Finite Element Modeling of Reinforced Concrete Column after Exposure to Fire

Ali Raad Ismael Al-janabi[1] & Mohd Nizam Shakimon[2]

[1,2] Department of Civil Engineering, Faculty of Engineering and Technology Infrastructure, Infrastructure University, Kuala Lumpur, Malaysia
Correspondence: Ali Raad Ismael Al-janabi, Infrastructure University, Kuala Lumpur, Malaysia.
Email: ali.r.aljanabi@gmail.com

Received: October 2, 2017 Accepted: November 21, 2017 Online Published: December 1, 2017

doi: 10.23918/eajse.v3i2p181

Abstract: Reinforced Concrete is the most essential composition for construction which is a great combination to building structures in any desired form. Previously, the behavior of concrete during and after fire has often been taken for granted considering it as a non-combustible material. Although RC is non-combustible, steel bars and concrete degrade in strength during and after exposure to fire; "high temperatures". In the present research, non-linear Finite Element Modeling (FEM) was employed to build a 3D model of RC column with 150×150×1000 mm^3 dimensions in Abaqus. The column was analyzed after exposure to fire (EuroCode ISO834) under several values of axial loads. The aim of this study is to provide a 3D FE model to measure the vertical displacement and stress distribution across the RC column after exposure to fire considering the influence of load and heat rise. To ensure the accuracy of simulation, the obtained displacement data were validated and compared to experimental and theoretical results (Finite Element Modeling (FEM)) results available in the literature. The simulation indicated that the stress distribution varies after fire emphasizing the higher stress on the surface of the column. Moreover, the vertical displacement was increased after exposure to fire.

Keywords: Reinforced Concrete Column, Finite Element Analysis (FEA), Simulation, Abaqus, Fire

1. Introduction

Reinforced Concrete (RC) is widely used as construction material owing to high shaping ability that serves the needs at a reasonable cost. Furthermore, it can be designed to provide high compressive strength. Exposure to fire is an important aspect of RC which might lead to buildings collapse. Understanding the behavior of the reinforced concrete exposed to direct fire or rapid increase in temperature helps us to evaluate the level of safety of our built structures.

In the past two decades, many researchers have studied the behavior of RC exposed to fire. Understanding the behavior of column in extreme temperatures helps to determine the structure's safety level in the case of fire. It was indicated that the ultimate load for a column exposed to fire is 20-40% less than the column before experiencing fire (Bikhiet, El-Shafey & El-Hashimy, 2014). A significant reduction in the modulus of elasticity also occurred in high temperature, which indicated a reduction of 62-72% after being heated up to 400℃ (Kadhum, Fawzi & Hashim, n.d). Moreover, Kadhum (n.d) showed that cracks might develop on the surface of concrete after exposure to fire due to loss of water content in the concrete. Furthermore, the type of aggregate is influential of the High

Strength Concrete (HSC) thermal properties at elevated temperatures (Kodur & Sultan, 2003). The presence of carbonate aggregate in HSC increases fire resistance.

To gain a comprehensive understanding of concrete column in fire, FEM has often been used to create the behavior of materials in high temperatures. A non-linear finite element model was created by Bikhiet et al. (2014) to investigate and determine the behavior of the reinforced concrete column exposed to fire for 15 minutes and axially loaded. The model was used to determine the values of maximum load, vertical displacements analyzed columns. The model was divided into finite element model Hexahedron meshing shape using type coupled thermal displacement type. The mechanical properties of steel and concrete were measured from the experimental work; these properties were used to create the model. The analysis has showed the following that the column ultimate axial displacement increase while the columns' ultimate failure loads decrease after exposed to fire. The maximum lateral displacement along column axis occurred mainly at the mid of the column. Furthermore, the failure load of a column exposed to fire reduces by approximately 20% to 35%. The research conducted by Bikhiet et al. (2014) will be further compared with the displacement results obtained in the present study.

Experimental results from Bikhiet et al. (2014) indicated that concrete begins to lose about 30% of its compressive strength when heated up to 300℃ and loses about 70% of its compressive strength when heated up to 600℃. Concrete's modulus of elasticity reaches 60% of its original value at 300℃ and reaches 15% of its original value at 600℃.

The present study is covering the simulation of RC column that is exposed to fire. The column is to be exposed to a range of flame temperature for 15 minutes under axial load, to observe and study the behavior of the column during fire; and the changes that may occur especially the changes in the vertical displacement of the column during fire. By creating a model of reinforced concrete column using Abaqus software, fire exposure is simulated and the changes occurring in the column is observable. The objective of this study is to investigate the vertical displacement and stress distribution across a RC column after its exposure to fire. The model of RC column in 3D is created in Abaqus software and the fire load is defined according to ISO834 standard fire curve.

2. Simulation

Finite element method was used to creating a 3D model of reinforced concrete column under load and exposed to high temperature in Abaqus. Each element concrete and steel were modeled separately based on the physical properties that are needed to create the elements for this study. Concrete and steel reinforcement elements were designed base on the density, modulus of elasticity, poisson's ratio, conductivity and specific heat. Steel bars and concrete parts were created separately then assembled to gather using the Tie method to create one part and create interaction between concrete and the steel bars. By defining the thermal properties of each material Abaqus was able to simulate the variation of the temperature through the column cross-section. Where the temperature is highest at the external surface and starts to degrade through the cross-section toward the center. The element type was chosen to be coupled temperature-displacement before meshing the parts. Specimen C4 data and results taken from Bikhiet et al. (2014) were used to validate the FE model of this study. C4 specimen has a compressive strength of 425 kg/cm^2, modulus of elasticity of 180 t/cm^2, High grade steel for reinforcement, the specimen has dimensions of (15*15*100) cm. The model in this study was created with the same dimensions of (15*15*100) cm and concrete cover of 25mm. The column was determined to be fixed at the bottom and free at the top. To ensure that the created model has a similar physical property as specimen C4 the same modulus of elasticity was assigned in Abaqus. Density (2400kg/m^3), poisson's ratio (0.15), specific heat (900J/kg), conductivity of concrete (1.33w/mk) were defined as well to simulate concrete. The physical properties of high grade steel were defined in Abaqus according to the Euro-code with a diameter assumed to be 16mm, 4 steel bars were used. In the case of fire, the temperature is not a constant. The temperature rises with respect to time, according to the Euro-Code ISO834 fire curve defined in equation (1) and illustrated in Figure 1.

$$T = To + 345\log_{10}(8t+1) \quad (1)$$

where *T* is the temperature, *To* is the room temperature and *t* is time. Using the fire curve ISO834 it is possible to determine the temperature at any specific time.

The reinforced concrete column was exposed to fire for 15 minutes and axially loaded with a verity of loads ranging from 10 to 80 tons. After 15 minutes, the temperature increases to 718 °C which is obtained from Equation (1) and used for the simulations. The rise of temperature was simulated by creating an amplitude in Abaqus that follow the fire curve and define the changes in the temperature every minute. The temperature was calculated for each minute using Equation (1).

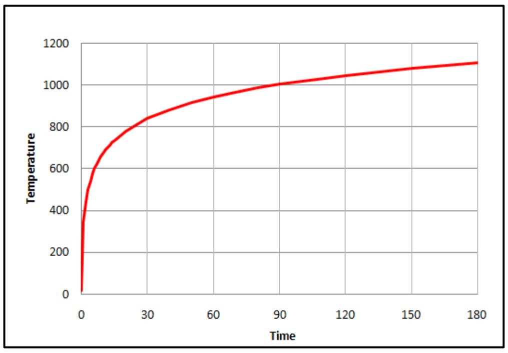

Figure 1: Standard Fire Curve (Eurocode Backgrounds and Applications, 2014)

3. Results and Discussion

3.1 Stress Distribution

To show how the increment of temperature affect the RC column, a compression of the stress distribution in the column cross section before and after fire exposure was accomplished, showing the changes in the stress distribution across the column (Figure 2). By comparing the stress distribution in the cross section of the column before and after fire, the stress in column increased when temperature increased as shown in Figure 2(b) and Figure 2(c).

The red contour specifies the highest stress area. As it is clear in Figure 2(c), after the RC Column is exposed to fire, the highest stress occurred on the outer surface of the column where the temperature is the highest. Concluding that, when the temperature of the RC column increases so does the stress, especially in the outer layer of the column.

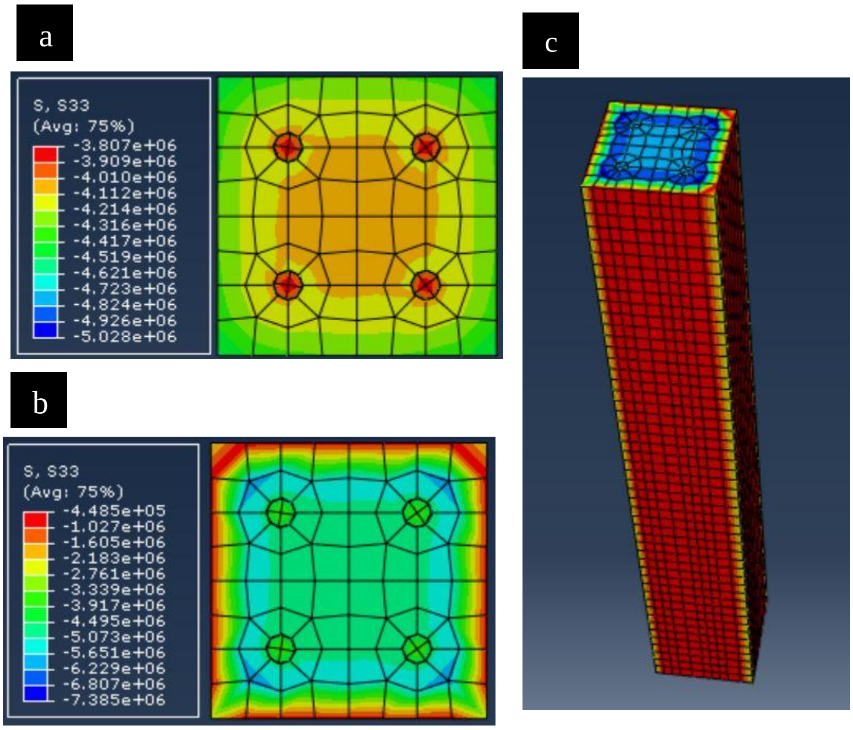

Figure 2: State of stress in reinforced concrete column (a) before; after fire (b) in 2D and (c) 3D under 10 tons of load

3.2 Displacement

It is very important to measure the displacement occurrence under load while the temperature is increasing to understand and determine the behavior of the RC column exposed to fire. It was found that the displacement increases with rise in temperature, as evident in Figure 3.

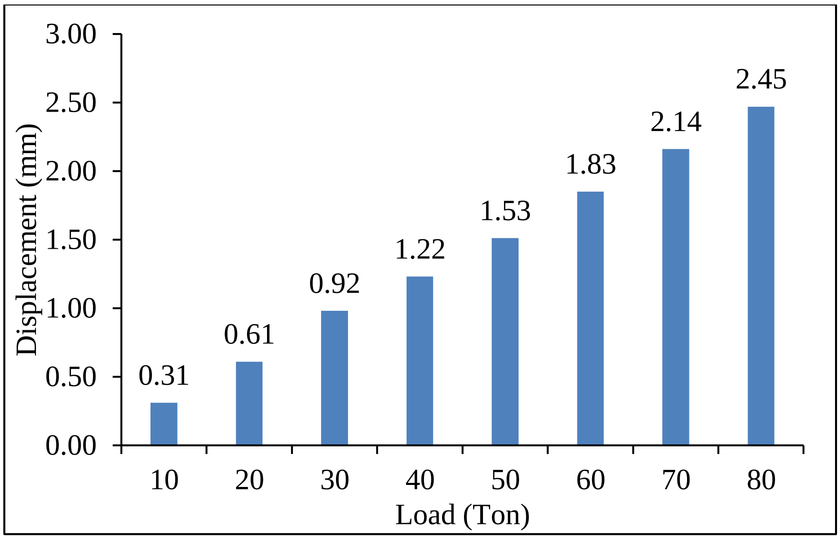

Figure 3: Simulation results of load-vertical displacement relationship

The obtained results were compared to the simulation and experimental work of Bikhiet et al. (2014). It was found that the result where attained from Abaqus are very close to the results of the theoretical work (FEM based) as shown in Figure 4. It is very expected to have a large different from the experimental work since during the experimental work it is very difficult to control all the factors that affect the experiment results. Having a very similar result to the theoretical work (FEM analysis) proves that the simulation was done correctly and provided another way to test the behavior of RC column exposed to fire with errors tabulated in Table 1.

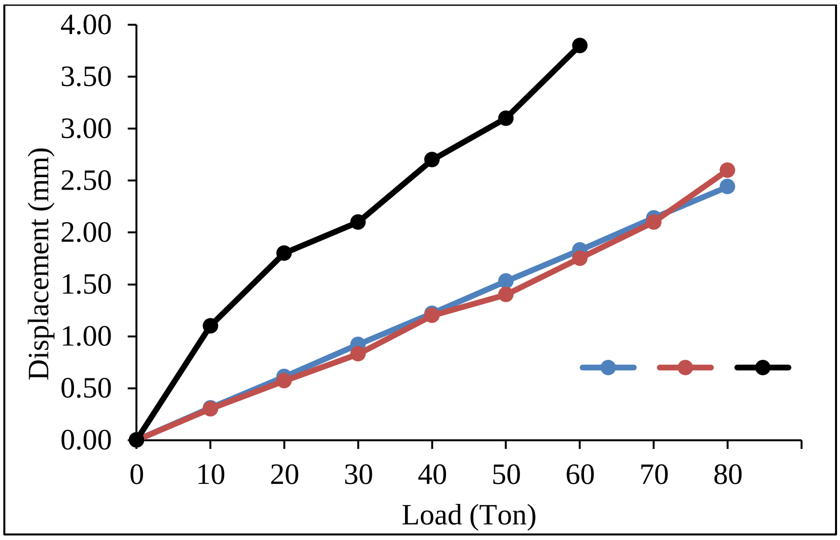

Figure 4: Comparison of load-vertical displacement of results obtained in this article with theory and experiment accomplished by Bikhiet et al (2014)

Table 1:Validation table

Load (Ton)	Theory (mm)	Abaqus (mm)	Error %	≤ 10% Pass
10	0.30	0.31	3.2%	Pass
20	0.57	0.61	6.5%	Pass
30	0.83	0.92	9.7%	Pass
40	1.20	1.22	1.6%	Pass
50	1.40	1.53	8.3%	Pass
60	1.75	1.83	4.3%	Pass
70	2.10	2.14	1.8%	Pass
80	2.60	2.45	6.1%	Pass

4. Conclusion

In the present study, Finite Element Modeling (FEM) was used to simulate the behavior of reinforced concrete column exposed to fire several under axial loads using Abaqus software. The influence of heat-rise on vertical displacement and stress distribution across the RC column was observed. The reinforced concrete column was exposed to fire for 15 minutes causing a rise in the temperature in the outer up surface of the column to 718 °C. A variety of loads were applied to the RC column ranging from 10 tons to 80 tons. It was found that the increase of the surface temperature along with the applied load caused a larger displacement, if the temperature and the load increase the column will fail faster. Furthermore, the simulation showed how the temperature completely affects the state of stress and its distribution. It was found that the stress is high in the steel bars before fire but after fire the highest stress was found on the outer surface of the column due to high temperature of the surface of the column. Moreover, the vertical displacement was recorded and compared to the simulation and experimental results attained from the literature. The results were almost identical to what has been done previously.

References

Bikhiet, M., El-Shafey, N., & El-Hashimy, H. (2014). Behavior of reinforced concrete short columns exposed to fire. *Alexandria Engineering Journal*, 53, 643-653.

Kadhum, M., Fawzi, N., & Hashim, K. (n.d.). Experimental Studies on The Fire Endurance of Reinforced Concrete Beams.

Kadhum, M. (n.d.). Effect of Burning by Fire Flame on the Behavior of Reinforced Concrete Beam Models.

Kodur, V., & Sultan, M. (2003). Effect of temperature on thermal properties of high-strength concrete. *Journal of Materials in Civil Engineering*, 15, 101-107.

Eurocode Backgrounds and Applications. (2014). In Euro Code ISO 834 curve, ed.