IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 15, Issue 6 Ver. I (Nov. - Dec. 2018), PP 45-54
www.iosrjournals.org

Nonlinear Analysis of the Effect of Horizontal Construction Joint on the Flexural Behavior of Reinforced Concrete Slabs

Mohamed abou elmaaty amin[1],Mohamed Mahmoud hussein[2],
Mohamed Elshrakawy [3]
[1](Civil engineering department, Faculty of engineering, Fayoum university,Egypt)
[2](Civil engineering department, Faculty of engineering, Zagazig university,Egypt)
[3](Civil engineering department, Giza Engineering Institute,Ministry of Higher Education, Egypt)
Corresponding Author: Mohamed abou elmaaty amin

Abstract: *The Study of the behavior of slabs under the influence of horizontal construction joints has become necessity in the case of large thickness slabs such that raft and some special cases of flat slabs which were casted in two or more layers. The first layer is usually casted up to cover the reinforcement steel mesh and the other layers were casting to get the actual thickness of slabs. The purpose of this research is to examine the effect of horizontal construction joints on the behavior of simply supported slabs. A commercial non linear finite element program, such as "ANSYS version 12" [1] was used to study the previous slabs. The results which obtained from theortical analysis wre comared with that obtained from experimintal work [2]. The main parametric study used in this search was the case of interface layer between the two layers of the reinforced concrete slabs. A comparison was made between the slab which was casted as one layer and to that slabs which were casted in multy layers with different interface properties the results showed that the results from theortical modeling and experimintal work were nearly approximate as well as the studding of horizontal construction joints on the flexural behavior of slabs is very important.*

Key Words: Nonlinear, finite element, horizontal joints, flexural behavior slabs.

Date of Submission: 09-11-2018 Date of acceptance: 24-11-2018

I. Introduction

Horizontal lift joints in roller-compacted concrete structures as shown in Fig. 1 are the planes of the weakest subject to leakage, deterioration, and possible failure from tensile or shear stresses [3].

Fig. 1- Construction of horizontal joint [1]

All codes of specification treatment the horizontal joints by different ways , ACI Code [4] recommended that the concrete surface of construction joint shall be cleaned and laitance removed. Immediately before new concrete is placed, all construction joints shall be wetted and standing water removed. British Standard: BS 8110 [5] recommended that It is necessary for a joint to transfer tensile or shear stresses, the surface of the first pour should be roughened to increase the bond strength and to provide aggregate interlock. With horizontal joints, the joint surface should, if possible, be roughened, without disturbing the coarse aggregate particles. Australian Standard: AS 1480-1982 [6] recommended that before fresh concrete is placed against hardened concrete at a construction joint, the joint surface of hardened concrete shall be thoroughly roughened and cleaned, so that all loose or soft material, free water, foreign matter and laitance are removed. Indian Standard: IS 456:1978 [7] state that for horizontal joints, the surface shall be covered with a layer of mortar about 10 to 15 mm thick composed of cement and sand in the same ratio as the cement and sand in concrete mix. This layer of cement slurry or mortar shall be freshly mixed and applied immediately before placing the concrete. Egyptian Code 2017 [8] state that when the work has to be resumed on the horizontal construction joint (after more than one day), the surface of the hardened concrete shall be carefully scrabbled to expose the coarse aggregate. The surface shall be cleaned and loose or soft material removed. The surface shall be then thoroughly wetted. A layer of water-cement grout or bonding new to old concrete paint shall be sprayed.

The aims of this research is to investigate the different techniques of bending the horizontal joint in reinforced concrete slab by using several friction factors between the old layer and the new layer of reinforced concrete slabs and compare the results with that slabs which cast in one layer. A commercial non linear finite element program, such as "ANSYS version 12" was used to study the previous slabs.

II. Methodology

The finite element method was chosen in this search [9] to study the non-linear structural behavior of a slabs which were casted in multi layers under applied dead & live loads. Both analytical and finite element solutions are based on the governing differential equations. The largest difficulty using analytical methods is to find a function that fulfills the differential equation and the boundary condition over the entire body. A commercial non linear finite element program, such as "ANSYS version 12" was used to study the previous slabs.

2.1 Non-linear structural analysis

2.1.1 Material non-linearity

The material non-linearity is assumed to occur due to the nonlinear stress-strain relationship. Many factors can influence a materials stress-strain property, including load history, environmental conditions (such as temperature) and the amount of time that a load is put on.

2.1.2 Geometric non-linearity

Geometric nonlinearities refer to the nonlinearities in the structure or component due to the changing geometry as it deflects. That is, the stiffness [K] is a function of the displacements {U}. Therefore stiffness changes as the shape changes. Geometric non-linearity can be sorted out in the large stock analysis and large deviation analysis. The finite element yields a set of simultaneous equations that can be written in the form shown in equation (1):

$$[K]\{U\} = \{F\} \quad (1)$$

Where:

[k] = Stiffness matrix.
{U} = Vector of unknown displacements.
{F} = Vector of applied load.

2-2 The elements which used to represent finite element model

2-2-1 Concrete element

The concrete material is subjected to two possible failure modes. Cracking in tension and crushing in compression. The 3D solid element (SOLID65) was selected to perform this analysis using "ANSYS version 12", because it is capable of both cracking in tension and crushing in compression. (SOLID65) allows for four different materials within the element, one matrix material and a maximum of three independent reinforcing materials. In concrete applications, the solid capability of the element is used to model concrete.

The element (SOLID65) is defined by 8 nodes having three degrees of freedom each. Eight integration points were used for evaluating element stiffness.

The geometry, node location, and the coordinate system for this element are shown in Fig. 1. The element material is assumed to be isotropic and the most important aspect of this element is the treatment of nonlinear material properties where concrete is capable of directional cracking and crushing besides incorporating plastic and creep behavior.

In the previous paragraph, we noted that the element (SOLID65) had the capability of cracking and crushing. The cracking and crushing are the most significant factors contributing to nonlinear behavior of the concrete. If the material at an integration point fails in compression, the material is assumed to crush at this point. For the concrete element used (SOLID65), crushing is defined as the complete deterioration of the structural integrity of the material, i.e. under the crushing conditions, the material strength is assumed to be degraded to an extent such that the contribution of the element stiffness at the integration point in question can be ignored.

There are two techniques of crack representation in any finite element program. Sceared crack modeling and the discrete modeling. The first type occurs by adjusting of the material properties to introduce a plane of weakness in a direction normal to the crack face. While the second type occurs by separation of appropriate nodes of adjoining elements. The crack modeling adopted by (ANSYS) program is the sceared crack representation.

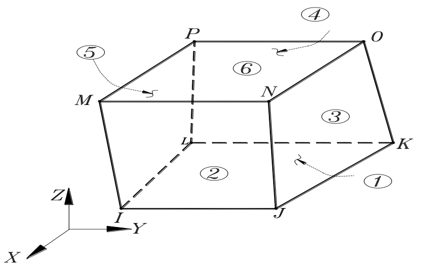

Fig. 1 Three dimensional reinforced elements

2-2-2 Internal reinforcement element

To model the reinforcing steel bars, the 3D spar element (LINK 180) available in the element library of the (ANSYS) program was used. The three dimensional spar elements (LINK 180) is a uniaxial tension-compression element with three degrees of freedom at each node. No bending is considered for this element. The element is also capable of plastic deformation, creep, swelling, and stress stiffing. The geometry, node location, and the coordinate system for this element are shown in Fig. 2 the element is defined by two nodes, the cross-sectional area, an initial strain, and the material properties. The X-axis of the element is oriented along length from node I towards J. the solution output associated with the element is the nodal displacements included in the overall nodal solution and some other additional element output as shown in Fig. 2

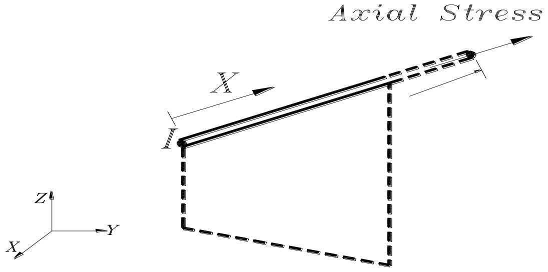

Fig. 2 Solution output associated with element (LINK 180)

2-3-3 Supporting element

SOLID185 is used for 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials as shown in Fig. 3.

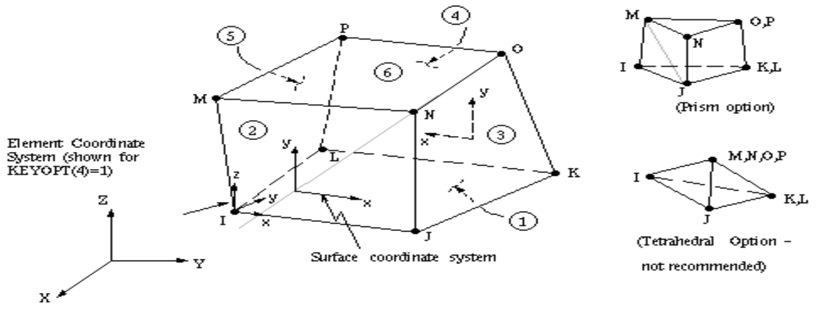

Fig. 3 Solid185 element in 3-D

III. Finite Element model

3.1 Model details

A finite element program (ANSYS V12) [1] was used to study the horizontal joints on reinforced concrete slabs. Three-dimensional finite element models were developed to simulate the envelope response of the test slab specimens were square with 1100 mm side length and 60 mm thickness. The test specimens were simply supported along two edges .

3.2 Mesh configuration

The model used a mesh of element size ranging from a minimum of 15x15x15 mm to a maximum of 25x25x25 mm. The finite element mesh is shown in Fig. (4-A,B,C). Due to symmetry about the X-Y plane only one half of the model was used in this analysis.

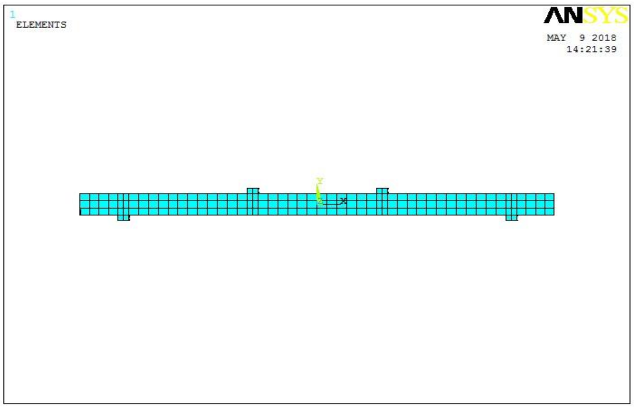

Fig. 4-A The finite element mesh’s

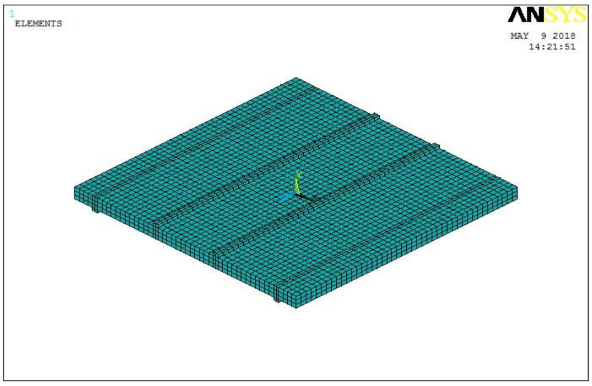

Fig. 4-B The finite element mesh’s

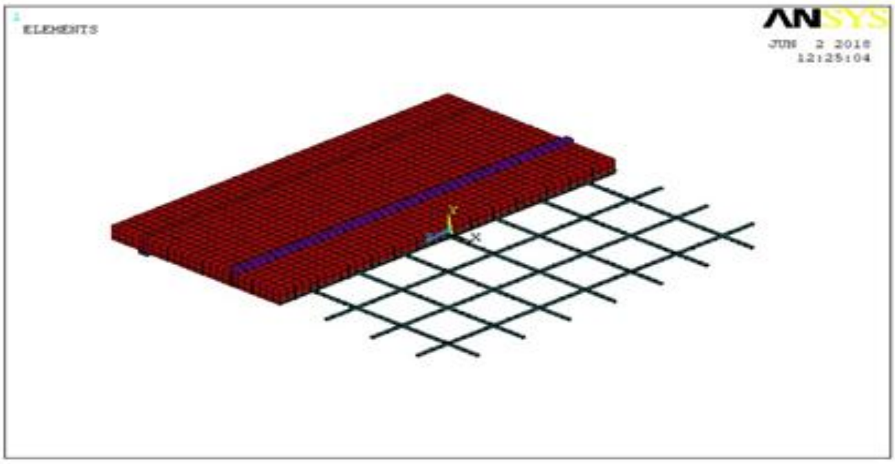

Fig. 4-C The finite element mesh’s

3.3 Model restraints

The details of Slab restrains are shown in Fig. (5-A,B). The start, end of the slab was restrained in the vertical direction and the horizontal direction.

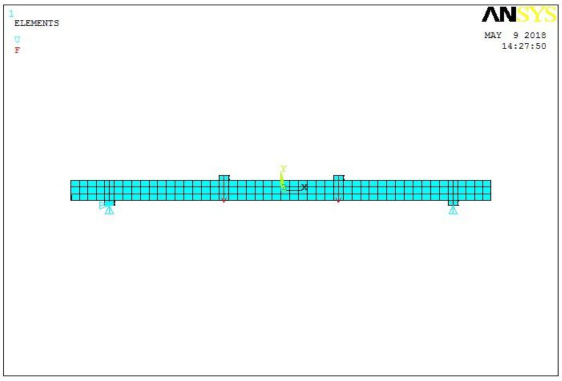

Fig. (5-A) The details of Slab restrains

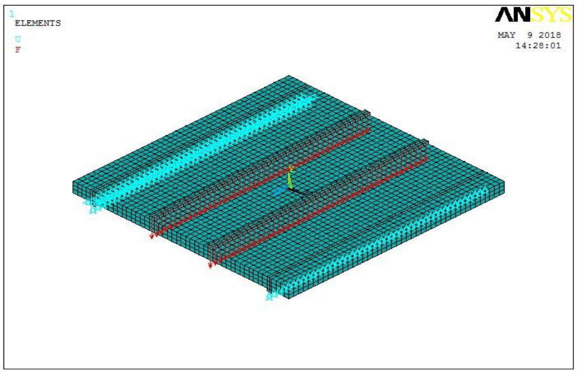

Fig. (5-B) The details of Slab restrains

3.4 Loading scheme and loading increments

The slab was exposed to tow line load located over the nodes at the upper of the Slab. In (ANSYS) the load can be applied in steps. Each load step is divided to load increments, as shown in Fig. 6.The (ANSYS) solution requires the user to define a maximum number of iterations for each load increment. Within this number of iterations the solution will continue to the next load step if the out of balance forces are within a prescribed limit. For the analysis under hand only one load step was used to define the load on the column. The load on the column was gradually increased until failure occurred. The size of the load increments were chosen to help achieve convergence and at the same time attains an acceptable level of accuracy. Small load increments usually lead to better accuracy and improved convergence with the penalty of more computational cost.

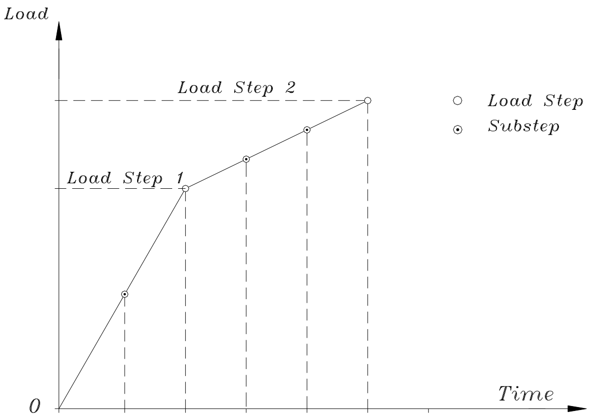

Fig. 6 Load steps, sub steps and time

3.5 Material properties

3.5.1 Concrete

In the finite element model, the Young's modulus for concrete was taken 2.4x105 (kg/m2) and Poisson's ratio was assumed to be (0.2). Additional concrete material data needed for (SOLID65) were the shear transfer coefficients, tensile stresses, and compressive stresses. Typical shear transfer coefficients range from 0.0 to1.0, with 0.0 representing a smooth crack (complete loss of shear transfer) and 1.0 representing a rough crack (no loss of shear transfer). This specification may be made for both the closed and open crack.In general the concrete material data table consisted of nine parameters; these parameters are described in Table 1.

Absence of these additional parameters removes the cracking and crushing capability of the element. A value of -1 for constant 3 or 4 also removes the cracking or crushing capability, respectively. If constants 1-4 are input and constants 5-8 are omitted, default values are used for the latter constants. If any one of constants 5-8 is input, defaults are not used and all 8 constants must be input. In the current analysis, the crushing capability of concrete was omitted, and the stress-strain relationship shown in Fig. 7 was used instead, to simulate the crushing of concrete.

Table (1) General concrete parameters

parameter	Definition	model
1	Shear transfer coefficient for an open crack.	0.3
2	Shear transfer coefficient for closed crack.	1
3	Uniaxial tensile stress (positive).	20
4	Uniaxial crushing stress (positive).	-1
5	Biaxial crushing stress (positive).	0
6	Ambient hydrostatic stress state used with (parameters 7 and 8).	0
7	Biaxial crushing stress (positive) under the Ambient Hydrostatic stress state (parameters 6).	0
8	Uniaxial crushing stress (positive) under the ambient Hydrostatic stress state (parameters 6).	0
9	Stiffness multiplier for cracked tensile condition. Used if the tensile stress relaxation is activated after cracking.	0.9

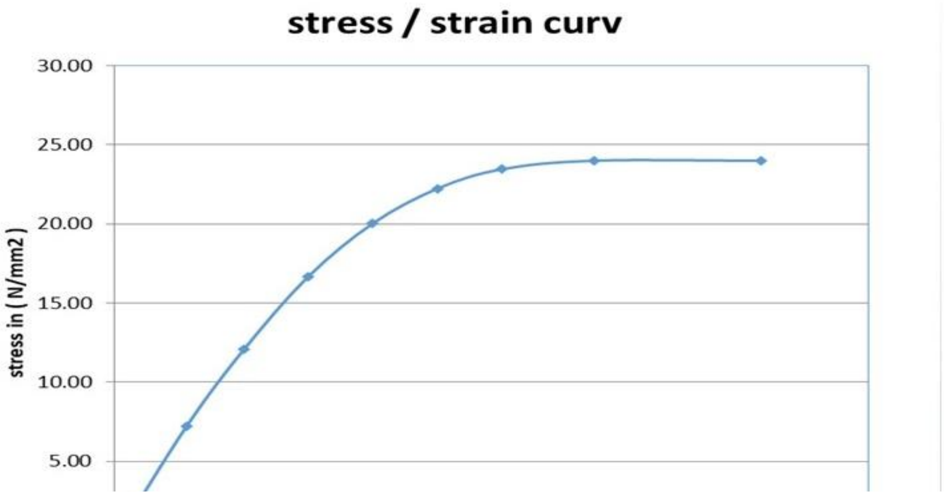

Fig. 7 Stress-strain relationship in compression for concrete

3.5.2 Internal reinforcement

The idealized stress-strain curve for the internal vertical and horizontal reinforcement used in the finite element model is shown in Fig. 8.The yield stress, Fy = 330 (N/mm^2).

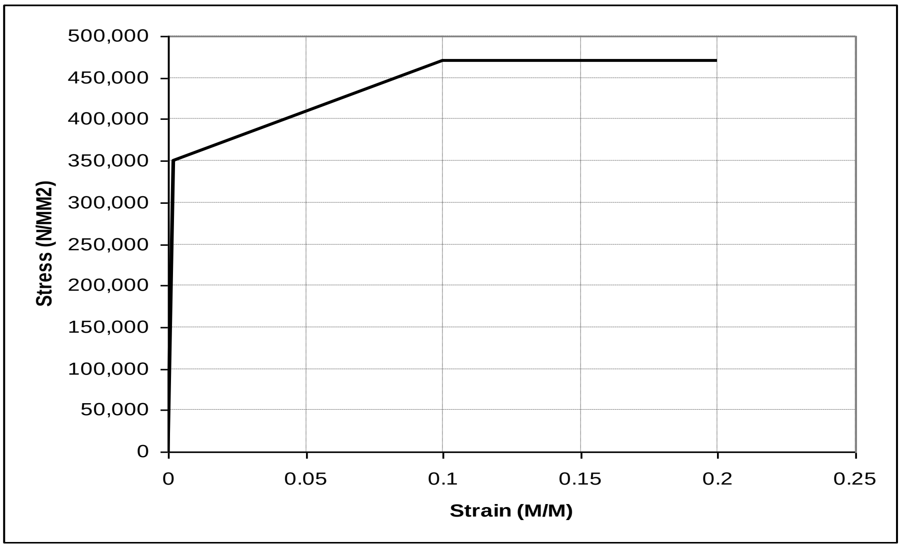

Fig. 8 Idealized stress-strain for reinforcing steel bars and steel angle

IV. Verification between Experimental &Analytical model

Metwally, I. M . et . al [2] was tested different slabs experimentally in (HBRC) lab To study the effect of horizontal construction joint, Table 2 and Fig. 9 show the tested slabs and their properties. For the purpose of verification between experimental results and theoretical results which obtained from finite element program the following model will be drawn.

Table 2 Tested Slabs

Slab name	Slab DIM (mm)	Slab RFT	Slab Case	Case of interface
Ss	1100*1100*60	6 Ø 8 /m/	One layer	NO
So	1100*1100*60	6 Ø 8 /m/	Two layer	50 % Friction
S1	1100*1100*60	6 Ø 8 /m/	Two layer	Bonded

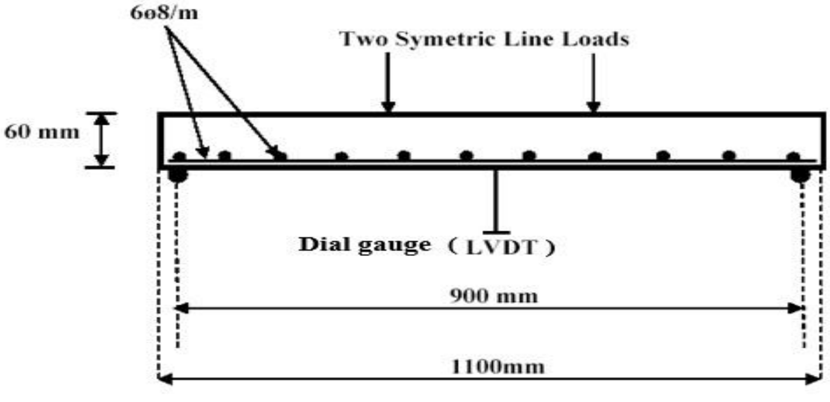

Fig. 9 Typical concrete dimensions and reinforcement details (All dimensions in mm)

V. Results and Discussion

5.1 Ultimate Load

Fig. 10 shows the ultimate load capacity obtained from experimental and analytical analysis for tested slabs. The Fig. shows the results which obtained from analytical analysis are nearly approximate to that obtained from experimental work.

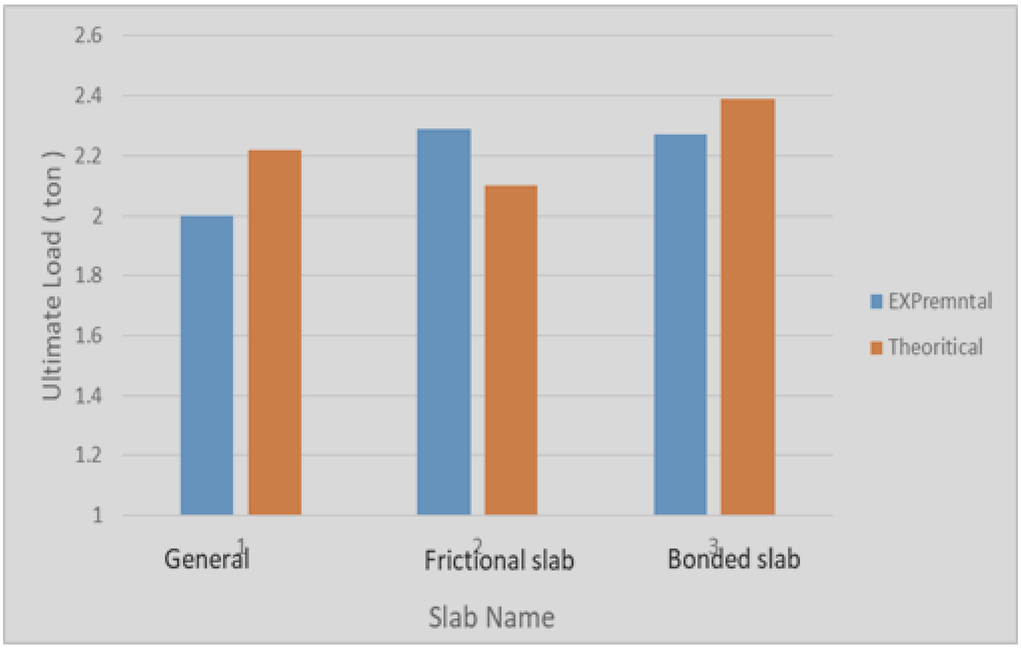

Fig. 10 Comparison in Ultimate load between the Experimental & Analytical slabs

5.2 Load-Deflection Response

Fig 11 shows the load & deflection curve from Experimental and analytical analysis for tested slab (Ss).

The Fig. shows the results which obtained from analytical analysis are nearly approximate to that obtained from experimental work. Fig 12 shows the comparison between load & deflection curve from Experimental and analytical analysis for slab as two layer (So). The Fig. shows the results which obtained from analytical analysis are nearly approximate to that obtained from experimental work.

Fig 13 shows the comparison between load & deflection curve from Experimental and analytical analysis for slab as two layer (S1). The Fig. shows the results which obtained from analytical analysis are nearly approximate to that obtained from experimental work.

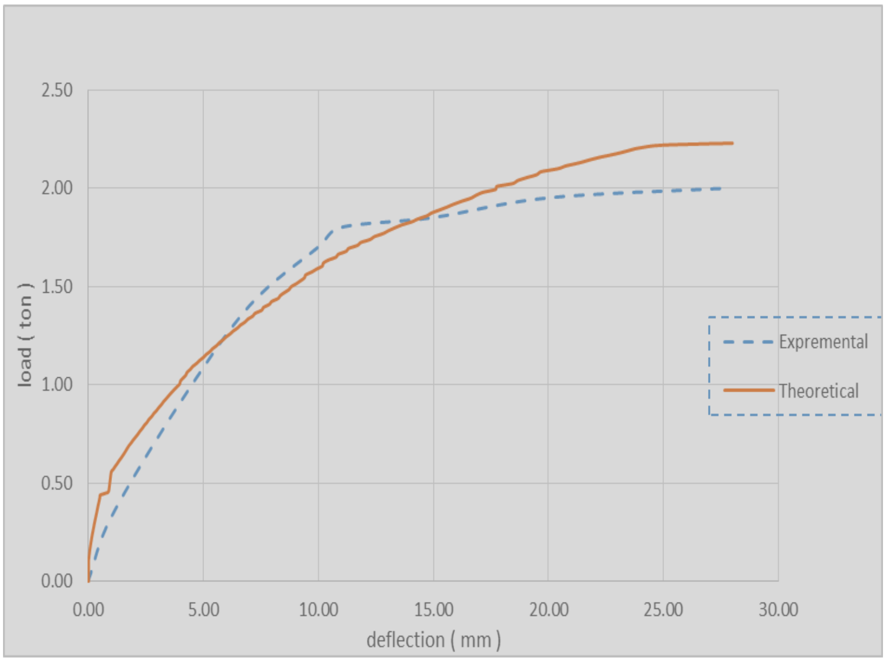

Fig. 11 Load-deflection for reference slab (Ss)

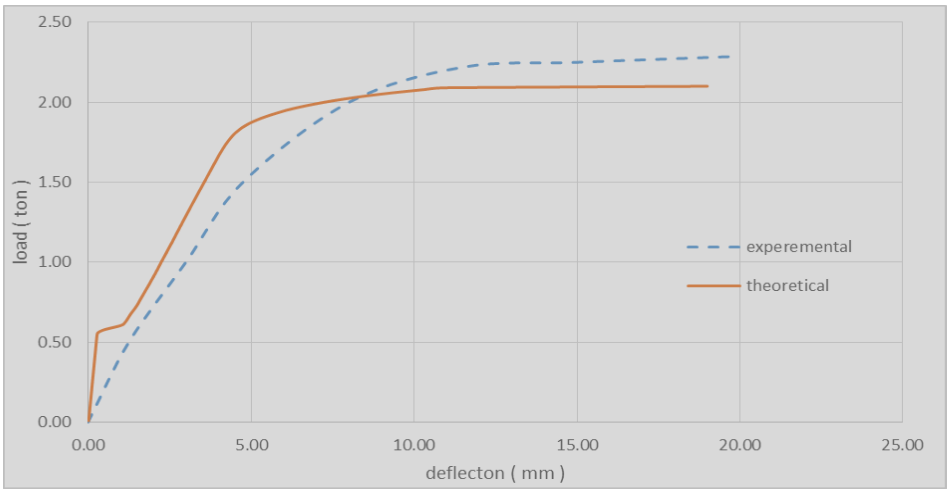

Fig. 12 Load-deflection for slab (so)

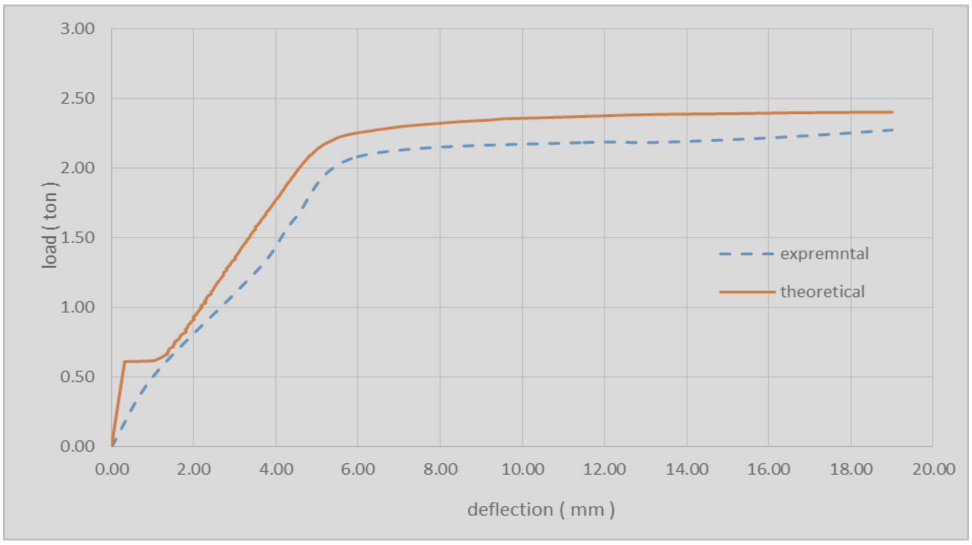

Fig. 13 Load-deflection for slab (s1)

5.-3 Toughness

Flexural toughness or energy absorption is defined as the area under the load-deflection curve up to a deflection equal to the span length divided by 150 [4]. The flexural toughness values for various slabs calculated at the designated deflection of 7 mm are shown in Table 3 and Fig.14 respectively.

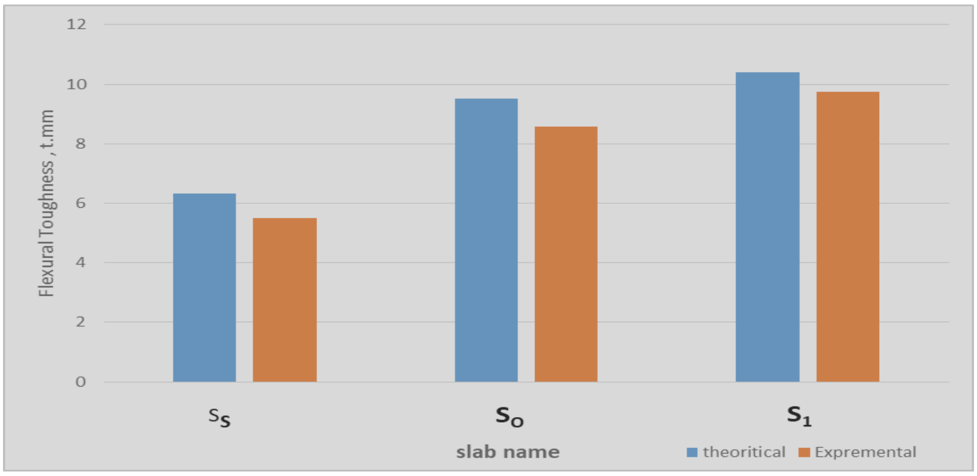

Fig. 14 Comparison in Flexural Toughness between the Experimental & Analytical slabs

Table 3 Results of the tested R. C. Slabs

Slab Coad	Ultimate Load (ton)			Deflection (mm)			Toughness (ton, mm)		
	EXP	Theo	%Diff	EXP	Theo	%Diff	EXP	Theo	%Diff
Ss	2	2.22	10 %	27.9	25	10.5%	5.5	6.34	13%
So	2.29	2.1	8.3 %	20	19	5 %	8.57	9.50	10.8%
S1	2.27	2.39	5 %	19	18.9	1%	9.76	10.40	6.5%

Fig 14 shows the comparison the toughness between Experimental and analytical analysis for slabs. The Fig. shows the results which obtained from analytical analysis are nearly approximate to that obtained from experimental work.

VI. Conclusion

Table 3 shows a comparison between the results which obtained from experimental and theoretical analysis. The results show that the different between experimental and theoretical analysis varies from (5% to 10%) for ultimate load and (1% to 10.5%) for deflection and (6.5% to 13%) for toughness.

According to the previous results the numerical models can be used to analysis the behavior of multi layers slabs (Slab with horizontal joint) under different parametric study as the effect of change the compressive strength of the concrete layers, number of layers thickness of layers, interface surface between layers.

References

[1]. ANSYS Verification Manual "Release 12.0" ANSYS, Inc., United States.

[2]. Metwally, I. M., and Issa, M. S., "Influence Of Horizontal Construction Joint On The Flexural Behavior Of Reinforced Concrete Slabs " *HBRC Journal, Vol.3, No. 3 December* 2007 pp (81-91).

[3]. Hess, J. R., Jan., "RCC Lift-Joint Strength", Concrete International, 2002, pp. 50-56.

[4]. ACI Committee 318R-89, "Commentary on Building Code Requirements for Reinforced,Concrete", 1989.

[5]. British Standard: BS 8110:, "Structural Use of Concrete", Part 1, Code of Practice forDesign and Construction of Construction Joint, 1985.

[6]. Australian Standard: AS: 1480-1982.

[7]. Indian Standard: IS: 456-1978.

[8]. Egyptia Code of Practice for Reinforced Concrete Structures: Construction and Design, 2017.

[9]. Carey, G.F., "A unified approach to three finite element theories for geometric nonlinearity".*Computer Methods in Applied Mechanics And Engineering* 4 (1974) pp (69-79).

Mohamed abou elmaaty amin. “Nonlinear Analysis of the Effect of Horizontal Construction Joint on the Flexural Behavior of Reinforced Concrete Slabs.” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) , vol. 15, no. 6, 2018, pp. 38-54