INTERNATIONAL JOURNAL OF CIVIL AND STRUCTURAL ENGINEERING
Volume 3, No 3, 2013

Research article ISSN 0976 – 4399

Prediction of cracks in concrete frames by finite element modeling

Imad Adel Ahmad Al-Qasem[1], Ibrahim Mohamad Ahmad Arman[2]
1- Lecturer, Civil Engineering Department, Al-Najah National University, Nablus
2- Lecturer, Civil Engineering Department, Al-Najah National University, Nablus
imadqa@najah.edu
doi:10.6088/ijcser.201203013049

ABSTRACT

In recent years, the employment of computer techniques in engineering design work has become an irreversible trend. This is achieved through using computer as the tool for modeling, analysis, and design. The objective of this paper is to determine the stress in unreinforced concrete portal frame and predict the cracks which will occur in the beam and columns in this frame due to this stress. To do this, a technique utilizing finite element analysis is used in order to create computerized model using shell elements with SAP2000 program. This model is built to simulate the real state of beam and columns in the frame under different loading conditions.

Keywords: Cracks, portal frame, stress, finite element analysis, computerized model.

1. Introduction

Reinforced concrete (RC) frames consist of horizontal elements (beams) and vertical elements (columns) connected by rigid joints. These structures are cast monolithically, that is, beams and columns are casted in a single operation in order to act in unison. Reinforced concrete frames provide resistance to both gravity and lateral loads through bending in beams and columns. In this paper we will try to study a simple portal frame and understand its behavior under different load conditions. The main goal in this paper is to study the effects of stress in a portal frame and determine the cracks which occur due to this stress. The portal frame that will be studied in this paper is made of plain concrete; reinforcement is neglected. To do this, a computerized portal frame model by using shell elements is analyzed in three different cases, the first case we will determine stress and cracks which will occur in the beam and columns when the frame is subjected to gravity load. In the second case, stress and cracks which will occur in the beam and columns will be determined when the frame is subjected to horizontal load. In the third case, stress and cracks which will occur in beam and columns will be determined when differential settlement will occur in frame supports.

2. Model Description

A model represents portal frame can be built using one of the computer's structural analyses programs, and sets of input are given, such as load on frame, material characteristics, and model size. And output, such as stress in beam and columns. SAP2000 program is used to create and analyze finite element model for portal frame, considering that there is compatibility between this model and the actual case, as shown in figures 1 and 2. In order to represent a structural model for portal frame the following assumptions are used

Received on February 2013 Published on March 2013

1. The frame consists of one beam with dimensions (400 x 200 mm) carried by two columns with (400 x 200 mm).
2. The span length of frame is (4m) from center to center.
3. The clear height of columns in the frame equals 3m.
4. Pin hinges are used to support the joints of column supports in the model, but it works as fixed support in real modal.
5. The frame is made of concrete with compressive strength f'_c = 28 MPa.

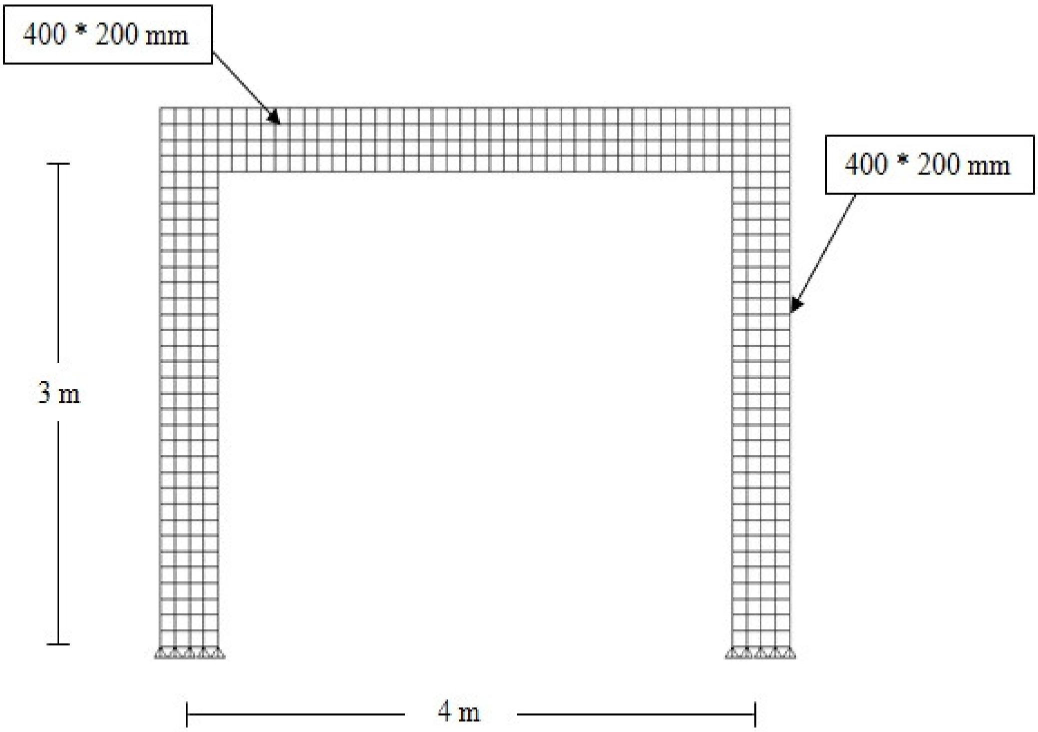

Figure 1: Finite element model for portal frame

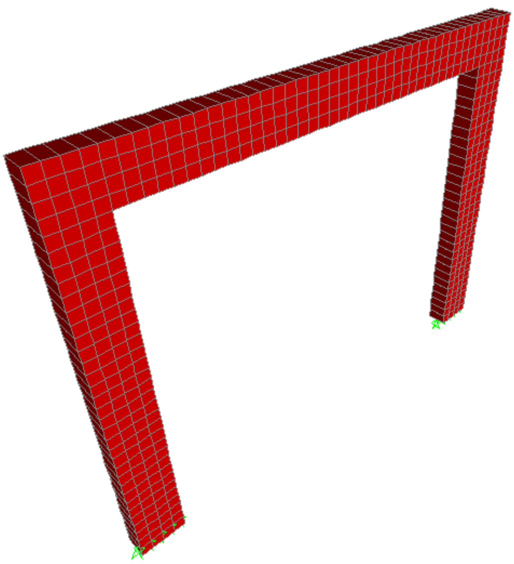

Figure 2: Finite element model for portal frame in 3-D

2.1 Case 1: Portal frame model under gravity load

In this case we will analyze the frame model under gravity load and see how the stress will distribute in the beam and columns, and see the cracks which will occur in frame due to this stress.

2.1.1 Load

In this case the following assumptions are used

1. The frame carries a gravity uniform load of 60 kN/m
2. The own weight of frame is neglected.
3. The uniform load on the beam is put as distributed load on the upper area of the beam. The gravity load value will be divided by the width of beam (60 / 0.2 = 300 kN/m^2).

2.1.2 Analysis

After the frame has been analyzed under the previous assumptions by using SAP2000 program, the following results are obtained:

1- The distribution of stress on beam and columns in the frame as shown in figures 3 and 4.
2- Determine the critical area for high tension stress in the frame which affect negatively on the concrete section and cause the cracks. (i.e) when saying that it's a high tension stress area, that's the area which has a tension stress greater than ($0.62 * \sqrt{f'_c} = 3.28$ MPa: equation 9-10, ACI318-08M).
3- Determine the location and the depth of cracks which will occur in the frame as shown in the Figure (5).

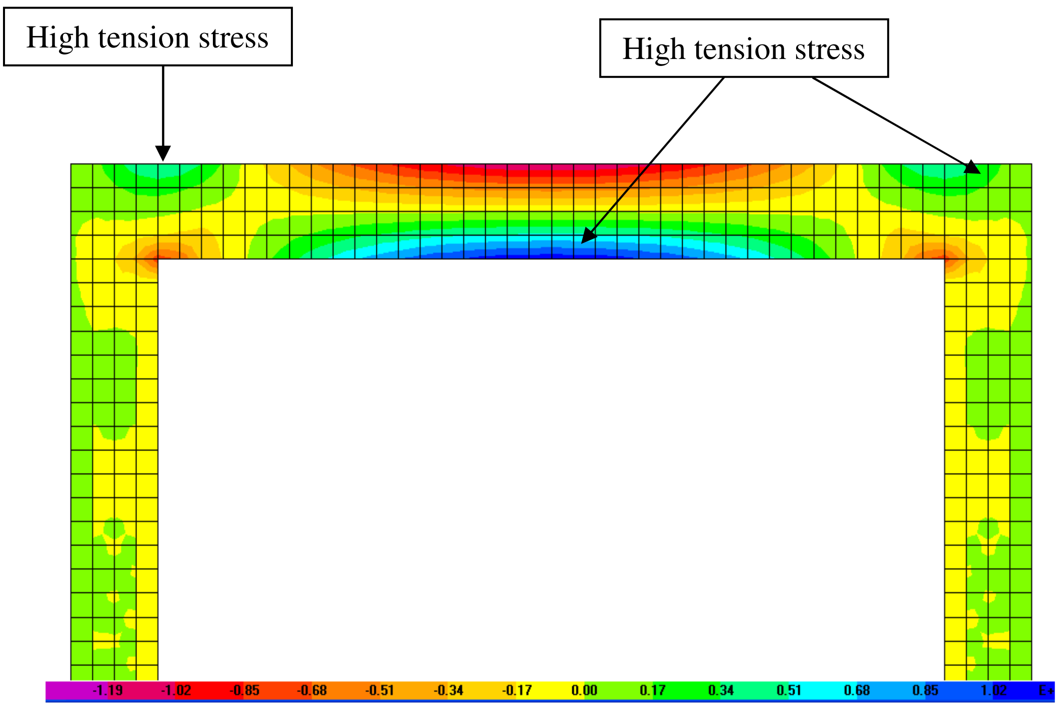

Figure 3: High tension stress on the beam under gravity load

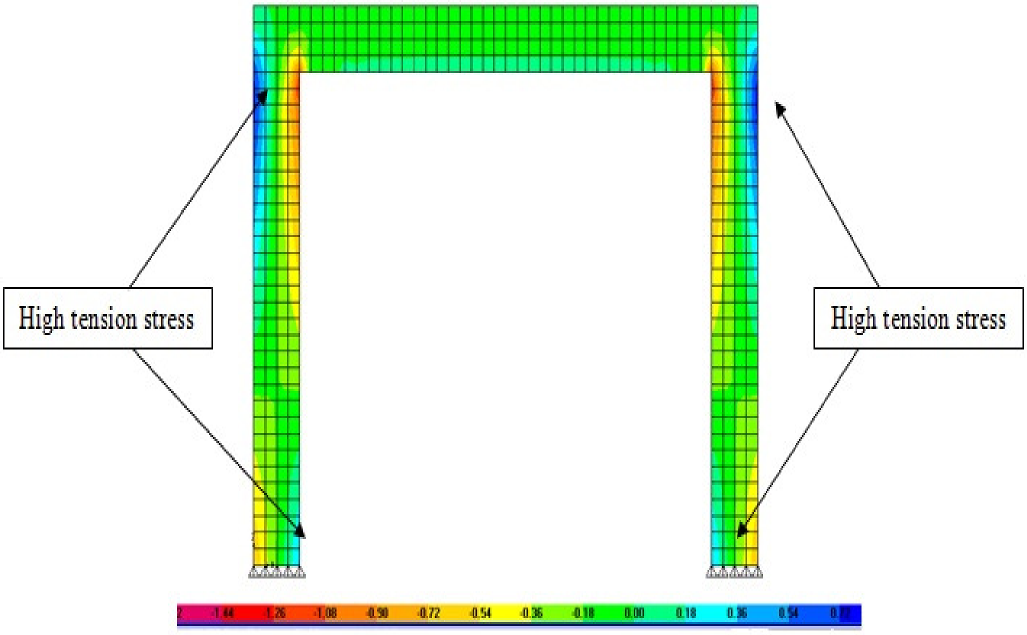

Figure 4: High tension stress on the columns under gravity load

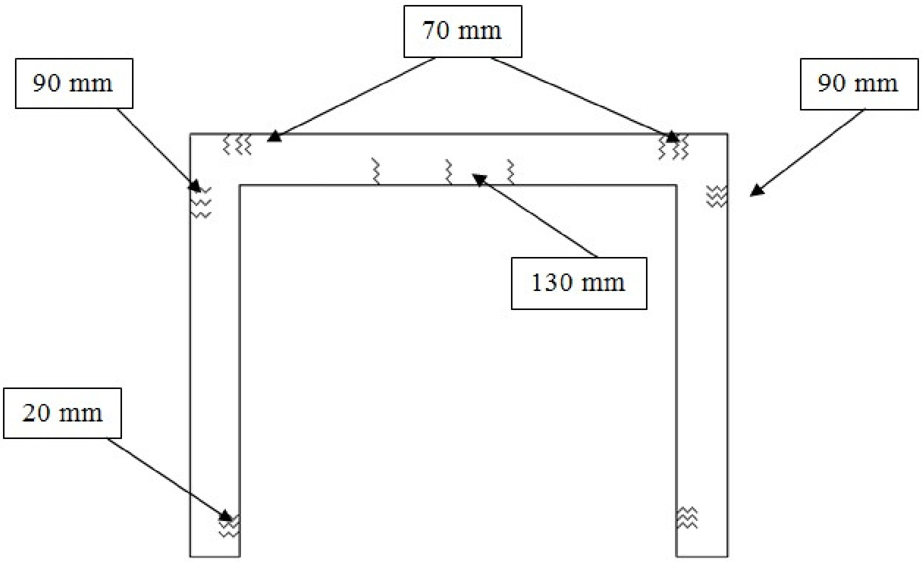

Figure 5: Cracks on the frame under gravity load

2.2 Case 2: Portal frame model under horizontal load

In this case we will analyze the frame model under horizontal load only, and see how the stress will distribute in the beam and columns, and see the cracks which will occur in frame due to this stress.

2.2.1 Load

In this case the following assumptions are used:

1. The frame is subjected to a horizontal concentrated load of 50 kN.
2. The own weight of frame is neglected.
3. The concentrated load on the joint of frame is distributed along the joint as shown in the figure 6.

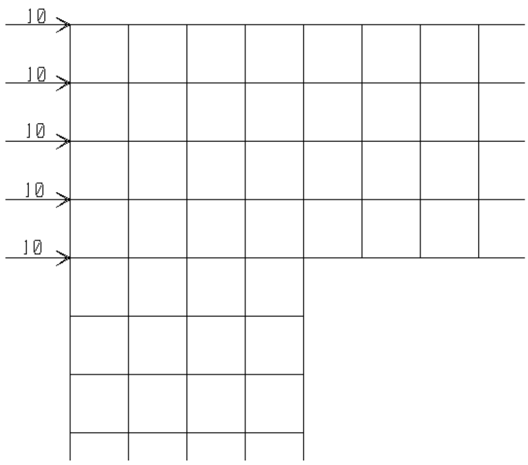

Figure 6: Distribution of the concentrated load along the joint

2.2.2 Analysis

After the frame has been analyzed under the previous assumptions by using SAP2000 program, the following results are obtained:

1- The distribution of stress on beam and columns in the frame as shown in the figures 7 and 8.
2- Determine the critical area for high tension stress in the frame which affect negatively on the concrete section and cause the cracks.
3- Determine the place and the depth of cracks which will occur in the frame as shown in the figure).

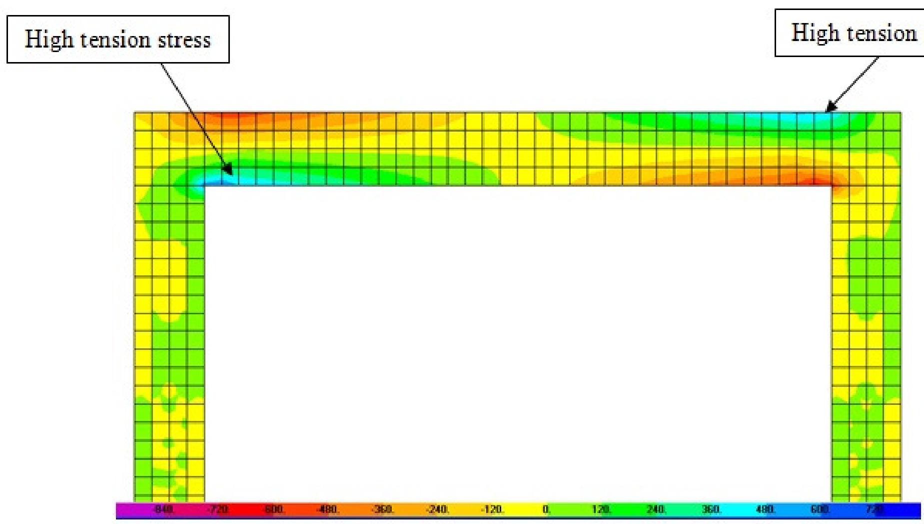

Figure 7: High tension stress on the beam under horizontal load

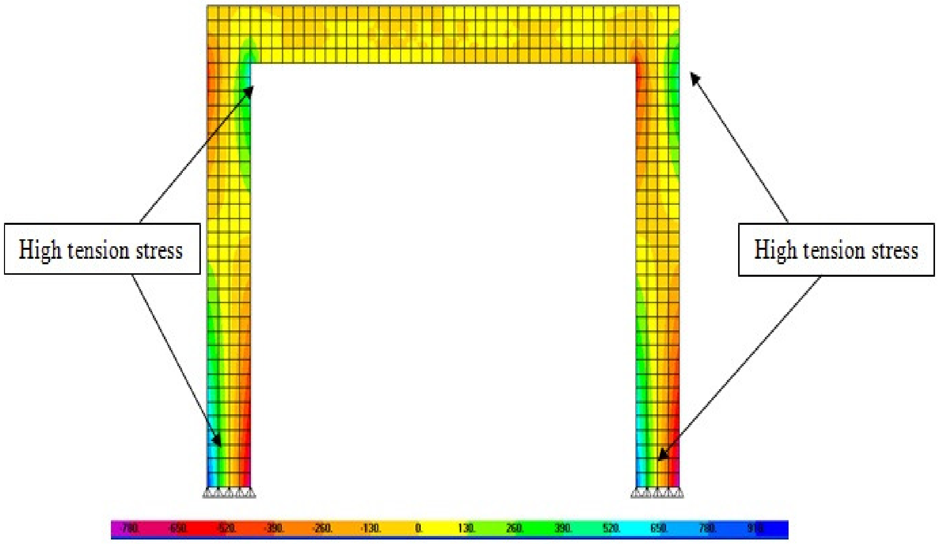

Figure 8: High tension stress on the columns under horizontal load

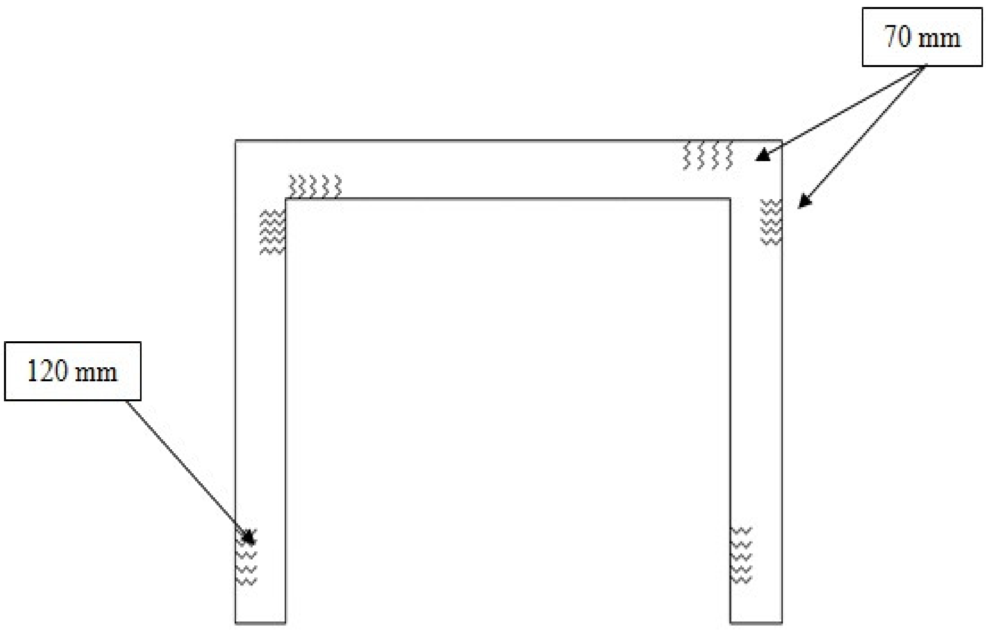

Figure 9: Cracks on the frame under horizontal load

2.3 Case 3: Portal frame model under differential settlement

In this case we will analyze the frame model under differential settlement only, and see how the stress will distribute in the beam and columns, and see the cracks which will occur in frame due to this stress.

2.3.1 Assumptions

In this case it is assumed that the right support in frame settles down 50 mm and the own weight of frame is neglected.

2.3.2 Analysis

After the frame has been analyzed under the previous assumptions by using SAP2000 program, the following results are obtained

1- The distribution of stress on beam and columns in the frame as shown in the figures 10 and 11.
2- Determine the critical area of high tension stress in the frame which affect negatively the concrete section and cause the cracks.
3- Determine the place and the depth of cracks which will occur in the frame as shown in the figure 12.

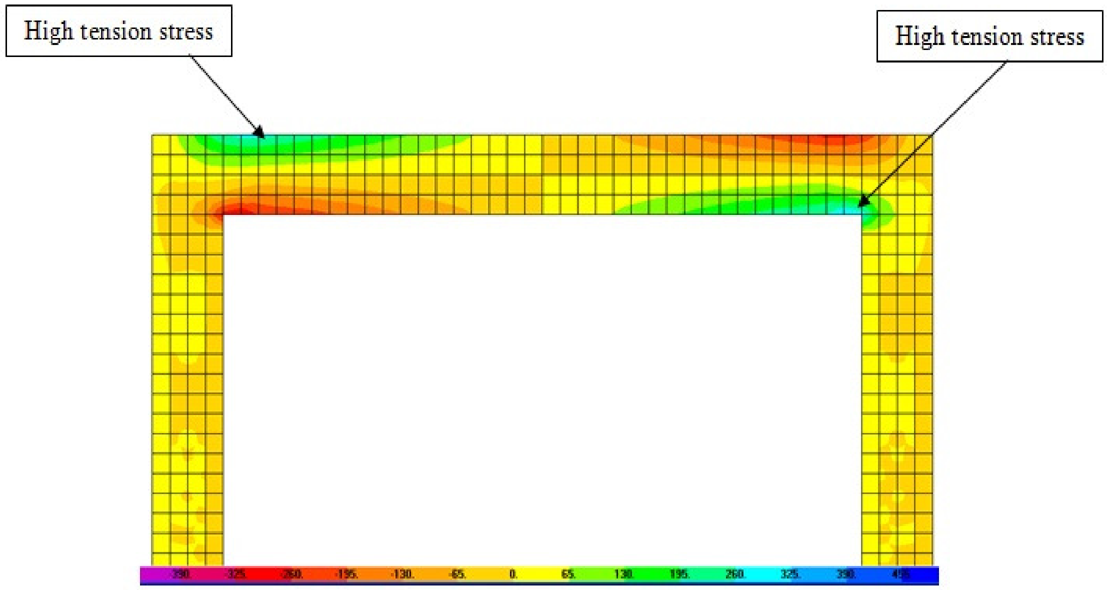

Figure 10: Tension stress on the beam under differential settlement

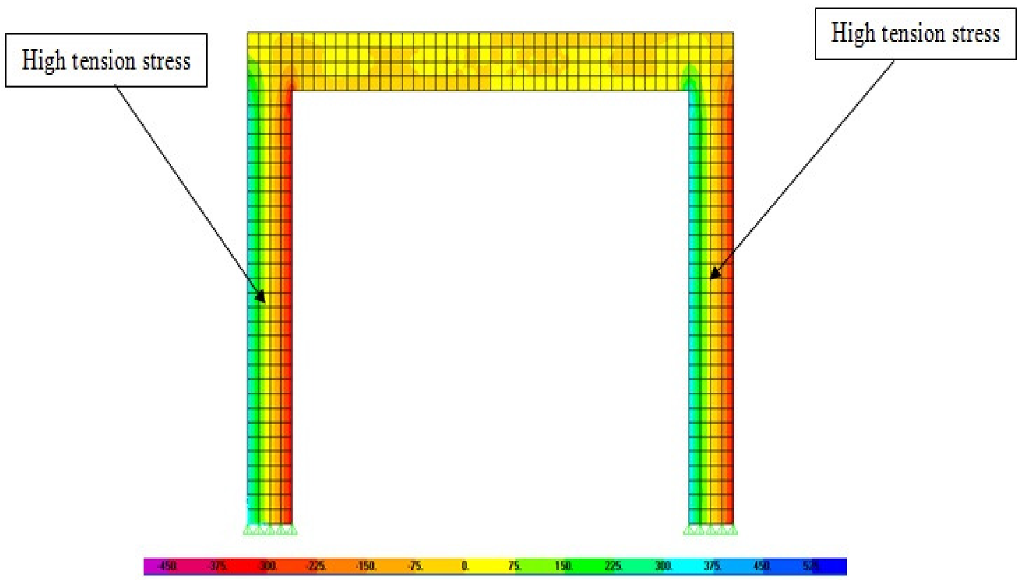

Figure 11: Tension stress on the columns under differential settlement

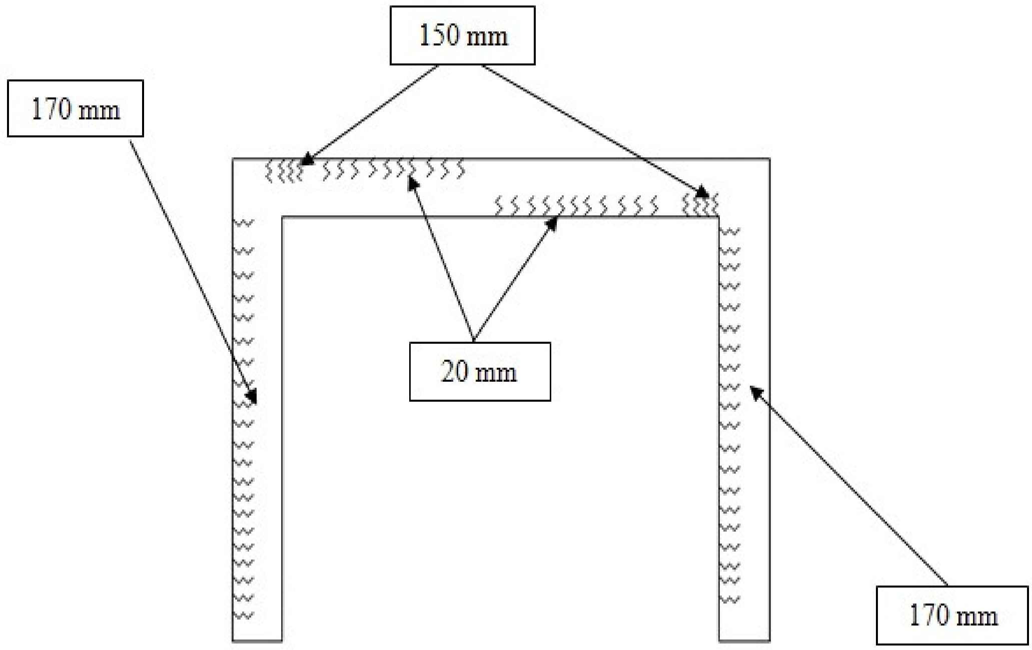

Figure 12: Cracks on the frame under differential settlement

3. Prediction of cracks in reinforced concrete frames

In this paper, the study deals with plain concrete frames. So, the flexural stiffness of the members depends on the gross moment of inertia without taking into account reinforcing steel. The existence of reinforcing steel will increase the section moment of inertia by a slight amount especially when the reinforcement ratio is small. The gross moment of inertia can be increased by about 5 to 15% by adding the reinforcing steel. The frames in this study can be reanalyzed taking into account reinforcing steel and the results can be obtained using an increased value of gross moment of inertia. Then, the extension and depth of cracks can be determined

4. Conclusion

According to this study, civil engineer can depend on 3-D model using shell finite elements to analyze building and find the cracks which may occur due to different loadings.
The depth of cracks can be predicted depending on the value of tensile stress. The crack happens when the stress at a point is larger than the modulus of rupture which is the assumed tensile strength of concrete. This study was based on the analysis of a plain concrete frame which is subjected to different loadings. A future studies can be done to predict crack formation in reinforced concrete frames using finite element modeling.

5. References

1. American Concrete Institute. Farmington Hills, MI., (2008), Building Code Requirements for Structural Concrete and Commentary (ACI 318M-08).

2. EL-Borgi.S., Smaoui.H., Cherif.F., Bahlous.S., and Ghrairi.A., (2004), Modal Identification and Finite Element Model Updating of a Reinforced Concrete Bridge, Emirates Journal for Engineering Research, 9 (2), pp 29-34.

3. Hashemi.A., and Mosalam K.M, (2004), Transient Analysis of Reinforced Concrete Frame With and Without Masonry Infill Wall Under Blast. Emirates Journal for Engineering Research. 9 (2), pp 97-103.

4. Yakut.A., (2004), Reinforced concrete frame construction available at http://www.world-housing.net/wp-content/uploads/2011/06/RC-Frame_Yakut.pdf, accessed during January 2013.