Journal of Engineering and Development, Vol. 17, No.2, 2013, ISSN 1813- 7822

Non- Linear FE Modeling of Two-Way Reinforced Concrete Slab of NSC, HSC and LWC under Concentrated Load

Lect. Shatha S. Kareem
Civil Eng. Dept
College of Engineering
Al Mustansiriya University.

Asst. lect. ShaimaaT. Sakinand
Highway and Transp. Eng. Dept
College of Engineering
Al Mustansiriya University.

Asst.Lect. Mohammad Z. Yousif
Highway and Transp. Eng. Dept
College of Engineering
Al Mustansiriya University.

Abstract

This paper presents the numerical study to simulate the behavior six specimens of two-way RC slab of normal strength concrete (NSC), high strength concrete (HSC) and light weight concrete (LWC) with two steel ratio of 0.005 and 0.002 under concentrated load. The finite element (FE) model is developed and the simulation environment is conducted using the commercial finite element program, ANSYS (9). The concrete material is modeled using solid elements that account for concrete cracks and other material nonlinearities. Link elements are used to model the internal steel reinforcement. The results of FE show good agreement with the experimental results.

Key Words: Reinforced concrete slab, nonlinear finite element modeling, flexural, punching, NSC, HSC and LWC modeling, and ANSYS.

التحليل اللاخطى بطريقة العناصر المحددة للبلاطات الخرسانية المسلحة باتجاهين و المصنوعة من الخرسانة الاعتيادية والعالية المقاومة والخفيفة الوزن تحت تاثير الاحمال المركزة

م.م. محمد زهير يوسف
قسم هندسة الطرق والنقل
كلية الهندسة
الجامعة المستنصرية

م.م. شيماء طارق ساكن
قسم الهندسة المدنية
كلية الهندسة
الجامعة المستنصرية

م. شذى صاحب كريم
قسم هندسة الطرق والنقل
كلية الهندسة
الجامعة المستنصرية

الخلاصة:

يقدم هذا البحث دراسة عددية لتمثيل سلوك البلاطات الخرسانية المسلحة باتجاهين و المصنوعة من الخرسانة الاعتيادية المقاومة (NSC) و الخرسانة العالية المقاومة (HSC) و الخرسانة خفيفة الوزن (LWC) مع نسب مختلفة من حديد التسليح (٠.٠٠٢، ٠.٠٠٥) تحت تأثير أحمال مركزة. تم تطوير نموذج التحليل العددي باستخدام برنامج ANSYS 9 . لقد تم تمثيل الخرسانة باستخدام عناصر صلبة تأخذ بنظر الاعتبار تأثير التصدعات و باقي الخواص اللاخطية . كما تم استعمال عناصر ربط لتمثيل حديد التسليح وقد أظهرت النتائج المستخرجة من التحليل العددي تقارب جيد مع النتائج العملية.

1. Introduction:

A "**flat slab**" is a reinforced concrete floor constructed without beams or girder and supported by columns that may flare out at tops to form conical capitals, and when no drop panel or only a negligible one is provided, the floor becomes a "**flat plate". [1]**

Concrete flat slab floors provide an elegant form of construction, which simplifies and speeds up site operations, allows easy and flexible partition of space and reduces the overall height of buildings **[2].**

Any attempts for engineering analysis can be done conveniently and fast using versatile FEA packages. These result in the modernization of structural modeling by new generation practical engineers, in order to verify their structural designs. Nonlinear material models have been integrated in many general purpose FE codes.

The objective of this study was to develop a numerical model where material properties of two way RC panels can be included in detailed analyses for static investigations in future.

Due to its availability to inclustry, a commercial FEA package is preferably in order to use the model in general design practice. A three dimensional nonlinear FE model of two way RC panels was developed by general purpose FEA package ANSYS 9.

This paper presents 3D nonlinear finite element (FE) model, developed using commercial finite element program , ANSYS 9, to simulate the behavior of reinforced concrete slab specimens in both punching and flexural failure of normal strength concrete (NSC), high strength concrete (HSC) and light weight concrete (LWC) under concentrated load tested by Muhammed 2007 **[3].** The results obtained from the FE analysis are compared with experimental data of for the RC slabs. The numerical results are presented in terms of ultimate load carrying capacity and deformational characteristics. The comparisons between the FE results and the experimental data demonstrate the accuracy and validity of the FE model. The FE modeling of the problem can serve as numerical platform for performance prediction of RC slabs.

2. Case Study :

The case study is defined by tested specimen shown in **figure (1),** it is a reinforced concrete slab casted in laboratory. Its dimensions are (450 × 450 × 50 mm) (length × width × thickness) , Three concrete types were used in this study: normal strength concrete, high strength concrete, and lightweight concrete, with compressive strengths (f'_c) of (40, 75, 29.34 N/mm^2) and dry unit weights (w_c) of (2360, 2432, 1808 kg/m^3) respectively. They were tested to failure under a concentrated load through a central column of dimension 30 × 30 mm. The slabs were simply supported along the four edges with a clear span of 420 mm in each direction, corner up lifts were prevented by placing loads at corners .

Plain wires 6 mm in diameter were used as flexural reinforcement placed in the tension face of the slab, the average yield strength was 283 N/mm^2. The wires were uniformly spaced and placed in two directions at 150 mm c/c and 300 mm c/c spacing each way to obtain the desired steel ratios of (0.005 and 0.002) for punching and flexural tests respectively. A clear cover of 15 mm was provided for the mesh.

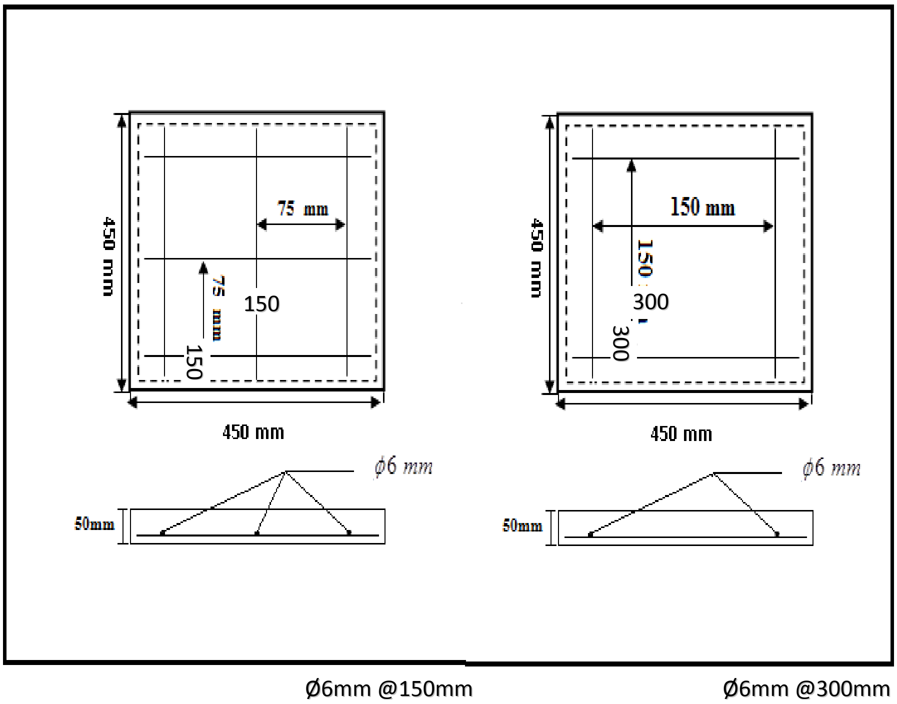

Figure (1): Dimensions and Reinforcement Details of Slabs [3]

3. Finite Element Model:

3-1 Finite Element Model of Concrete:

The three dimensional 8-node brick element (Solid 65) is used for model of concrete. The element has eight corner nodes, and each node has three degree of freedom (translation in the X, Y and Z direction). The geometry and node locations for this element type are shown in **Figure (2).** The concrete is assumed to be homogeneous and initially isotropic.

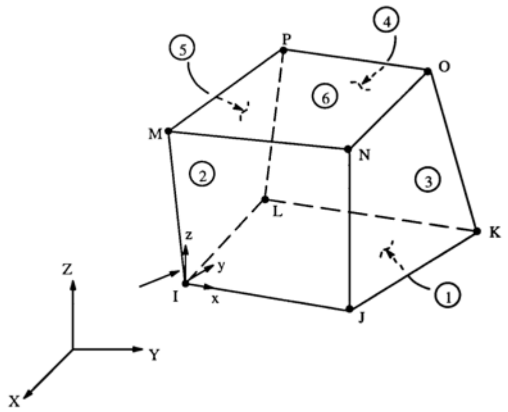

Figure (2): Solid45 – 3-D solid [4]

The ANSYS program requires the uniaxial stress-strain relationship for concrete in compression. Numerical expressions (Desayi and Krishnan 1964) [5], defined by Equations 1 and 2 were used along with Equation 3 to construct the uniaxial compressive stress-strain curve for concrete. **Figure (3)** shows a typical stress-strain curve for normal weight concrete. The multi-linear curves were used to help with convergence of the nonlinear solution algorithm.

$$f_c = \frac{\varepsilon E_c}{1+\left(\frac{\varepsilon}{\varepsilon_o}\right)^2} \text{ for } \varepsilon_1 \leq \varepsilon \leq \varepsilon_o \ \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \ (1)$$

$$\varepsilon_o = \frac{2f'_c}{E_c} \ \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \ (2)$$

$$E_c = \frac{\sigma}{\varepsilon} \text{ For } 0 \leq \varepsilon \leq \varepsilon_1 \ \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \ (3)$$

Where:

σ = stress at any strain ε, N/mm^2.

ε = strain at stress f .

ε_o = strain at the ultimate compressive strength f'_c .

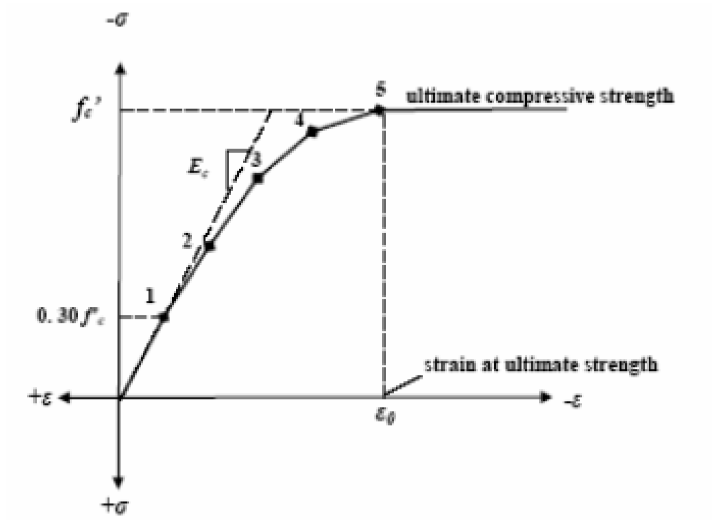

Figure (3): Simplified Compressive Uniaxial Stress- Strain Curve for Concrete [6]

The concrete cylindrical compressive strength $f\acute{c}$ was taken to be 0.8fcu , the modulus of elasticity, E_c, can be calculated with a reasonable accuracy for values of weight (w_c) between (1440 -2480) kg/m^3 from the empirical formula (ACI code 318-2005) **[7].**

$$E_c = 0.043 (w_c)^{1.5} (f'_c)^{0.5}$$

in which, w_c is the air-dry unit weight of concrete in kg/m^3, f'_c is the cylinder compressive strength of concrete in N/mm^2 and E_c is the modulus of elasticity of concrete in N/mm^2.

The ultimate uniaxial tensile strength (modulus of rupture, f_r) is taken as ($0.62\sqrt{f'_c}$) for all cases to estimate the ability of concrete to transfer shear force across the crack interface, a shear transfer coefficient (β) is introduced which represents a shear strength reduction factor for concrete across the crack face **[4].**The shear transfer coefficient used in this study is equal to 0.25 for the case of open crack (β_o) and 0.9 for the case of closed crack (β_c).

3-2 Finite Element Model of steel reinforcement:

Modeling of reinforcing steel in finite elements is much simpler than the modeling of concrete. Link8 element was used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degree of freedom- translations in nodal x, y and z directions. This element is also capable of plastic deformation. A perfect bond between the concrete and steel reinforcement is considered. The geometry and node locations for this element type are shown in **Figure 4.**

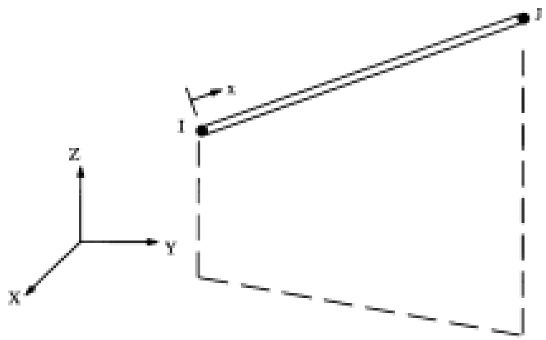

Figure (4): Link8 – 3-D spar [4]

In the present study the steel reinforcement was connected between nodes of each adjacent concrete solid element, so the two materials shared the same nodes. the steel reinforcement for the FE model is assumed to be an elastic- perfectly plastic and identical in tension and compression as shown in **figure (5).** Material properties for the steel reinforcement for all six models are as follows:

Elastic modulus, E_s= 200,000 MPa, Yield stress, f_y= 283MPa & Poisson's ratio ν = 0.3.

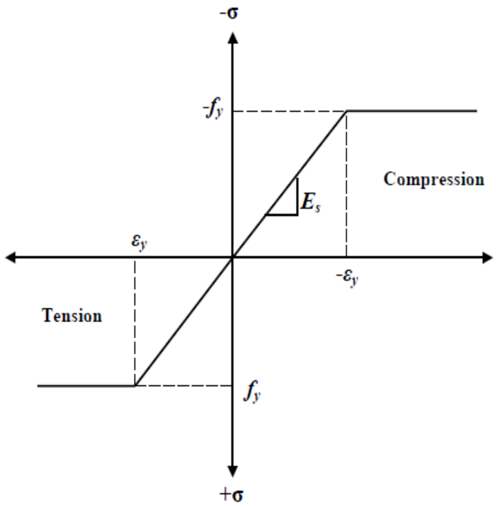

Figure (5): Stress-strain curve for steel reinforcement

4. Finite Element Idealization:

The reinforced concrete slabs are modeled by (3600) 8-node brick elements to represent the concrete material. The reinforcement bars are modeled by using (120) and (270) three-dimensional two-node bar elements (link8) for flexural and punching specimens respectively, which share with concrete elements in the same nodes. The external load is represented by equivalent nodal loads.

The finite element mesh, boundary conditions and loading of reinforced concrete slab specimens for flexural and punching are shown in **Figure (6)** and **Figure (7).**

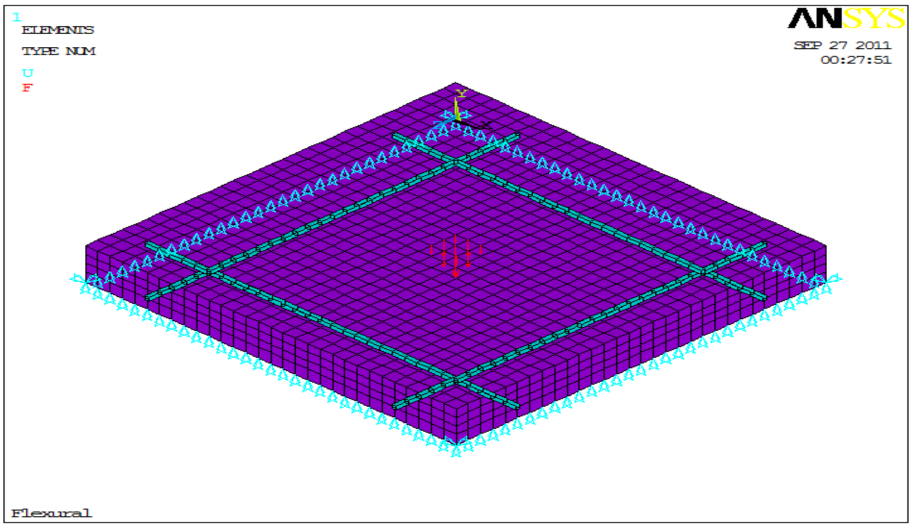

Figure (6): FE mesh, boundary condition and loading for flexural specimen slab

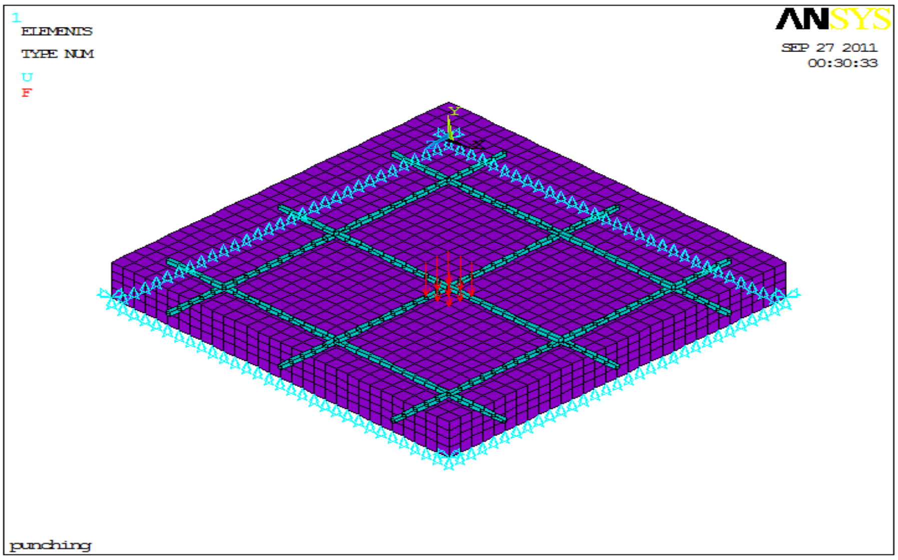

Figure (7): FE mesh, boundary condition and loading for punching specimen slab

Generally, the calculation was conducted by using 8-point $(2 \times 2 \times 2)$ integration rule, and full Newton-Raphson method to carry out the nonlinear analysis. A convergence tolerance of (1%) is used. Uniform increments of load have been used for applying the external load.

5. Results And Discussion:

The validation of the FE models was conducted by comparing the load carrying capacity and load-deflection response of the experimental model 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. **Figure (8)** shows the load-deflection plots of reinforced concrete slab specimens . In general, the load-deflection plots for the slabs from the finite element analyses agree well with the experimental data. The finite element load-deflection plots in the linear range are somewhat stiffer than the experimental plots. After first cracking, the stiffness of the finite element models is again higher than that of the experimental slabs. There are several effects that may cause the higher stiffness's in the finite element models. First, the finite element models do not include the micro cracks. The micro cracks reduce the stiffness of the experimental slab. Next, perfect bond between the concrete and steel reinforcing is assumed in the finite element analyses, but the assumption would not be true for the experimental slabs. As bond slip occurs, the composite action between the concrete and steel reinforcing is lost. Thus, the

overall stiffness of the experimental slabs is expected to be lower than for the finite element models. A compression between the crack pattern of F.E. and experimental is shown in the **figure (9).**

Table (1) shows comparisons between the ultimate loads of the experimental and the final loads from the finite element models. The final loads for the finite element models are the last applied load steps before the solution diverges due to numerous cracks and large deflections.

Table (1): Comparisons between experimental ultimate loads and FE final loads

	Specimen	Failure Load		(Exp/FE)
		Exp.	FE	
Punching	NSC1	18	18	1
	HSC1	26	26	1
	LWC1	13	14	.93
Flexural	NSC2	11	12	.92
	HSC2	18	18	1
	LWC2	9	8	1.125

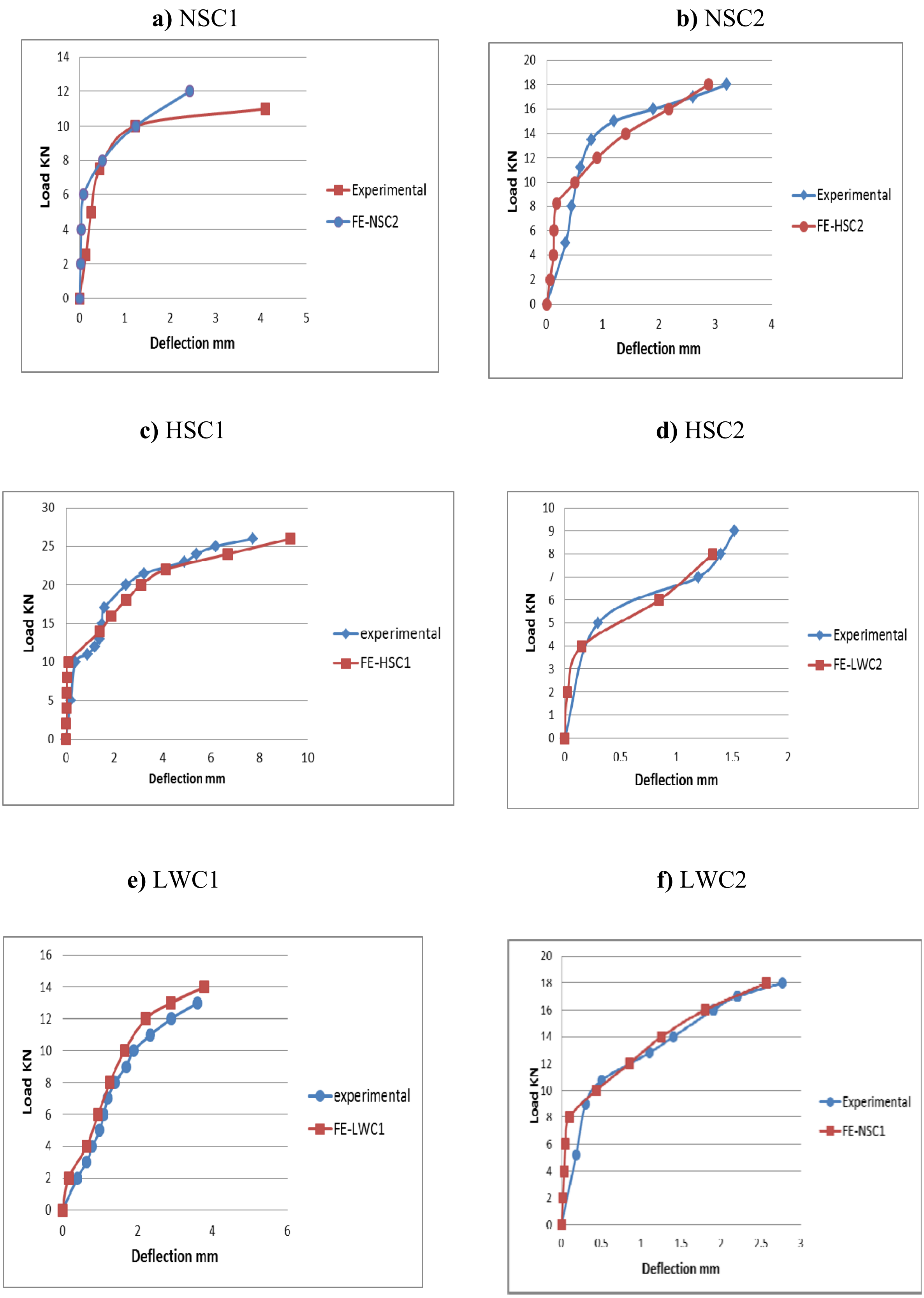

Figure (8): comparison between experimental and FE model

6. Conclusions:

This paper has presented the FEA of the two way RC panels with different concrete type (NSC, HSC and LWC) and two type of failure (flexural and punching). The following finding are drawn from this work:

1. The load _deflection relationships for the FE model have good agreements with experimental results.
2. Nonlinear FEA predicted the failure types (flexural and punching) and the ultimate load capacity.
3. The nonlinear FEA model was able to simulate the RC panel's behavior for each concrete type.
4. The present FE model can be used in additional studies to develop design rules for two way RC panels.

7. Reference:

1. Phan, L.T.; and Carino, N.J.,**"Review of Mechanical Properties of HSC at Elevated Temperature"**, Journal of Materials in Civil Engineering, Feb.1998, pp.58-64.
2. Kheyroddin A, HoseiniVaezS.R. and Naderpour H.,.**"Numerical Analysis of Slab-Column Connections Strengthened with Carbon Fiber Reinforced Polymers"**. Journal of Applied Sciences, 2008 8: 420-431.
3. Muhammed N. J. **" Residual Strengths of Reinforced Normal Strength, High Strength and Lightweight Concrete Panels Exposed To High Temperatures"**, MSc thesis, civil engineering dept., college of engineering, AL Mustansirya University, May 2007.
4. "**ANSYS Manual**", Version 9.0, U.S.A. 2004
5. **Desayi, P. and Krishnan, S., "Equation for the Stress-Strain Curve of Concrete,"** Journal of the American Concrete Institute, 61, pp. 345-350, March 1964.
6. Bangash, M. Y. H., **"Concrete and Concrete Structures: Numerical Modeling and Applications"**, Elsevier Science Publishers Ltd., London, England, 1989.
7. ACI Committee 318,**"Building Code Requirements for Structural concrete (ACI 318-2005, ACI 318R-2005)"**, American Concrete Institute, 2005.