Journal of Engineering and Sustainable Development
Vol. 23, No.03, May 2019
ISSN 2520-0917
https://doi.org/10.31272/jeasd.23.3.7

VALIDATION OF FINITE ELEMENT MODELING FOR RECTANGULAR REINFORCED CONCRETE BEAMS WITH WEB OPENINGS

Dr. Ahlam Sader Mohammed

Lecturer, Building and Construction Engineering Department, University of Technology, Baghdad, Iraq

Abstract: The present paper aims to validate ABAQUS finite element model to analysis the rectangular reinforced concrete beams in order to study web opening effect on the shear, and ultimate strength of these beams. The validation has been achieved through comparison of load-- deflection curves obtained from the finite element model with those of experimental works of another researcher. The dimensions and flexural reinforcement of the beams were kept the same. The investigated test parameters were the size and the location of the openings. Comparison between load-deflection curves obtained from finite element model with those of experimental works indicates good agreement such that the proposed model can be adopted in future for further studies.

Key Words*: Rectangular Reinforced Concrete Beams, Web Openings, Shear Strength, Finite Element Modeling, ABAQUS.*

شرعنة تمثيل العناصر المحددة للعتبات الخرسانية المستطيلة المسلحة الحاوية على فتحات الوترة

الخلاصة: تهدف الدراسة الحالية إلى اجراء دراسة مقارنة لتحليل عتبات خرسانية مسلحة مستطيلة باستخدام برنامج أباكوس من أجل دراسة تأثير وجود فتحات على قابلية تحمل هذه العتبات لقوى القص وقوة المقاومة القصوى. وقد تحققت عملية التحقق من خلال المقارنة بين منحنيات الحمل والهطول التي تم الحصول عليها من نموذج العناصر المحددة باستخدام برنامج الاباكوس مع النتائج التجريبية لدراسة عملية قام بأجرائها احد الباحثين. ولغرض دراسة تأثير وجود هذه الفتحات، تم تثبيت الابعاد وخواص مادة الكونكريت ومقدار التسليح الداخلي لجميع العتبات المدروسة. تم التركيز على دراسة حجم ومواقع هذه الفتحات في العتبات. وتشير المقارنة بين منحنيات الحمل والانحراف التي تم الحصول عليها من نموذج العناصر المحدودة مع نماذج الأعمال التجريبية إلى توافق جيد بحيث يمكن اعتماد النموذج المقترح في المستقبل لإجراء مزيد من الدراسات.

1. Introduction

Generally, ducts and pipes that interconnect structural beams are important to provide accommodate essential services such as water supply, air-conditionings, sewage, electricity, and computer network [1];

*Corresponding Author : *AhlamSader@ yahoo.com*

Transverse openings inside these pipes and ducts may reduce the overall stiffness of such beams which led to deformation and additional deflection under the service load and considerable distribution of forces and internal moments in a continuous beam [2]. Also, passing utility services through openings in the floor beam webs might minimize the required storey height and encourages the designers to reduce the height of the structure which leads to more cost-effective design. Including transverse openings in the web of a reinforced concrete beam results in unexpected changes in the dimensions of the cross section of the beam and hence, the corners of the opening would be subjected to stress concentration and it is may induce transverse cracks in the beam [3]. So, the effect of openings on the strength and behavior of reinforced concrete beams must be considered and the design of these beams needs special concern.

This study aims to verify an ABAQUS finite element model that proposed to simulate the shear strength for rectangular reinforced concrete beams. Experimental works of Nilesh H. Saksena [4] have been adopted as the reference experimental works for the validation process. The validated finite element model may be adopted later by authors or other researchers to investigate other case studies where experimental work is either not possible or costly to execute.

2. Adopted Experimental Work of Saksena

2.1 Beams Details

As mentioned previously, the experimental results of Nilesh H. Saksena were adopted as a reference experimental works for the verification purpose. Five simply supported rectangular beams with normal concrete were casted and tested to study the effect of the size and location of circular openings on shear and ultimate strength of reinforced concrete beams. The specimens were composed of two groups (I, II) and control beam based on size and location of opening. Group I had different opening at distance L/8 and group II had different opening at distance L/4 and control beam that does not has opening to serve as a control one.

To study the effect of parameters correlated to opening in the beams, the amount of shear and flexural reinforcement, their strength and the stirrups spacing, along all beams were held constant. Beam's length was of (2 m) which were loaded over supports with span length of (1.7 m). The width of section was (150 mm) while the depth of cross-section was (200 mm)), the effective depth was considered as (174 mm) and the expanse of compression fiber to compression reinforcement (24mm). the shear span to effective depth ratio was (a/d= 3.25). The amount of flexural bottom reinforced was 3Φ12 mm in diameter for all specimens (beams) while 2Φ8mm diameter were used for the top reinforcement. M20 grade of concrete andFe415 grade of steel were used.

All beams were tested to failure with a two point flexural test after 28 days curing period. The details of the loading frame are shown in Figure 1 [4].

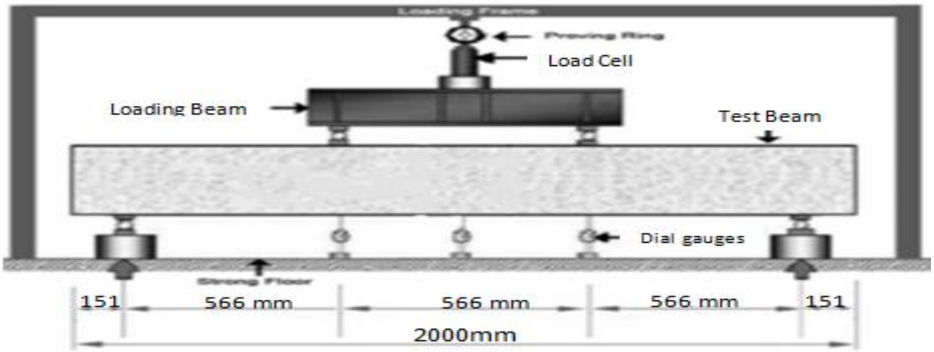


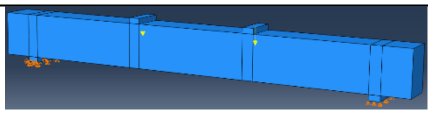
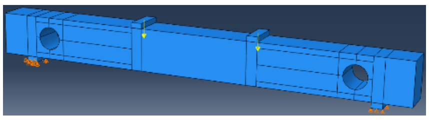
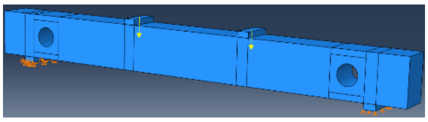
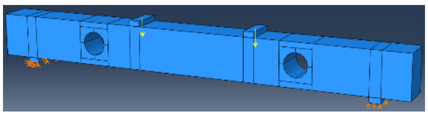
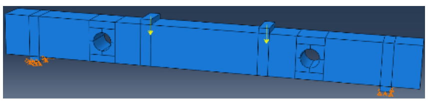
Figure 1: loading frame [4].

2.2 Web Openings

As shown in Figure 2 and Table 1. Beam1 was solid beam (control beam), Beam2 had 110 mm (0.55D) opening at L/8 distance from support, Beam3 had 90 mm (0.45D) opening at L/8 distance from the support, Beam4 had 110 mm opening at L/4 distance, Beam5 had 90 mm opening at L/4 distance from the support.

Figure 2: The beams with openings [4].

Table 1: Details of beams openings.

Beam No.	Beams form	Opening diameter (mm)
B1		No opening
B2		110
B3		90
B4		110
B5		90

3. Finite Element Representation

This paragraph will summarize finite element analytical details for the reinforced concrete beams with and without opening. The well-known ABAQUS program was used for this purpose.

3.1 Solid Modeling, Analysis, and Visualization

Numerical analysis was carried out by using the finite element software ABAQUS numerical program. Many marketable programs exist with finite element analysis capabilities for different engineering discipline. They could solve multiplicity problems from simple linear static analysis to nonlinear transient analysis. A only some of these codes, such as ANSYS and ABAQUS, have extraordinary capabilities for analyzing composite materials and they accept custom, user-programmed constitutive equations and element formulations. Since these software packages not only provide analysis tools, geometric modeling, and visualization of results, but also they can be integrated in the larger design, production, and product life-cycle process.

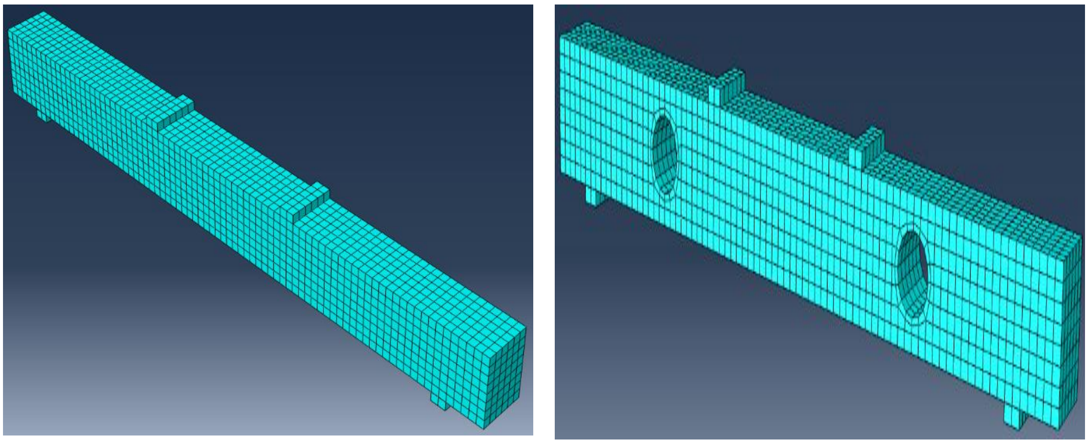
Modern finite element analysis software is commonly organized into three blocks or stages: the preprocessor, the processor, and the post-processor. In the pre-processor stage, the model is built defining the geometry, material properties, and element type. Furthermore, Loads and boundary conditions were entered in the pre-processor as well. This information could also be entered during the solution phase. With these information, the processor block can compute the stiffness matrix and force vector. In the last block, the post, processor–derived results, like stress, strain, and failure loads are computed. The solution can be reviewed using graphic programs [5].

The typical three dimensional stress elements in ABAQUS can be used to model the concrete [6]. These elements are supplied with the appropriate integration system based on the experimental response of the specimen. In present work, the concrete was modeled using C3D8R linear brick 3D finite element. The reinforcing steel bars could be modeled by using solid, beam and truss elements. The use of solid elements is computationally expensive and therefore it was not chosen n the present work. T3D2 two node 3D linear truss element was chosen to represent the internal steel reinforcing bars. The embedded region interaction type had been used in modeling the interaction between reinforcement and concrete while the tie type has been adopted to simulate the interaction between the plates and concrete.

3.2 Modeling of Geometry and Finite Element Discretization

The modeling of geometry was obtained specifying all nodes, their coordinates, and the element connectivity. The connectivity information allows the program to assemble the element stiffness matrix and the vector of element equivalent force to obtain the global equilibrium equations. There are two ways to generate the geometry model; the first one is to create the mesh manually. The second is to use solid modeling, and then mesh the solid to get the node and element distribution. The second approach has been adopted in this study.

In order to obtain reasonable results (outputs) from the finite element model, all of the elements were deliberately assigned the same mesh size to ensure that each two different materials share the same node. The form of mesh selected in the model is structured. The adopted finite element meshes for the beams of the present work is shown in figure 3 for the solid and beams with opening.



(a) Solid beam (b) Beam with openings

Figure 3: Finite element meshes used to simulate of the specimens.

3.3 Material and Section Properties

According to the investigation, material properties could be linear (linear elastic analysis) or nonlinear (e.g. damage mechanics analysis), isotropic or orthotropic, constant or heat-dependent. Entering the materials properties is one of the most important aspects of a successful analysis of composite materials [5].

In the present work, the concrete damage plasticity has been used to represents behavior of the concrete [7]. This approach assumes that the main two failure mechanisms are compressive crushing and tensile cracking of the concrete material. In the linear elastic range the behavior has been distinct by the elastic modulus ($E_c = 4700\sqrt{f_c'}$) which calculated according to ACI 318M- 14 building code [8]. the Poisson's ratio of 0.15 is assumed.

In the plastic range damage parameters, a description of tensile-compressive behavior was requested. The five plastic damage parameters namely the dilation angle of 40, the flow potential eccentricity of 0.1, the ratio of initial biaxial compressive yield stress to initial uniaxial compressive yield stress of 1.16, the ratio of the second stress invariant on the tensile meridian to that on the compressive meridian of 0.667, and the viscosity parameter of 0.0 have been adopted according to the recommendation [6] and [7]. The concrete compressive behavior has been modeled with the well-known stress-strain relationship proposed by EN1992-1-1

Transversal and The longitudinal steel bars have been modeled with a bilinear elastic-perfectly plastic model. In the linear elastic range, the behavior is defined by the young modulus of 200GPa and the Poisson's ratio of 0.3.

3.4 Applied Loads and Boundary Conditions

In structural analysis, loads were defined by forces, pressures, inertia forces (as gravity), and specified displacements, all applied to the model. Loads can be applied on nodes by means of concentrated forces and moments. Also, loads can be distributed over the elements as: surface loads, body loads, inertia loads…….. etc.

A surface load was a distributed load applied over a surface, for example a pressure. A body load was volumetric load, for example expansion of material by temperature increase in structural analysis. Inertia loads are those attributable to the inertia of body, such as gravitational acceleration, angular velocity, and acceleration.

The applied concentrated load on any node was directly added to the force vector. However, element interpolation functions are used to compute the equivalent forces vector due to distributed loads. The applied boundary and loading conditions were shown in figure 4. The analysis process was normally broken down into several steps, each representing different loading and constraint conditions. The minimum number of steps is two: an initial step and at least one additional step. No loads could be applied on the initial step, only boundary conditions [5].

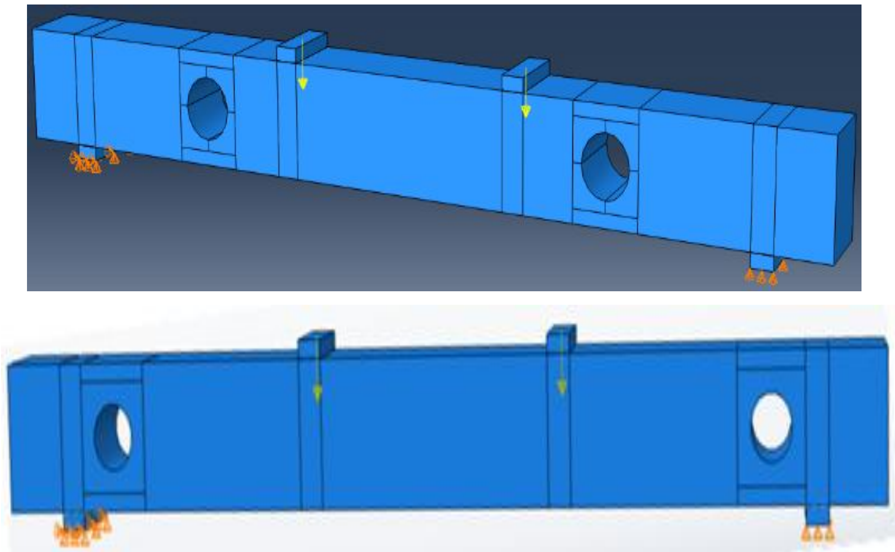


Figure 4: Boundary and loading conditions used in the analysis

4 .Results and Discussion

4.1 Ultimate Load Capacity

Table 2 presents the results of the comparative verification study for the ultimate loads results of the experimental work together with the final loads from the finite element models $(P_u)_{FEM}$.. The results were clear enough to reflect the acceptable agreement between the experimental and analytical data.

The percentage difference for all the analyzed beams in case of ultimate load was less than 10 %. Results of the Table 2 are also presented by bar charts as shown in figure 5 for more illustration.

Table 2: Comparison between the experimental results and finite element analysis results

Beam No.	a/d	Ultimate load (kN)		% difference
		$(P_u)_{FEM}$	$(P_u)_{Exp}$	
B1	3.2	88.89	89	0.12
B2		39.25	35.5	10.56
B3		60.17	60.35	0.27
B4		46.31	46.14	0.37
B5		63.26	67.45	6.21

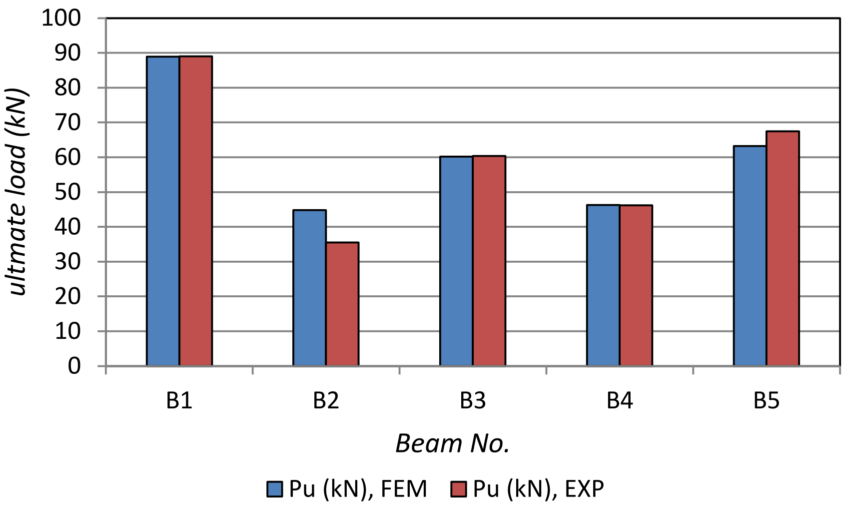


Figure 5: comparative verification study for ultimate loads of the experimental and numerical results

4.2 Load-Midspan Deflection Behavior

Deflection was measured at midspan at the center of the bottom face of the beam. Figure 6 show the load deflection curve of the beams for both the experimental and numerical data. The curves showed clearly the acceptable agreement between the finite element and the experimental results throughout the entire range of behavior. In general, the finite element model of the analyzed deep beams were stiffer in behavior compared with the actual beams, that may be clarified as; perfect bond between the concrete elements and the reinforcement is assumed in the ABAQUS analysis, but for the actual beams the assumption would not be true because the slip may occurs. Also, the presence of the microcracks (which is generated by drying handling and shrinkage) would reduce the stiffness of the actual beams, while the finite element models do not include such cracks as result to factors that are not incorporated into the models.

4.3 Crack Pattern

Figures 7 to 11 show the crack patterns obtained by ABAQUS program at failure load for the tested beams.

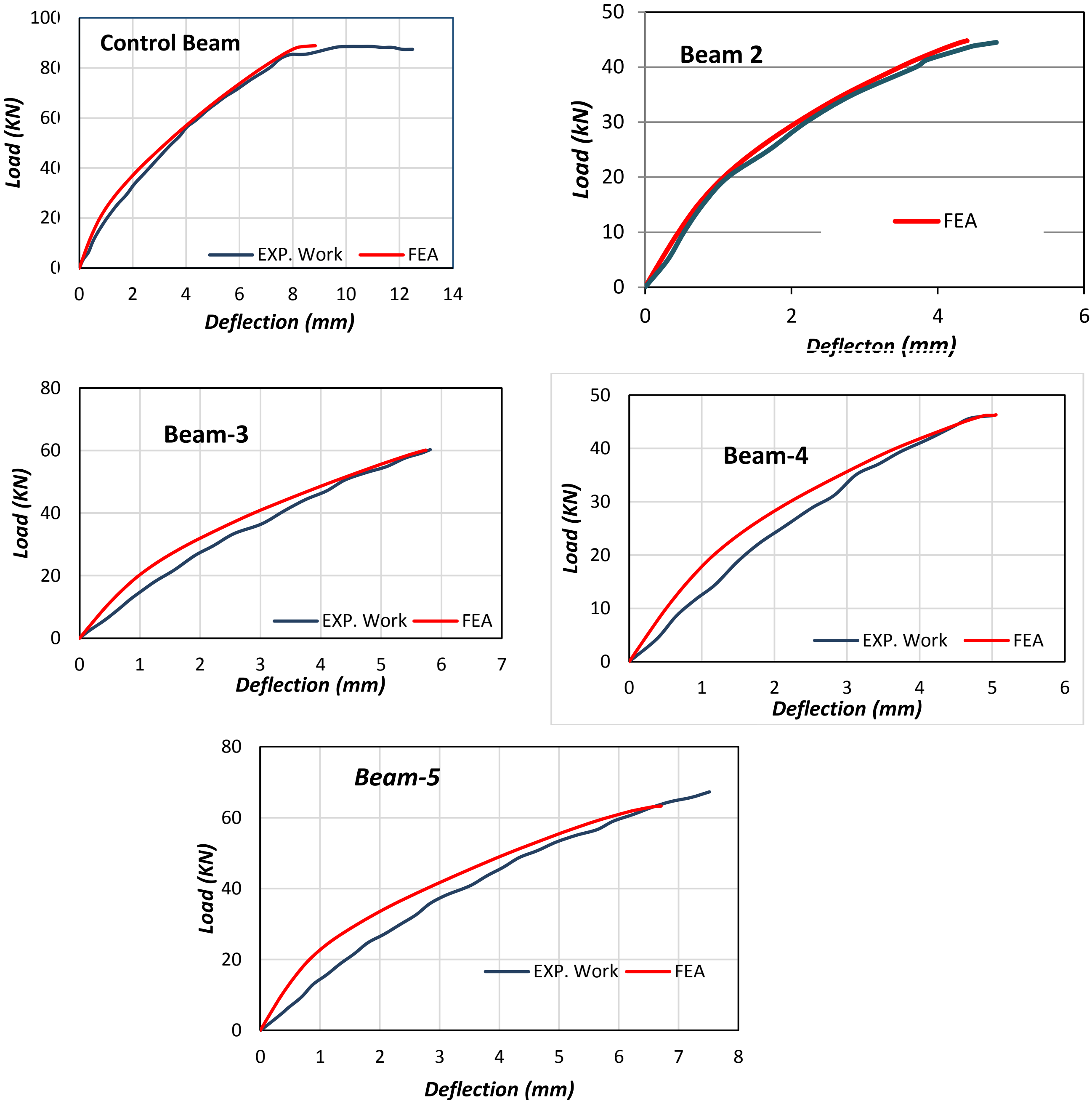


Figure 6: Load-midspan deflection relationship of the analyzed beams

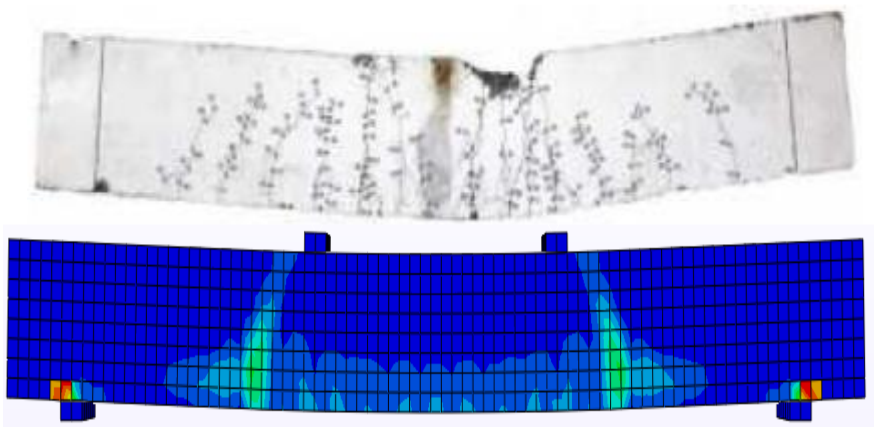


Figure 7: crack pattern of unstrenghtened control deep beam DB1.

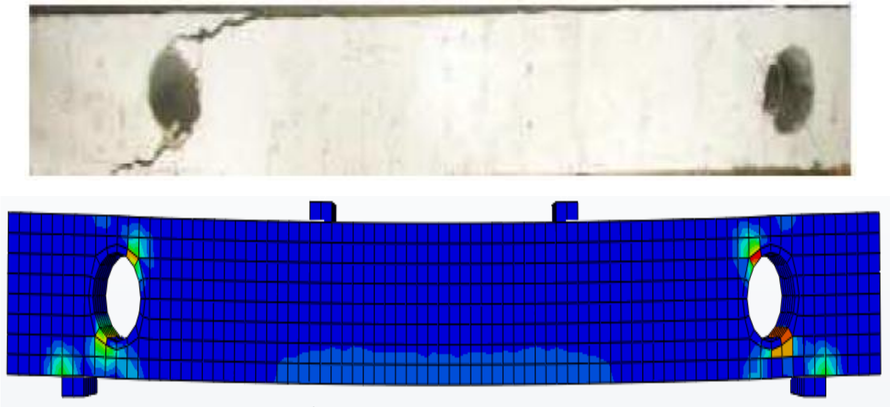


Figure 8: crack pattern of unstrenghtened control deep beam DB2.

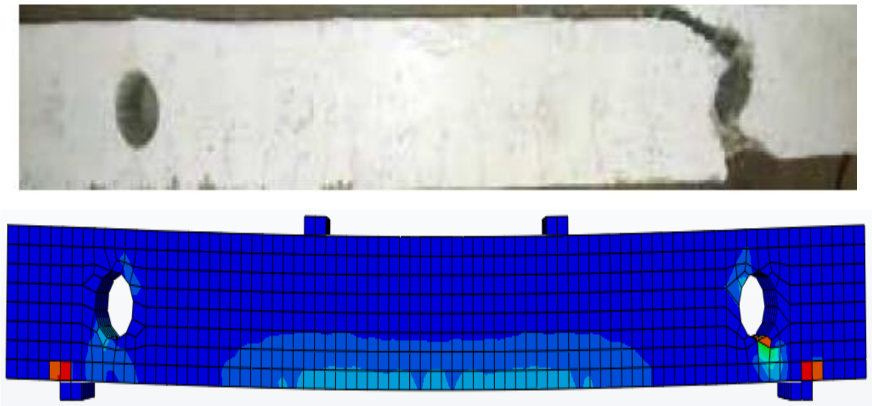


Figure 9: crack pattern of unstrenghtened control deep beam DB3

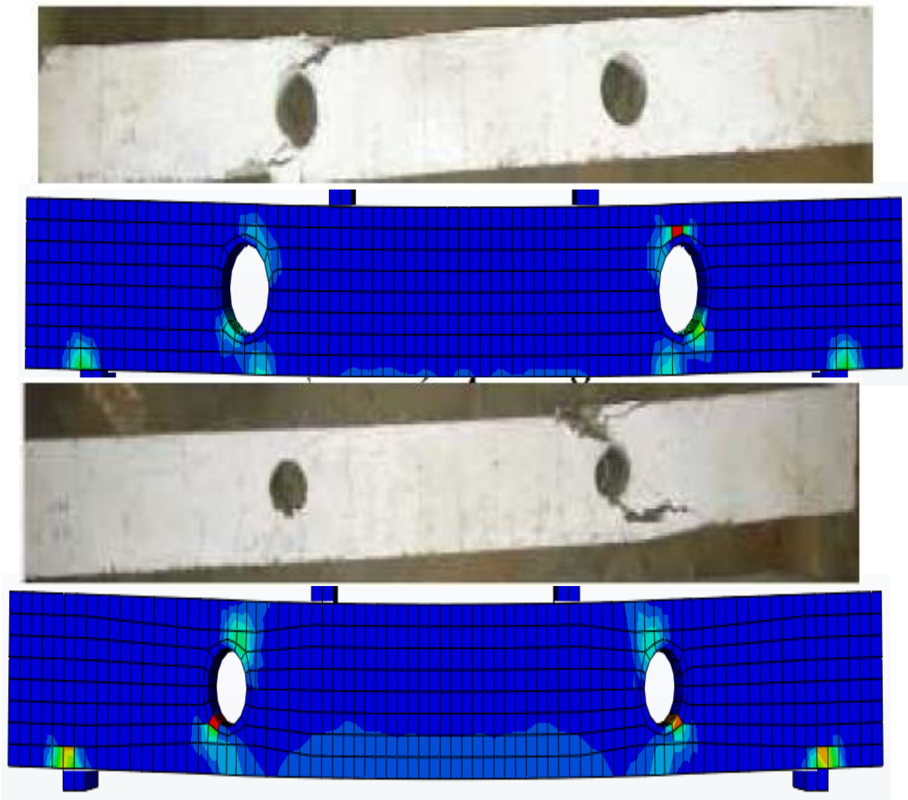


Figure 10: crack pattern of unstrenghtened control deep beam DB5

5. Summary and Conclusions

This paper aims to validate a finite element model to be use in analysis of rectangular reinforced concrete beams with openings. The validation procedure has based on comparison of ultimate loads and load-deflection curves obtained from the numerical finite element model by using ABAQUS computer program with those obtained from experimental work of Nilesh H. Saksena. Based on the numerical and experimental results discussed in this paper, the following conclusion are drawn

1) The comparison indicates a good agreement between the finite element and experimental results. Therefore the proposed finite element model including modeling of geometry, material, assembly, meshing, and applying of load can be adopted in future works with adequate confidante.
2) It is verified that the finite element analysis can accurately predict the load-deformation, load capacity and failure mode beams with openings. It can also capture cracking process for the shear-flexural peeling and end peeling failures, similar to the experiment.
3) Even though the number of tested deep beams was inadequate, a consistent tendency in their response is observed. Nonetheless, further experimental and numerical research may be needed to confirm the reproducibility of this study.

6. References

1. Mansur, M.A.; and Kiang-Hwee,T.(1999) *"Concrete Beams With Openings: Analysis and Design",* Boca Raton, Florida, CRC Press LLC, 220 pp.
2. Jin-Seop, L; and Sang-Sik, k., (2001)*" Shear Behavior of Reinforced Concrete Beams with Web Openings"*, Journal Of The Korea Concrete Institute , Vol. 13, No. 6, pp. 619-628.
3. Waleed E. El-Demerdash, , (2013*) "Design of Reinforced Concrete Beams with opening",* M.Sc. Thesis in structural design, El-Mansoura University/Egypt.
4. Nilesh, H. Saksena; and Patel, P.G., (2013*) "Experimental Study of Reinforced Concrete Beam with Web Openings"*, International Journal of Advanced Engineering Research and Studies, Vol. II, No. III, April-June, Pp. 66-68.
5. Barbero, E. J. (2013). *"Finite Element Analysis of Composite Materials Using ABAQUS TM"*, West Virginia: Taylor and Francis Group, LLC.
6. ABAQUS, *"Abaqus 6.12 Abaqus/CAE User's Manual"*, Inc., (2012) available at http://www.maths.cam.ac.uk/computing/software/abaqus_docs/docs/V6.12/pdf)books CAE.pdf.
7. Lubliner, J.; Oliver, J.; Oller, S. and Oñate, E., (1989)"A plastic damage model for concrete," International Journal of Solids and Structures, Vol. 25, No. 3, (). pp. 299-326.
8. ACI Committee 318, (2014) *" Building Code Requirements for Structural Concrete (ACI 318M-14) and Commentary on Building Code Requirements for Structural Concrete (ACI 318R-14),"* American Concrete Institute, Farmington Hill, MI, 524pp.