1962). Brudlinieteorier, Copenhagen, Denmark. (in Danish) (English translation: Yield Line Theory, Cement and Concrete Association, London, UK.)
[8] L. Dahmani, A. Khennane, S. Kaci, ”Crack identification in reinforced concrete beams using ANSYS software” , Strength of Material Journal, Ed. Springer New York , Vol.42, N°2, Mai **2010**. Pages 232-240.
[9] O. O. R Famiyesin, K. M. A Hossain, Y. H. Chia and P. A. slade, “Numerical and Analytical predictions of the Limit Load of Rectangular Two Way Slabs”, Computers and Structures, vol.79, 2001, pp.43-52.
[10] Rajesh Kumar, V. Kumar2, Alok Kumar3 and Shivang D. Agrawal, “Yield Line Analysis of Rectangular Slabs by Finite Element Method”, in Proc. of the Second Intl. Conf. on Advances In Civil, Structural and Environmental Engineering- ACSEE 2014.USA.