

Non Linear Finite Element Modeling of Reinforced Concrete Flat Plate Slabs with openings

● **Dr. Serwan Kh.Rafiq** - lecturer ●

Department of Civil Engineering – College of Engineering - University of Sulaimani

Received : 01/07/2016

Accepted : 09/10/2016

DOI Link: <https://doi.org/10.17656/sjes.10044>

Abstract



In this study, a model using ABAQUS/ EXPLICIT program to predict punching shear resistance of reinforced concrete flat plates was proposed with opening in different situation subjected to vertical loading. A three dimensional non-linear finite element program (ABAQUS/EXPLICIT) based on Eight-Nodes hexahedral element damage-plasticity was utilized for modeling concrete with opening, in this investigation. Reinforcing bars were represented by one dimensional element embedded in the solid elements and for both tension and compression linear elastic-plastic behavior is assumed. Nonlinear analysis is used and the result of finite element shows good agreement with the experimental results.

Keywords: Finite Element, Flat Plate Slabs, Abaqus/Explicit.

1. Introduction

Reinforced concrete flat slab has commonly used in constructions, as it is providing numbers of benefit to the designer. Flat Slab is faster, cheaper and simple formwork Comparing to the structures with beam column construction^[1]. Nonlinear analysis programs have become increasingly popular in recent years, as engineers attempt to more realistically model the behavior of structures subjected to all types of loading^[2].

Many tests have been done to study the punching shear failure in flat reinforced concrete slabs^[3-6]. The failure in flat slabs without shear reinforcement is brittle, the cracks propagate inside the slab^[7]. The need for more testing, which requires time, space and financial supports, makes the numerical solutions and finite element simulations in dispensable.

Finite Element (FE) simulations can give an insight into punching shear failure and supplement the test results. These FE models

must be, however, calibrated based on selected experimental results. These calibrated models can be then used for parametric investigations addressing many and different aspects of punching shear.^[7]

The development of numerical analytical method, such as finite element method, permit realistic evaluation of internal stresses and displacements on which the limit requirements may be based on improved predicted structural response. Furthermore, such refined analytical solutions help in understanding and interpreting the observed behavior of structural members.

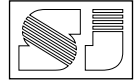
2. Research Significance

Evaluation and Modeling the punching shear behavior for reinforced concrete flat plate slabs with opening at different locations using (ABAQUS/EXPLICIT) software and to propose predicting equations for punching shear resistance.

3. Description Of the Model

3.1. Configuration and Properties

The test slabs configuration and geometry are adapted from the experiment work that conducted by Borges et al^[8]. The selected experimental work in this study are slabs with opening (No. 4 and 7), were 200 mm thick and 3.0 m square on plan of typical details shown in Fig. 1. At their centers, they were supported by a thick steel plate 200 x 600 mm on plan. The flexural reinforcement of the slabs was 16 mm bars at 100 mm centers both ways in the top, extending full width, with 12.5 mm hairpin (U-type) bars extending 675 mm horizontally in the top and bottom at the slab edges. At the bottom of all the slabs were 8 mm bars at 150 mm spacing both ways, were provided extending full width. A top clear cover of 20 mm and bottom clear cover of 25 mm to the outermost reinforcing steel bars or



shear reinforcement were adopted, and the measured stress-strain properties are shown in Fig. 2. Where shear reinforcement was used, it was in the form of stud rails. The studs were 8 or 10 mm diameter deformed bars welded to a 40 x 10 mm plate rail at the bottom and a 40 x 40 mm plates at the top.

A typical stud rail is shown in Fig. 3 in relation to the flexural reinforcement, together with an overview of the test setup. The yield strengths of the shear reinforcement were 598 MPa for the 8.0 mm bars and 593 MPa for the 10.0 mm bars. Slab 4 had a single opening 200 x 300 mm at one short side of the support, while Slab 7 had pairs of symmetrically 200 x 300 mm openings one at each short side of the plate. Detail of the flexural reinforcement near the openings was varied in these tests. Slab 4 contains main bars were interrupted by openings; the steel area was made up by additional full-length bars near the openings. Slab 7, without continuous bars were added, instead 12.5 mm U-bars were placed. Adjacent to the cut bars, some anchorage for horizontal tension and some shear reinforcement at the edges of the openings was provided. For slab 4, both additional continuous and U-bars were provided. Figure 4 shows the relevant details for Slabs 4 contained shear reinforcement and additional full-length bars near the openings. Each slab had two symmetrical openings (200 x 300) mm as in Slab 7; the main steel had the same details as in Slab 7.

3.2. Modeling the structure by ABAQUS program

In order to verify that the developed program is applicable to the reinforced concrete flat plate slabs with opening, analysis of two flat plates is carried out. One of them has single opening (slab No. 4) and the other has a pair of openings (slab No. 7). Eight-Noded hexahedral brick element is utilized to model the concrete. Modeling of reinforcing bars is much simpler than the modeling of concrete; an elasto-plastic truss element is used to model it. In the modeling, perfect bond is assumed between the concrete and reinforcement, within the embedded region, which shares the same node in both concrete and reinforcement, is used for the contact between the node of reinforcement and the node of concrete. The modeling process including geometry of the model, loading and columns' boundary condition, analysis type, material data, element section properties and meshing effects and output request were addressed.

3.3. Geometry of the model

A three dimensional (3D) finite element model has been simulated for full scale reinforced concrete flat plates with opening. Test data by Liana et al^[8] on reinforced concrete flat plates with opening in different situation subjected to vertical loading is utilized. Two specimens were used (slabs 4 and 7). The slab dimension and thickness for S7 slabs are 3000 mm square and 200mm, respectively, with two openings. The slab dimensions and thickness for those designated as S4 are 3000 mm square and 200 mm, respectively, with one opening. A 200mm x 600 mm isolated column was used in the middle of the slab to represent the internal slab-column connection. The extending column s height below and above the slab is 1000mm as shown in Fig. 5.

3.4. Columns' boundary condition

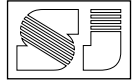
According to the tests, only the top and bottom of the column were restrained; at the bottom only the face rotation was allowed. Thus, modeling this condition using ABAQUS software is not easy. To solve this problem plate with the same column dimensions was created with a thickness of 40mm, which is depicted in Fig. 5.

3.5. Effect of Mesh Refinement

According to the finite element analysis theory, the finite element (FE) models with fine mesh (small element size) yields highly accurate results while may take longer computing time. On the other hand, those FE models with coarse mesh (large element size) may lead to less accurate results but smaller computing time^[9]. ABAQUS software chose the best type of meshing by default, Table 1, shows the data from the program. Generally, using a finer or courser mesh does not have a big impact refining mesh and lowering time step size increase accuracy. The accuracy that can be obtained from any FEA model is directly related to the finite element mesh that is used in the analysis; however, it is obvious that accurate results will be achieved by selecting a finer mesh, but this is time-consuming and not practical. Fig. 6 below, shows the mesh of slab

3.6. Reinforcement data properties

To model slab, column and stirrup reinforcement in ABAQUS software, a wire was chosen which has one dimension. To create the interaction between steel and concrete an embedded region was chosen for this model. Since



Thus, the principle equivalent tensile strains in the concrete can be used to track the crack formation in the finite element simulation. There are several parameters that might cause the higher stiffness in the finite element model. Micro crack is ignored in the finite element models. In addition, perfect bond is assumed between concrete and reinforcement, whereas this assumption is not correct for experimental slabs. This is partly because bond slip happens and the composite actions between concrete and reinforcement will occur. Therefore, the overall stiffness of the experimental slabs is lower than that given by the finite element model.

4.2- Effect of mesh size Slab4

To understand the effect of finer or courser mesh on the outcome of the analysis different mesh elements were chosen, and eventually the best results, which were close to experimental results, was chosen. The numerical results of analysis based on these meshes have been compared with the experimental results. Fig. 14 reveals that the finite element solution obtained using (20 x20) mm element is in a good agreement with the experimental moment-curvature curve. A relatively stiffer response has been observed using (40 x40) mm. Therefore the rest of the analyses have been carried out using 20 elements.

4.3- Concrete compressive strength

In this part, two specimens were simulated in cases of vertical loading; the main difference is the concrete strength. The results of ABAQUS were validated against the tests. According to the tests, concrete strength has a great influence on the stiffness. It is believed that slab stiffness increases when concrete strength increases. However, the ratio of increase is almost smaller than the root square of concrete. The same result was achieved using ABAQUS to model the specimen. It is clear from Figure 15 and 16 below that increasing the compressive strength makes the slab stiffer.

For Slab 4, the values of the predicted ultimate load were 625 ,678 , 744, 728, 821, and 906 kN for f_c' equal 30, 35, 41.4,50 and 60 MPa, respectively. The ratios of the predicted ultimate load to the value of the experimental ultimate load were 1.13and 1.24 for f_c' equal 50 and 60.

For Slab 7, the values of the predicted ultimate load were 581, 634 , 653, 681, 771, and 847 kN for f_c' equal 30, 35, 41.4,50 and 60 MPa, respectively. The ratios of the predicted ultimate

load to the value of the experimental ultimate load were 1.125and 1.236 for f_c' equal 50 and 60.

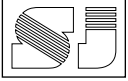
5. Conclusions

From the discussion of the results obtained from the numerical modeling, the following conclusions can be drawn:

- 1- Proposed ABAQUS models which adopted in the present work is suitable to predict the behavior of the reinforced concrete flat plate slabs. The numerical results were in good agreement with experimental load-deflection curve throughout the entire range of behavior.
- 2- The proposed tension-stiffening model, which was used to simulate the post-cracking behavior of reinforced concrete flat plate slabs, shows a good agreement with the experimental post-cracking response. The results obtained from finite element analysis are close with respect to the experimental data.
- 3- The proposed nonlinear finite element models predicted the failure type and the ultimate load capacity for reinforced concrete flat plate slabs with openings.
- 4- Concrete strength has a great influence on stiffness. Slab stiffness increases when concrete strength increases. The predicted ultimate load capacity is significant affected by increasing the value of the concrete compressive strength f_c' , The ratios of the predicted ultimate load to the value of the experimental ultimate load was increased 13 % and 24% for slab 4 and 12.5 % ,13.6% for slab 7 respectively, when compressive strength f_c' increased to 50 and 60 MPa . However, the ratio of increase is almost smaller than the root square of concrete compressive strength.

References

- 1- Sanjay P. N. , Mahesh P. K. , Umesh S. S. “ Behaviour of Flat Slab Rcc Structure Under Earthquake Loading, ” International Journal of Engineering Research & Technology Vol. 3, Issue 5, May - 2014.pp. 1386-1395.
- 2- Hueste M. B. D. and Wight J. K., “Nonlinear punching shear failure model for interior slab-column connections,” Journal of Structural Engineering, vol. 125, No. 9, 1999, pp. 997-1008.



- 3- Gomes, R. B., and Andrade, M. A. S., "Punching in Reinforced Concrete Flat Slabs with Openings," Developments in Computer Aided Design and Modeling for Structural Engineering, Civil-Comp Press, Edinburgh, UK, 1995, pp. 185-193.
- 4- Regan, P. E., "Shear Reinforcement of Flat Slabs," TRITA-BKN Bulletin 57, International Workshop Punching Shear Capacity of RC Slabs, Royal Institute of Technology, Sweden, 2000, pp. 97-107.
- 5- Kuang, L., and Teng, S., "Punching Shear Strength of Slabs with Openings and Supported on Rectangular Columns," School of Civil Engineering, Nanyang Technological University, Singapore, 2001, pp. 298 .
- 6- Teng, S., Cheong, H.,K.,and Kuang, K. L., and Geng, J.Z. " Punching Shear Strength of Slabs with Openings and Supported on Rectangular Columns," ACI Structural Journal, Vol. 101, No. 5, Sept.-Oct. 2004, pp. 678-687.
- 7- Genikomsou A.S. and Polak M.A. "Finite Element Analysis of a Reinforced Concrete Slab-Column Connection using ABAQUS", Structures Congress 2014 © ASCE, 2014, pp. 813-823.
- 8- Borges L.L.J., Melo G.S.,and Gormes R.B. "Punching Shear of Reinforced Concrete Flat Plates with Openings" ACI Structural Journal, Vol. 110, No. 4, July-August,2013, pp.547-556.
- 9- More S. T., Dr. Bindu R. S." Effect of Mesh Size on Finite Element Analysis of Plate Structure" International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 4, Issue 3, May 2015 pp 181-185.
- 10- Lubliner J.,Oliver J.,Oller S. and Onate E." A PLASTIC-DAMAGE MODEL FOR CONCRETE" International Journal of Solids and Structures Vol. 25, No. 3, 1989, pp. 299-326.
- 11- ABAQUS User's Manual, Version 6.8, Hibbitt, Karls-son & Sorensen, Inc., Pawtucket, Rhode Island, USA. (2008).

التحليل اللاخطي للعناصر المحددة لنمذجة البلاطات الخرسانية المسلحة المسطحة والحاوية على الفتحات

د . سيروان خورشيد رفيق - مدرس
(قسم الهندسة المدنية - جامعة السليمانية)

المستخلص :

تم في هذا البحث اقتراح نموذج باستخدام برنامج التحليل اللاخطي ABAQUS / Explicit للتنبؤ بمقاومة القص الثاقب للبلاطات الخرسانية المسلحة المسطحة والحاوية على الفتحات في أماكن مختلفة والمعرضة للتحميل الرأسي . تم استخدام برنامج ثلاثي الأبعاد الغير الخطية محدودة العناصر ABAQUS / Explicit على أساس شرائح سدسة الأضلاع ثمانية العقد ، لقد تم تمثيل حديد التسليح بعنصر احادي الأبعاد كجزء لا يتجزأ من العناصر الصلبة ولكل من اجهادات الشد والضغط ، وتم افتراض السلوك الخطي المرن ، بعدها تم استخدام كل من التحليل الخطي وغير الخطي في هذا البحث ، ونتيجة للمقارنة مع النتائج العملية اظهر المقترح تطابقا جيدا مع النتائج التجريبية للدراسات السابقة.

الكلمات المفتاحية : طريقة العناصر المحددة ، البلاطات الخرسانية المسلحة المسطحة ، ABAQUS/ Explicit .

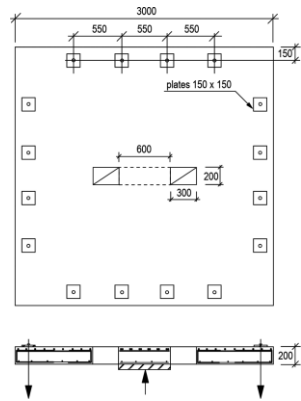
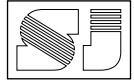


Fig. (1) : Typical detail of the tested slabs (Slab 7). (Dimensions are in mm) [8].

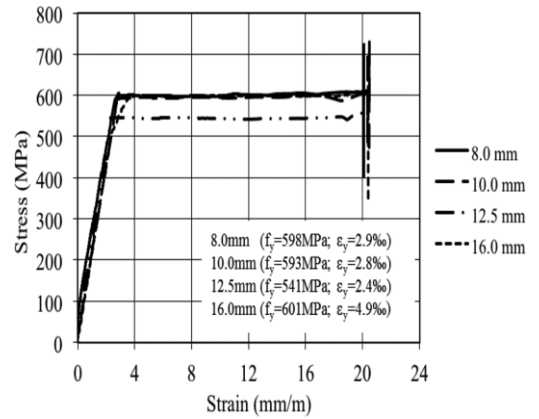
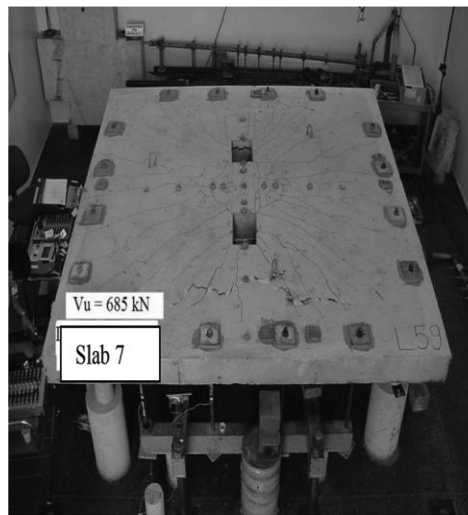
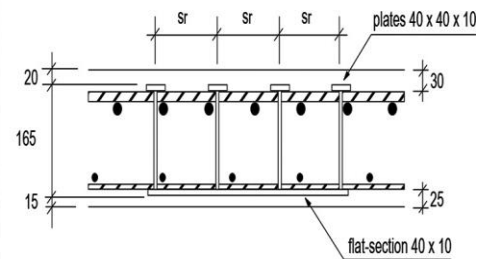


Fig. (2) - Measured stress-strain Properties for steel bars (8, 10, 12.5, and 16 mm) [8].

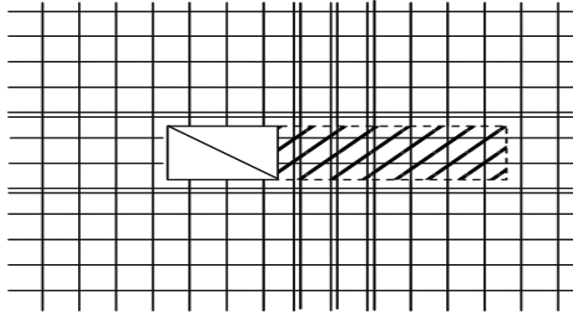
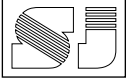


a. Specimens setup



b. Stud rail

Fig. (3) : Test setup (left) and typical detail of stud rail. (Dimensions are in mm) [8].



Slab 4

Fig. (4) : Details of flexural reinforcement near openings for Slabs 4^[9].

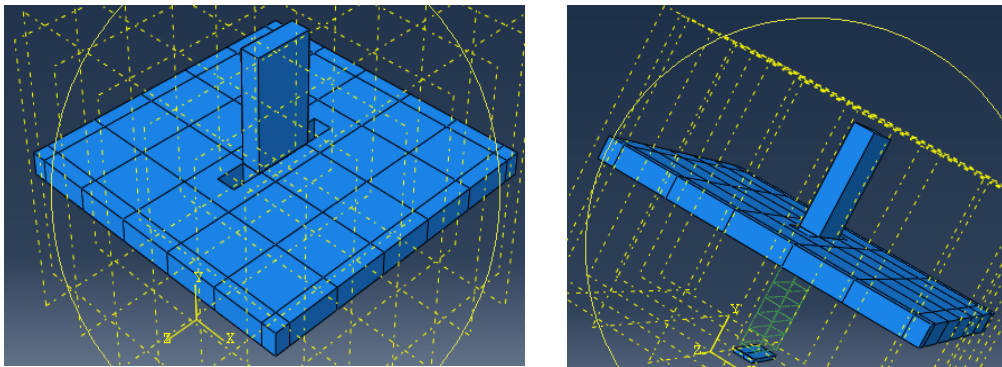
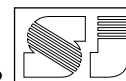
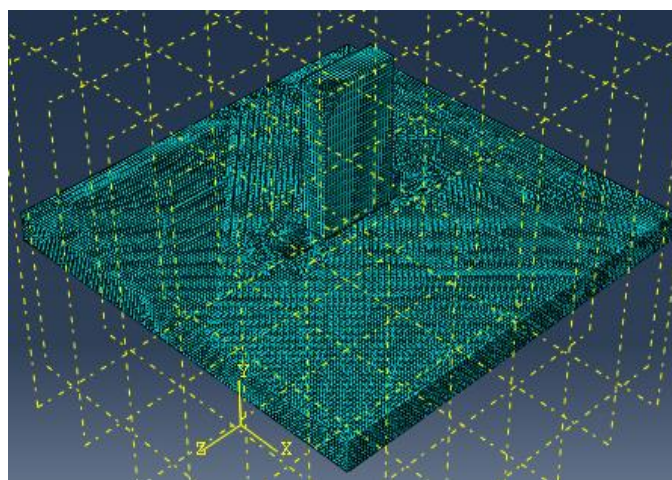


Fig. (5) : Geometry of the Model of Slabs 7.

**Table (1) : Data from ABAQUS Program.**

Mesh Size(mm)	Node Number	Element Number	Linear Hexahedra Element C3 DR8	Linear Element of Type T3D2
40mm	50942	42622	32590	10032
35mm	72574	61846	50316	11530
30mm	106962	92617	79213	13404
25mm	162460	142074	130544	11530
20mm	312585	280406	260350	20056

**Fig. (6) : Meshing of the Model Slabs 7.**

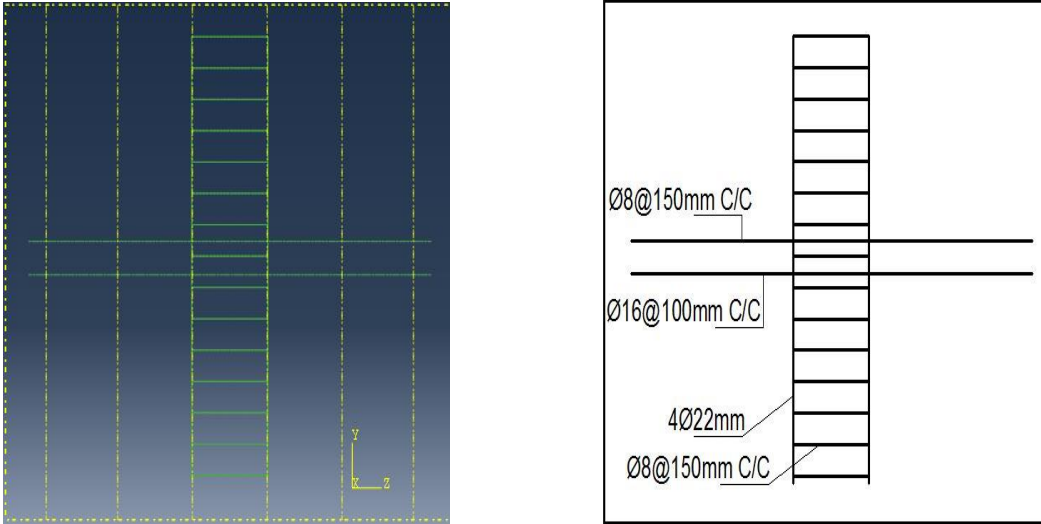
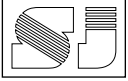


Fig. (7) : Reinforcement of the Model Slabs 7.

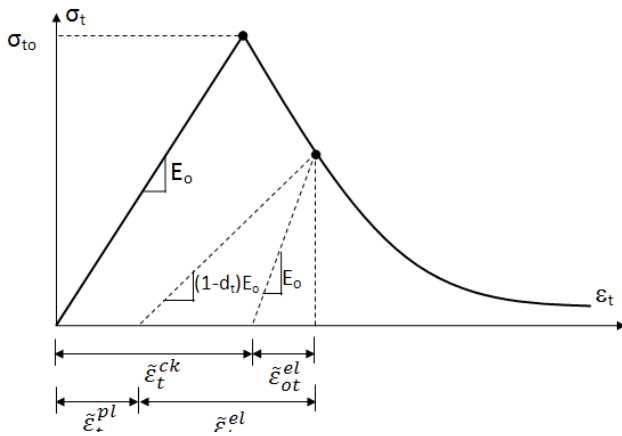


Fig. (8) : Tension stiffening adapted from ABAQUS [11].

$$\varepsilon^{pl} = \varepsilon_t^{ck} - \left(\frac{\sigma_t}{E_o} \frac{d_t}{1 + d_t} \right) \quad (1)$$

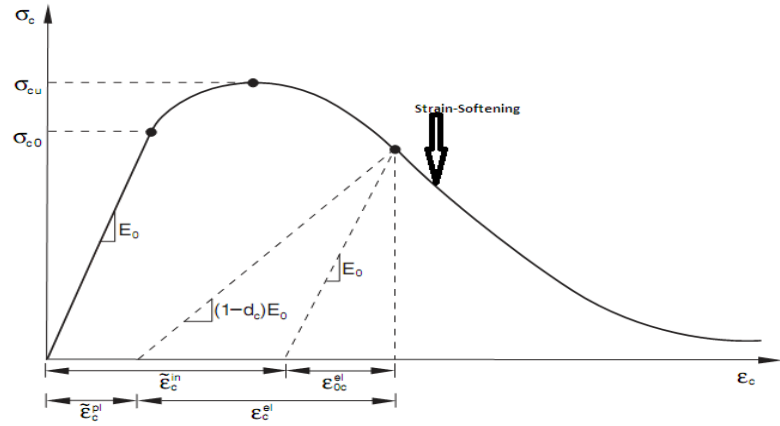
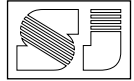


Fig. (9) : Compressive behavior associated with compression hardening ^[11].

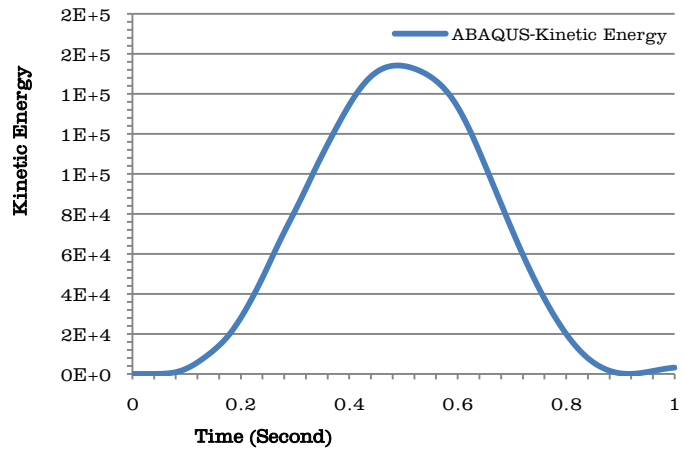


Fig. (10) : relationship between kinetic Energy and time.

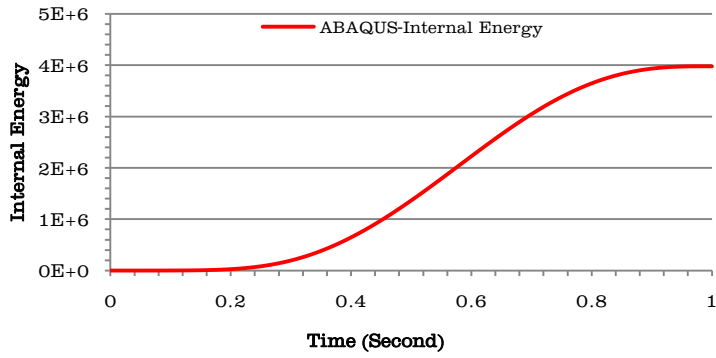
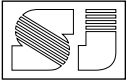


Fig. (11) : relationship between internal energy and time.

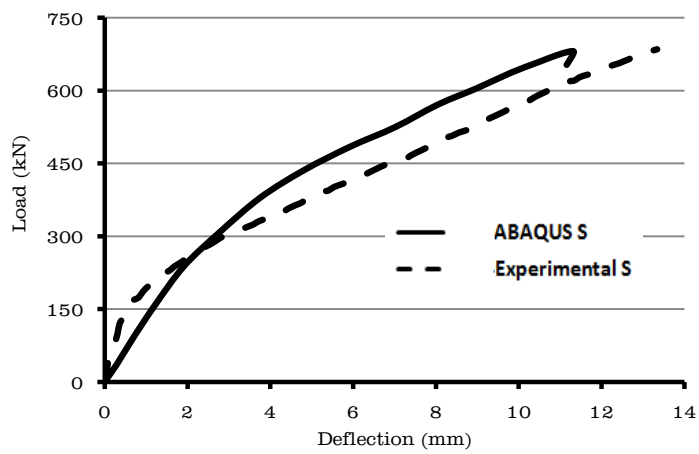


Fig. (12) : Load Deflection relationship for Slab 7.

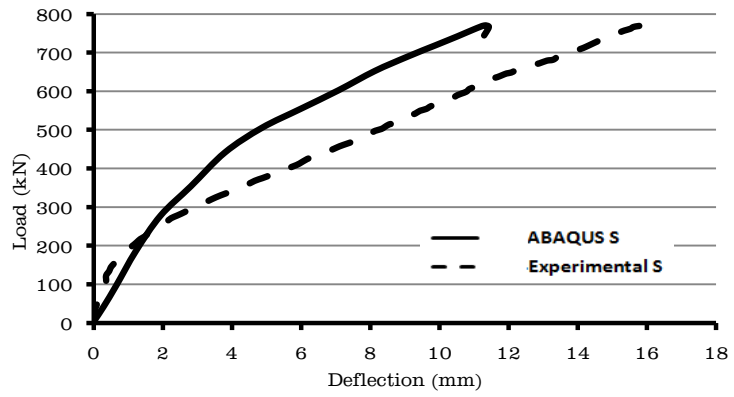
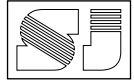


Fig. (13) : Load Deflection relationship for Slab 4.

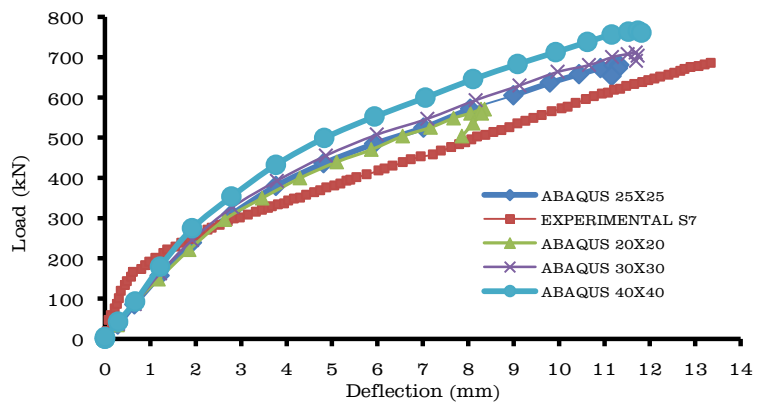


Fig. (14) : Effect of mesh type for slab S7.

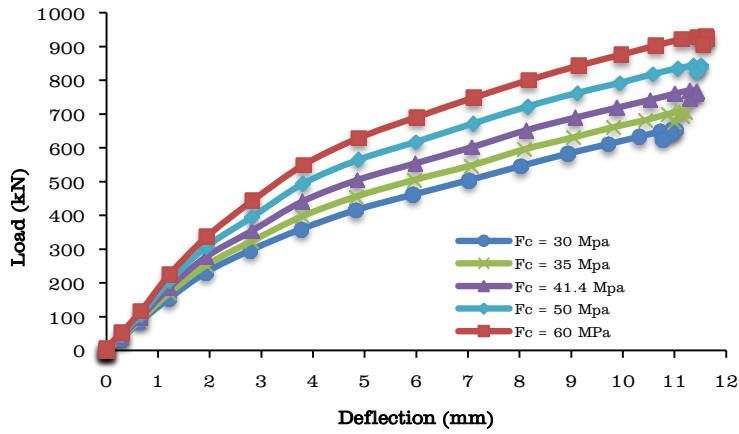
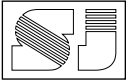


Fig. (15) : Effect of different concrete compressive strength for slab S4.

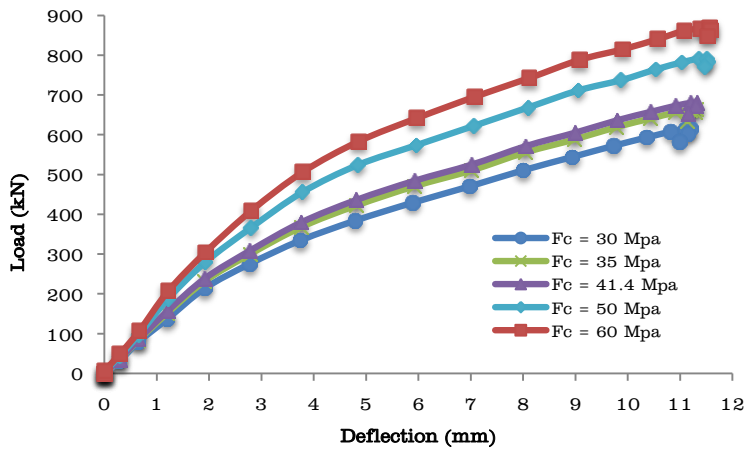


Fig. (16) : Effect of different concrete compressive strength for slab S7.