TuEngr Group

International Transaction Journal of Engineering, Management, & Applied Sciences & Technologies

http://TuEngr.com

Behaviors of Concrete Beam to Column Connections under Static Load Using Finite Element Method

Atthaphon Atichat [a], **Sayan Sirimontree** [a*], and **Boonsap Witchayangkoon** [a]

[a] *Department of Civil Engineering, Faculty of Engineering, Thammasat University, Pathumtani, THAILAND.*

ARTICLEINFO

Article history:
Received 04 September 2017
Received in revised form 17 November 2017
Accepted 23 November 2017
Available online 29 November 2017

Keywords:
ABAQUS;
Beam-Column Joint;
Reinforce concrete;
Structural stiffness;
Structural ductility.

ABSTRACT

Precast concrete structures have been widely used in the construction industry. This is due to the reduction of time and labor cost in construction of concrete building structures. Strength and ductility of connection of the precast element must be adequately designed to prevent failure of connection prior to failure of members. Experiments performed by previous researchers were done to proof the strength stiffness and ductility of beam-column connection but high cost and time consuming. In this work, the nonlinear finite element model was developed to analyze the behaviors of reinforced concrete beam-column connection of precast concrete structures. The model was verified by the experimental results performed by the previous work. It was found a good correlation between test and analytical results.

1. Introduction

The precast concrete building system has been widely used in many countries. It can reduce construction time and solve the shortage of labor. Quality of precast concrete members can be controlled thoroughly and effectively in a factory before the erection on the construction site. Precast concrete components can be column, beam, slab and wall. Important factors to consider in the design of precast concrete structure are strength, stiffness and ductility of connection of structural members. The connection must be strong enough to transmit force between the members without failure before the members reach their maximum strength. Many previous researchers have studied the behavior and strength of joints in the laboratory, but there are costs and time consuming. This research has developed a nonlinear finite element model to analyze the behavior and strength

*Corresponding author (S. Sirimontree). E-mail: ssayan@engr.tu.ac.th.

of beam-column connections to reduce experiments. Software, ABAQUS, was utilized in the analysis. The model is verified by the test results performed by (Thaweesak, 2015).

2. Literature Review

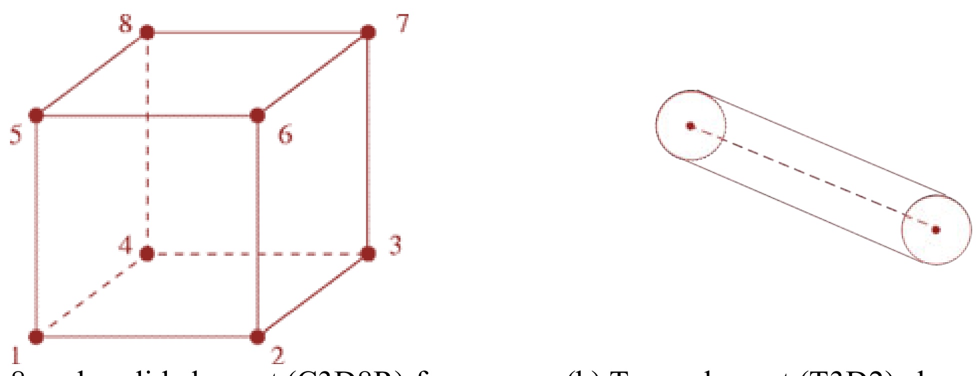
Long (2014) analyzed the behavior of beam-column joints under static load by a 3D finite element method with ABAQUS software. This model used the concrete damaged plasticity model and 8-node solid element (C3D8R) to simulate concrete and truss element (T3D2) elements to simulate reinforcing steel. The analytical results were verified to test results performed by Peng and Wang (2010).

Sinaei (2011) analyzed the behavior of carbon fiber reinforced concrete beam-column joints using ABAQUS software. This study proved and confirmed the ability of a program to analyze before compare with results of Mahini and Ronagh (2007). Analysis showed that the finite element was consistent with experiment. Finite element can thus be used to analyze reinforce concrete beam-column joint, in which this study is about.

3. Theoretical Approach

3.1 Element Types

ABAQUS (ABAQUS, 2008) is one of the most efficient software used in nonlinear finite element analysis, which is selected to be used in this work. This study uses 8-node solid element (C3D8R) to simulate concrete and truss element (T3D2) elements to simulate reinforcing steel as shown by Figure1.



(a) 8-node solid element (C3D8R) for concrete

(b) Truss element (T3D2) elements for reinforcing steel

Figure 1: Element used in the finite element model of beam-column connection.

3.2 The stress-strain relation of concrete

The stress-strain relation of concrete in compression and tension is accounted for compression and tension stiffening of reinforced concrete after cracking called damage plasticity model proposed by Hsu and Hsu (1994) as shown in Figure 2. The stress-strain relation of reinforcing steel is assumed elastic-perfectly plastic as shown in Figure 3. Equations (1)-(4) are used in the construction of material model of concrete in compression.

$$\sigma_c = \left(\frac{\beta\left(\frac{\varepsilon_c}{\varepsilon_o}\right)}{\beta - 1 + \left(\frac{\varepsilon_c}{\varepsilon_o}\right)^{\beta}} \right) \sigma_{cu} \qquad (1)$$

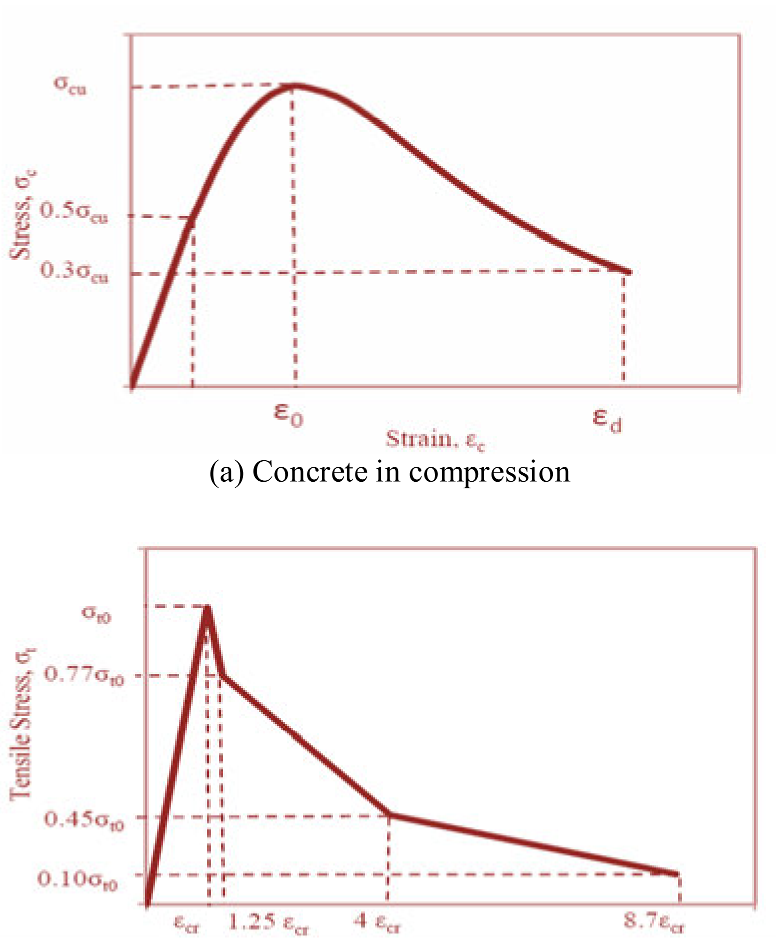
$$\beta = \frac{1}{1 - \left[\sigma_{cu} / \left(\varepsilon_0 E_0\right)\right]} \qquad (2)$$

$$\varepsilon_0 = 8.9 \times 10^{-5} \sigma_{cu} + 2.114 \times 10^{-3} \qquad (3)$$

$$E_0 = 1.2431 \times 10^{2} \sigma_{cu} + 3.28312 \times 10^{-3} \qquad (4)$$

Where σ_c refers to compressive stress, ε_c refers to compressive strain, and σ_{cu} refers to ultimate compressive stress. Note: $\varepsilon_d = 0.85\varepsilon_c$ and units of Equations (1) to (4) are in kip/in^2 (1Mpa = 0.145037743 kip/in^2). Point ε_d in Figure 2(a) is strain at the $0.3\ \sigma_{cu}$.

*Corresponding author (S. Sirimontree). E-mail: ssayan@engr.tu.ac.th.



(a) Concrete in compression

(b) Concrete in tension

Figure 2: Consitutive model of concrete used in ABAQUS (Wahalathantri et al., 2011)

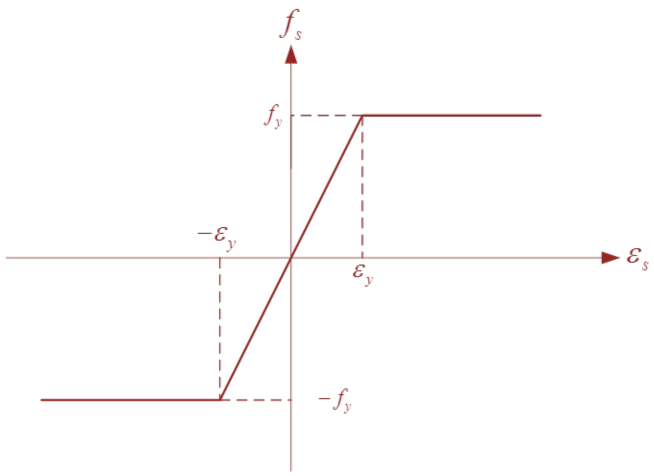


Figure 3: Elastic-perfectly plastic model of reinforcing steel.

4. Finite Element Modeling

Sopontanaporn (2015) tested two types of test specimen CJ-1 and PJ-1. CJ-1 is the control specimen, beam-column joint that cast monolithically as shown in Figure 4. The end of beams are connected to a column by steel plate in PJ-1. The beams are cast to encase steel plate inserted in precast column as shown in Figure 6. To increase the bond between concrete and steel plate, 30 DB16 are inserted into hole of steel plate before the casting of concrete beam as shown in Figure 7. Finite element models of two type specimens are shown in Figures 5 and 7.

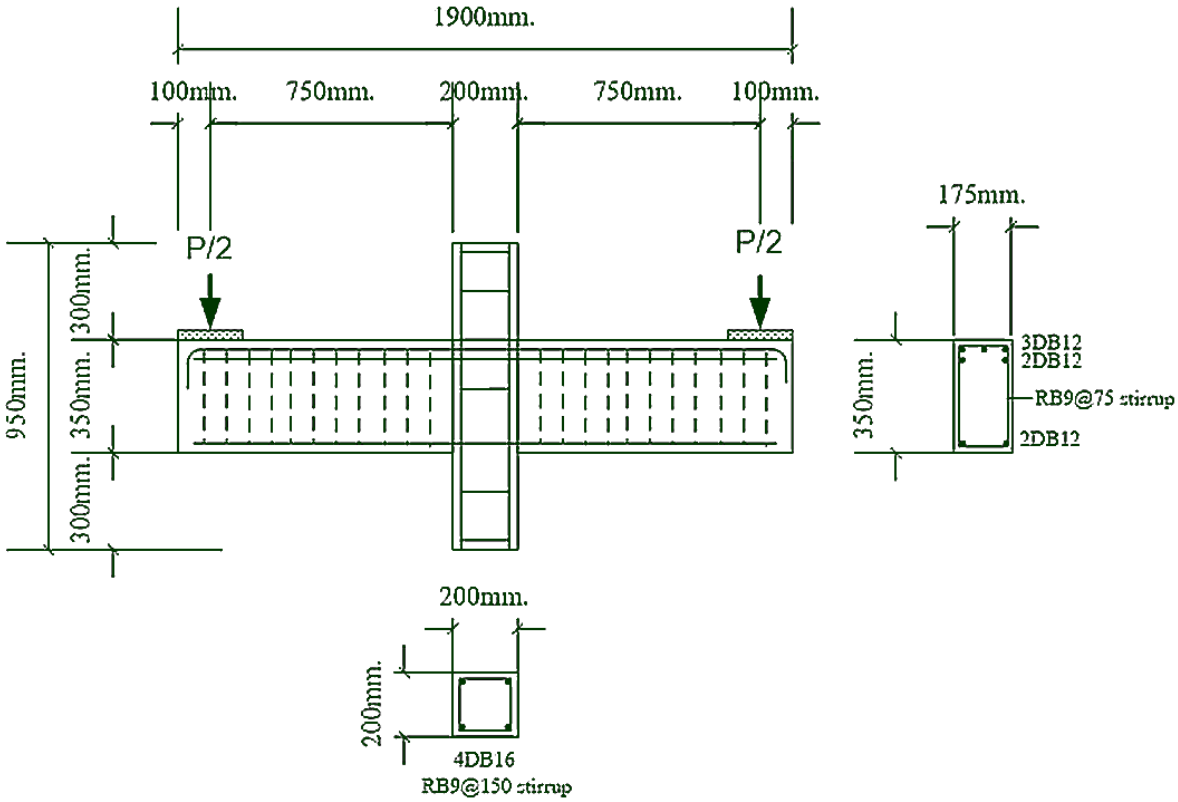


Figure 4: Details of control specimen CJ-1 of test performed by (Thaweesak, 2015).

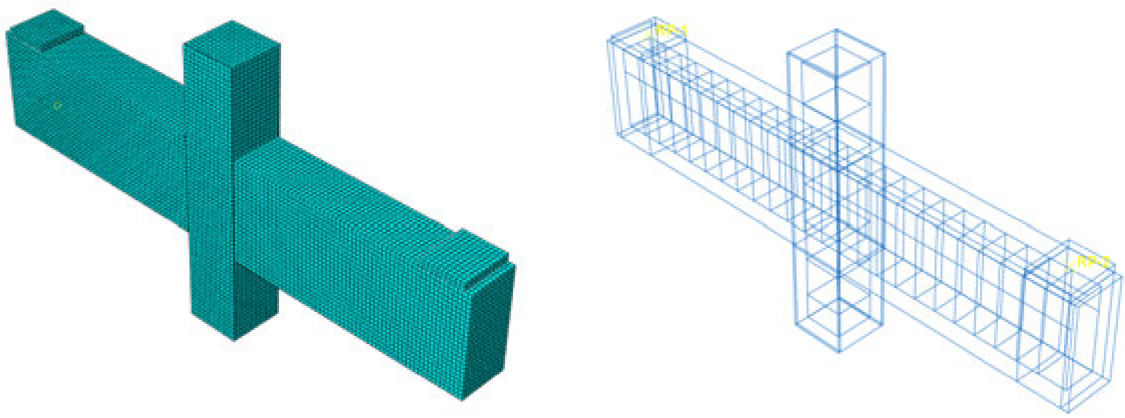


Figure 5: Finite element mesh of CJ-1.

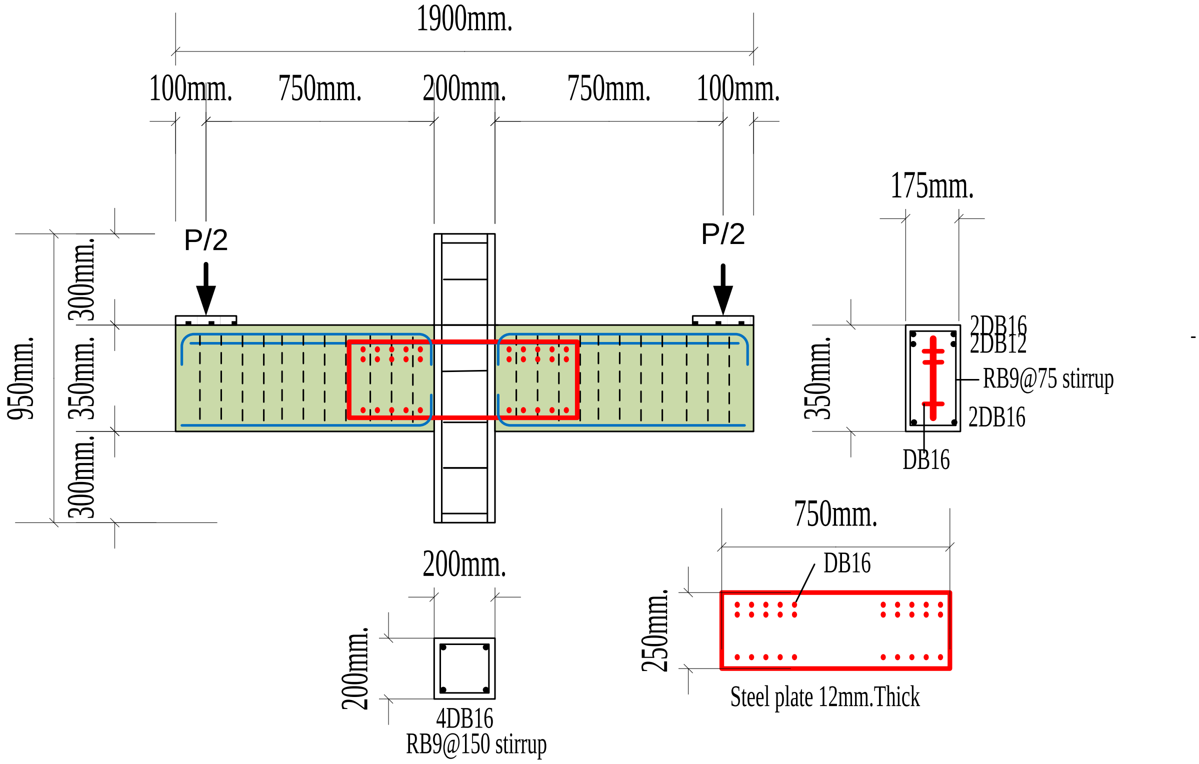


Figure 6: Details of specimen PJ-1 of test performed by (Sopontanaporn, 2015).

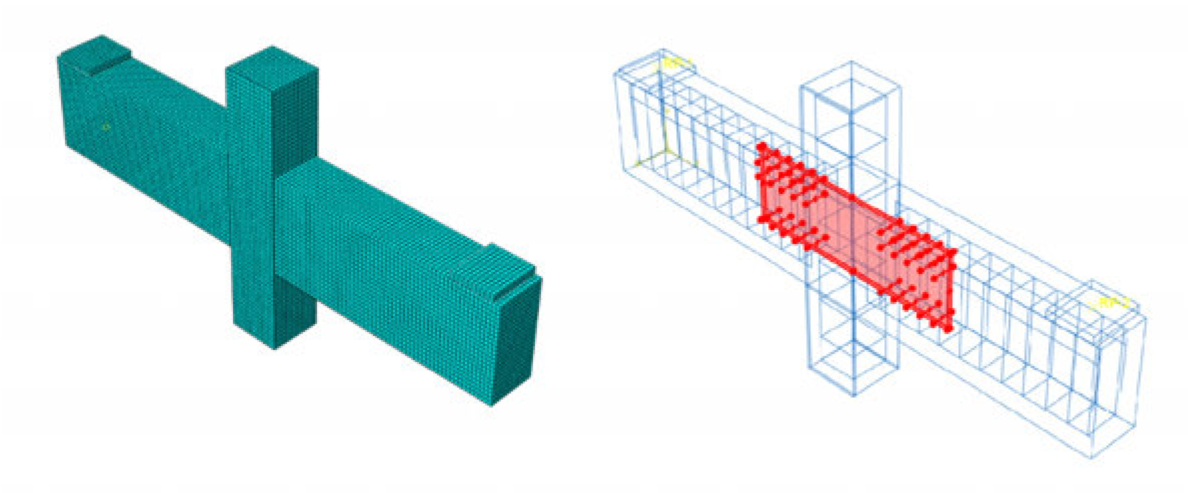


Figure 7: Finite element mesh of PJ-1.

Material properties of concrete, steel rebar and steel plate used in the analysis are shown in Tables 1 and 2.

Table 1: Concrete properties.

Concrete Ultimate strength	210 ksc
Young's modulus	25246.8 Mpa
Poisson's ratio	0.2
Dilation angle	31
Eccentricity	0.1
Compression plastic strain ratio	1.16
Invariant stress ratio	0.667
Viscosity	0

Table 2: Properties of steel rebar and steel plate.

Yield Strength of DB12, 16	595 Mpa
Yield Strength of RB9	290 Mpa
Yield Strength of Plate 9mm	480 Mpa
Yield Strength of Plate 12mm	337 Mpa
Poisson's ratio	0.3
Young's modulus	210000 Mpa

5. Analytical results

Effect of mesh size or amount of element on behavior under static loading of CJ-1 and PJ-1 are shown in figure 8 and 9. Convergence of the solution is found to be 50000 elements for CJ-1 and 21878 elements for PJ-1.

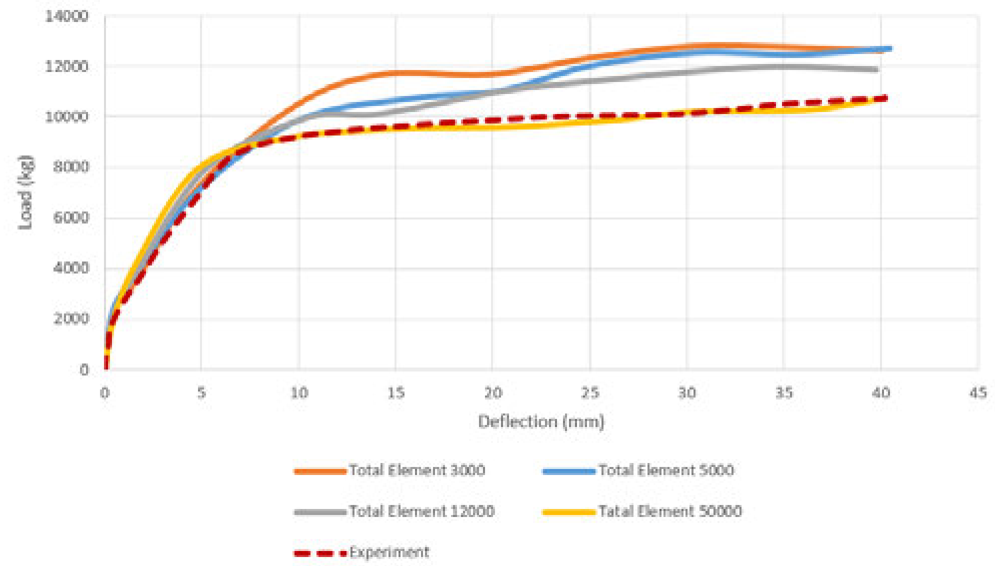


Figure 8: Mesh sensitivity on behavior under static loading of CJ-1

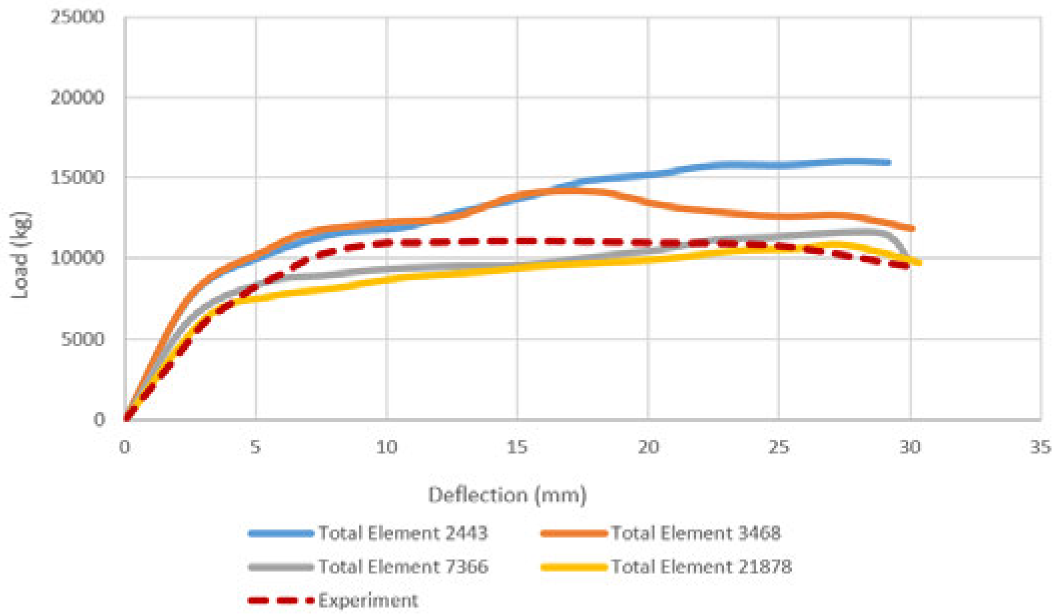


Figure 9: Mesh sensitivity on behavior under static loading of PJ-1.

The converge results of CJ-1 and PJ-1 are found to be good correlation with experimental results in both linear and nonlinear range. Strength and deformation of joints can be predicted accurately by nonlinear finite element model. Parametric study can be performed using this verified model

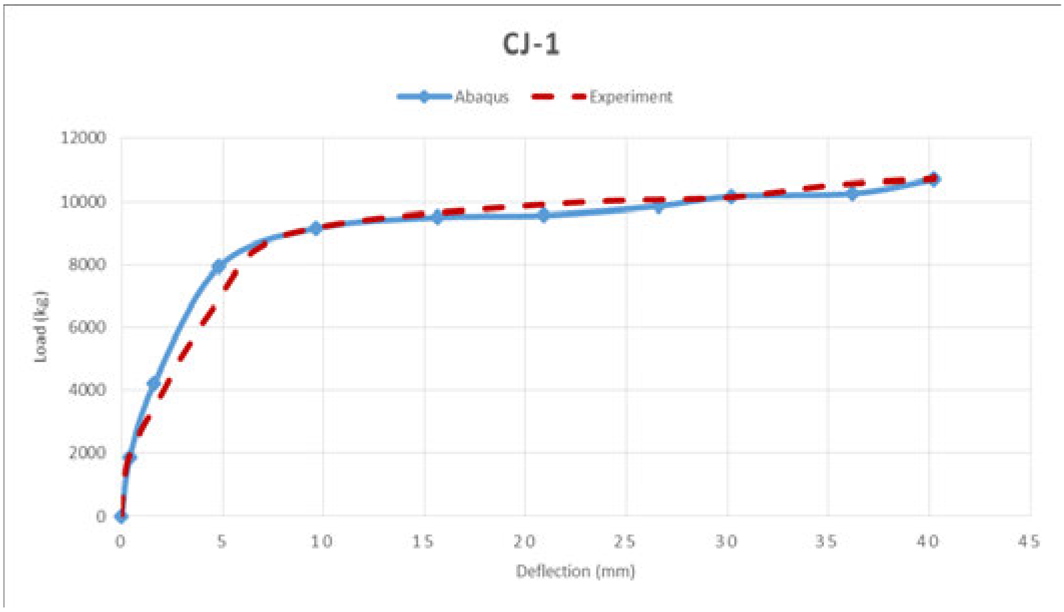


Figure 10: Comparison of test and analytical results of CJ-1

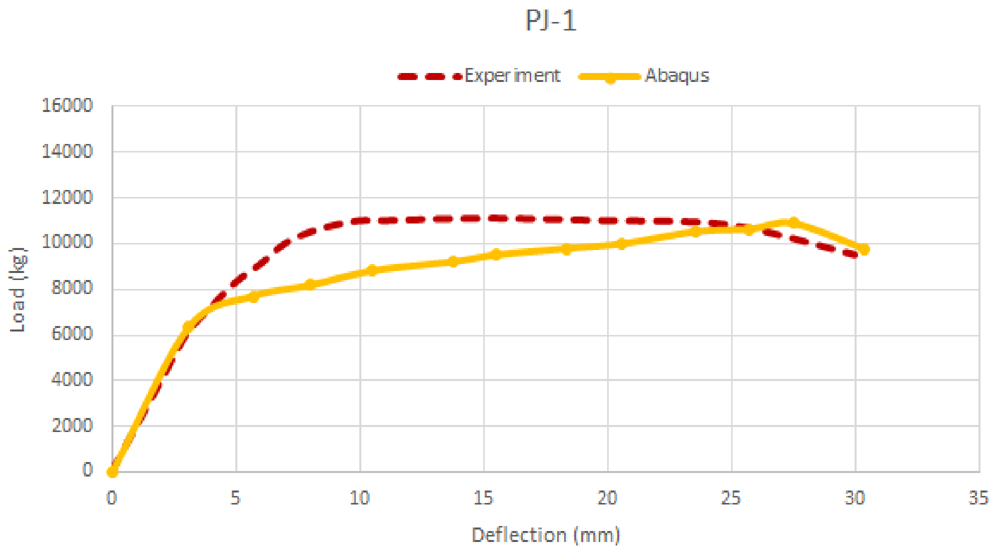


Figure 11: Comparison of test and analytical results of PJ-1

6. Conclusion

A nonlinear finite element model is developed to predict behavior of beam-column joints under static loading which were tested by the previous experiments performed by (Thaweesak, 2015). ABAQUS software was chosen to analyze using 8-node solid element, C3D8, for concrete and truss element, T3D2, for reinforcement. The damage plasticity model is applied to concrete and elastic-perfectly plastic model is applied to steel rebar and plate. Optimized number of elements leading to the convergence of a solution can be found by mesh sensitivity analysis. There are good correlation between converge finite element results and test results.

7. Acknowledgement

The authors would like to thank the Department of Civil Engineering. Thammasat University for providing facilities supports for this study.

8. References

ABAQUS. (2008). ABAQUS User's Manual 2008. ABAQUS Inc. Pawtucket. Rhode Island. USA.

Aydin Demir, (2016). 3D Numerical Modeling of RC Deep Beam Behavior by Nonlinear Finite Element Analysis. Disaster Science and Engineering, p. 13-18, 2(1)

Bangash, M.Y.H. (1989). Concrete and Concrete Structures. Numerical Modeling and Applications. London. Elsevier Science Publishers Ltd.

Deng Sihua, (2015). Nonlinear Analysis of Reinforced Concrete Beam Bending Failure Experimentation Based on ABAQUS. Beijing University of Civil Engineering and Architecture. Beijing China.

Long, X. (2014). Modelling of Two Dimensional Reinforced Concrete Beam-Column Joints Subjected to Monotonic Loading. School of Mechanics. Civil Engineering and Architecture. Northwestern Polytech.

Rasoul Nilforoush Hamedani, (2012). Numerical Evaluation of Structural Behavior of the Simply Supported FRP-RC Beams. Master of Science Thesis. Stockholm. Sweden.

Sopontanaporn, Taweesak. (2015). Behaviors of Precast Concrete Beam to Column Connections under Static Load. Thammasat University. Thailand.

Sinaei, Hamid. (2011). Numerical investigation on exterior reinforced concrete Beam-Column joint strengthened by composite fiber reinforced polymer (CFRP). Department of Civil Engineering. University of Malaya. 50603 Kuala Lumpur. Malaysia.

Wahalathantri, B. Thambiratnam, D. Chan, T. and Fawzia, (2011). A material model for flexural crack simulation in reinforced concrete elements using ABAQUS. Proceedings of the First International Conference on Engineering. Designing and Developing the Built Environment for Sustainable Wellbeing, pp. 260–264.

Atthaphon Atichat is a Master Candidate of Engineering (Civil Engineering) at Thammasat University. He received his Bachelor of Engineering in Civil Engineering and Construction Management from Thammasat University, Thailand. He is interested in concrete structures with prestressed technology.

Dr. Sayan Sirimontree earned his bachelor degree from Khonkaen University Thailand, master degree in Structural Engineering from Chulalongkorn University Thailand and PhD in Structural Engineering from Khonkaen University Thailand. He is an Associate Professor at Thammasat University Thailand. He is interested in durability of concrete, repair and strengthening of reinforced and prestressed concrete structures.

Dr. Boonsap Witchayangkoon is an Associate Professor of Department of Civil Engineering at Thammasat University. He received his B.Eng. from King Mongkut's University of Technology Thonburi with Honors. He earned his PhD from University of Maine, USA in Spatial Information Science & Engineering. Dr. Witchayangkoon current interests involve applications of emerging technologies to engineering.