Acta Polytechnica CTU Proceedings **33**:8–14, 2022
https://doi.org/10.14311/APP.2022.33.0008

3D FINITE ELEMENT SIMULATION OF REINFORCED CONCRETE COLUMN STRENGTHENED BY PARTIALLY CONFINED CARBON FIBRE REINFORCED POLYMER UNDER CONCENTRIC LOADING

Fariz Aswan Ahmad Zakwan[a], Ruqayyah Ismail[a,*], Hazrina Ahmad[a], Raizal Saifulnaz Muhammad Rashid[b]

[a] *School of Civil Engineering, College of Engineering, Universiti Teknologi MARA, Cawangan Pulau Pinang, Kampus Permatang Pauh, 13500 Permatang Pauh, Pulau Pinang, Malaysia*

[b] *Department of Civil Engineering, Faculty of Engineering, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia*

[*] corresponding author: fruqayyah812@uitm.edu.my

Abstract.

In this paper, numerical simulation initiated to investigate the mechanical response of reinforced concrete (RC) column strengthened by partially confined carbon fiber reinforced polymer (CFRP). An experimental investigation of the circular hollow unconfined and confined RC column enhances with a partially CFRP sheet has been performed to investigate its behavior under concentric loading on top of the RC column. From the experimental data, the critical buckling failure of the RC column observed. Therefore, the prediction behavior of the RC column strengthened with partially confined CFRP under concentric loading was made with the application of the general-purpose of ABAQUS finite element simulation. From the outcome of the simulation results, the predicted buckling failure compared and validated against the experimental data. The predicted model discovered to agrees well with the experimental work output.

Keywords: ABAQUS, CFRP, finite element, RC column.

1. Introduction

There is a need to incorporate a finite element analysis program to predict the structural behavior of unconfined and confined RC columns under specific types of loading. Additional material added to strengthen the unconfined RC column under those applied loading if often quite costly to implemented [1]. This scenario happened is since the more additional material used to strengthen the column, the more cost implication will involve. In previous research, the experimental program of unconfined and confined RC columns reinforced by partially enclosed CFRP under concentric loading as been completed successfully [2–4]. Therefore, the general purpose of the ABAQUS finite element simulation program introduced to validate the mechanical response output from the experimental results [5]. From then onwards, a different configuration of the CFRP layout onto the RC column can be implemented to investigate its behavior under a finite element approach. Besides, implementing finite element software would help in reducing the experimental program cost in buying CFRP to be used for the RC column. The mechanical behavior of the RC column can be predicted due to this various configuration of the CFRP strengthening method approach.

2. Methods

2.1. ABAQUS nonlinear analysis of circular hollow RC column

Three types of nonlinearities exist in concrete materials, namely material nonlinearity, geometric nonlinearity, and contact nonlinearity [6–11]. For material nonlinearity, the material used to create a concrete model behaves as a nonlinear stress-strain relationship. It is much more reliable to incorporate nonlinear properties of elements instead of just adopting linear features due to its precise computation of stress distributions, which enable us to verify any load-bearing capacity aspects of the model such as plastic collapse. For instance, ABAQUS provides the constitutive model to predict the nonlinear characteristic of the concrete material. There is three constitutive concrete model available that available to be used during the plastic stage of nonlinear characteristic, namely concrete smeared cracking, concrete damage plasticity (CDP), and cracking model. In this current research, the concrete damage plasticity (CDP) constitutive model adopted in the finite element analysis (FEA) simulation due to the complexity of the failure mode of the RC column under axial load.

For the second geometric nonlinearity material, large deflection is the most common term to define this type of nonlinearity behavior [7, 8, 10]. The deformation translated into rigid body translations and

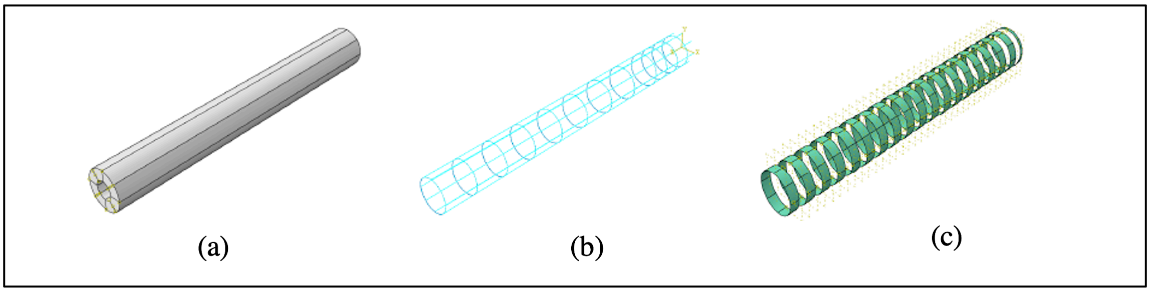

Figure 1. ABAQUS FEA model; (a) concrete model (b) reinforcement model (c) CFRP sheet model.

Dilation angle	Eccentricity	f_{bo}/f_{co}	K	Viscosity Parameter
35	0.1	1.16	0.6667	0

Table 1. Concrete Damage Plasticity value.

rotations, or substantial strains, or a mixture both. It is very crucial to incorporate significant deformation effect onto the final geometry or position of any component to avoid any over-predicted displacement. Also, activating geometric linearity will score a slight increase in computational work when material contact nonlinearity exists in the model.

In the third contact linearity material, it refers to boundary conditions between multiple components that exist in the model. In this boundary condition nonlinearity, the stiffness of the assembly my change when two or more parts still connected or split from initial contact.

2.2. Finite element model and material properties of circular hollow RC column

For the finite element model, solid element, truss element and shell element were selected during the numerical simulation stage for material concrete, reinforcement bars and CFRP sheet respectively. For solid element, an eight-noded linear brick solid element (C3D8) type applied for the RC column. For transverse and longitudinal reinforcement bars, a two-node linear 3-D truss element (T3D2) used for modeling purposes. Four-node doubly curved shell element employed for the CFRP sheet. All the models illustrated in Figure 1. The constitutive model used for concrete materials is Concrete Damaged Plasticity (CDP). The CDP value shown in Table 1. Meanwhile, for the CFRP sheet model, Hashin damage criteria were applied in the model to create typical snap patterns of composite CFRP material with concrete.

2.3. Finite element analysis of circular hollow RC column

The circular hollow unconfined RC column measured 2 m long vertically. The details of the dimensions and reinforcement illustrated, as shown in Figure 2. There are two experimental programs conducted previously, namely unconfined circular hollow RC column and confined circular hollow RC column strengthen with partially CFRP sheet. Both RC column sample was subjected full axial load acting on top of the RC column. Figure 3 represents the unconfined circular hollow RC column modeled in ABAQUS FEA. It can appear that axial load was acting on top of the column model while the full fixed boundary condition applied at the bottom surface of the column.

3. Results

3.1. Experimental results

The experimental results of the stress-strain diagram for circular hollow unconfined and confined RC column with CFRP wrapping under full axial load illustrated as in Figure 5. A full axial load applied on top of the RC column for both unconfined and confined RC columns, as shown in Figure 3 and Figure 4, respectively. It can be seen from Figure 5 that the RC column strengthens with CFRP can sustain much higher stress as compared to the regular unconfined RC column. The stress-strain value taken at the middle section along the RC column. When the strain value approximately 0.015, the predicted stress value of the confined RC column with the CFRP sheet is performing better than the unconfined RC column with 2 MPa increased to become 20 MPa. The unconfined RC column only manages to achieve 18 MPa. The improved confined circular hollow RC column strengthen with CFRP were able to sustain a higher load concerning higher strain value.

3.2. Finite element simulation results

The predicted outcome of the stress-strain diagram for both circular hollow unrestrained and restrained RC columns shown in Figure 6. From the figure, the confined circular hollow RC column with CFRP exhibits a stress value of 27 MPa, which is higher than the unconfined RC column of 18 MPa. Similar to the previous chapter, the predicted stress value taken at the middle section along the RC column.

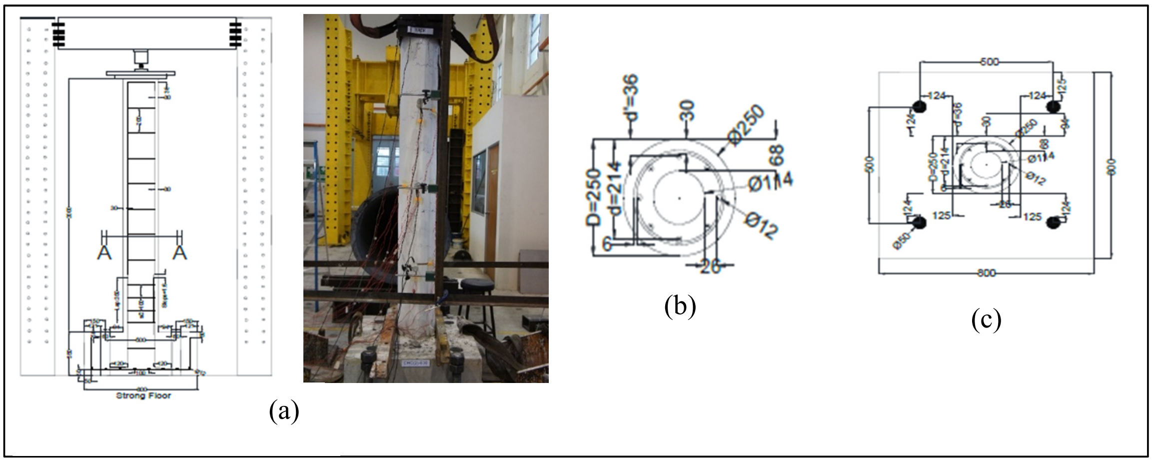

Figure 2. Circular hollow unconfined RC column detail specification; (a) Front view (b) A-A section view (c) Plan view [2].

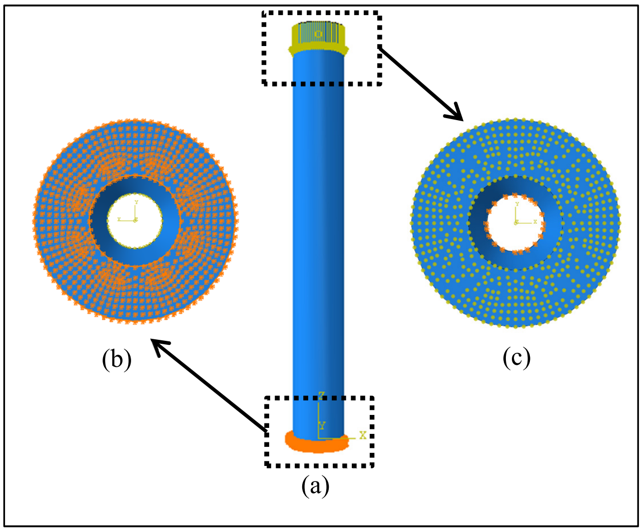

Figure 3. Circular hollow unconfined RC column modeled in ABAQUS FEA; (a) Front view (b) bottom plan view showing support boundary condition (c) Upper plan view showing full axial load imposed on top of the column.

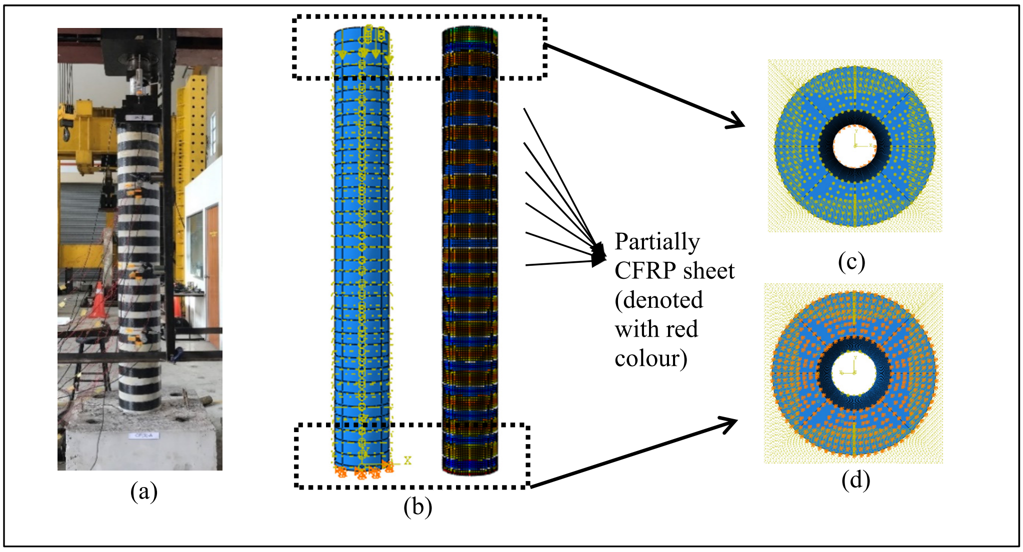

Figure 4. Circular hollow confined RC column strengthen with CFRP sheet experimental ser-up and modeled in ABAQUS FEA; (a) Front view from experimental set-up (b) Front view (c) Upper plan view showing full axial load imposed on top of the column (d) bottom plan view showing support boundary condition.

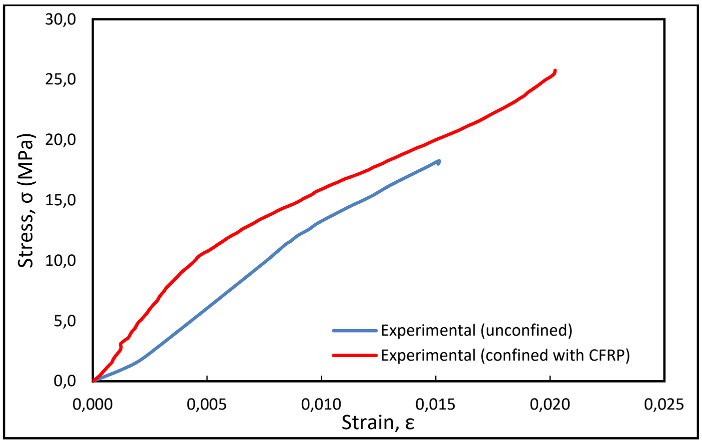

FIGURE 5. Stress-strain relationship of circular hollow unconfined and confined RC column with CFRP under full axial load.

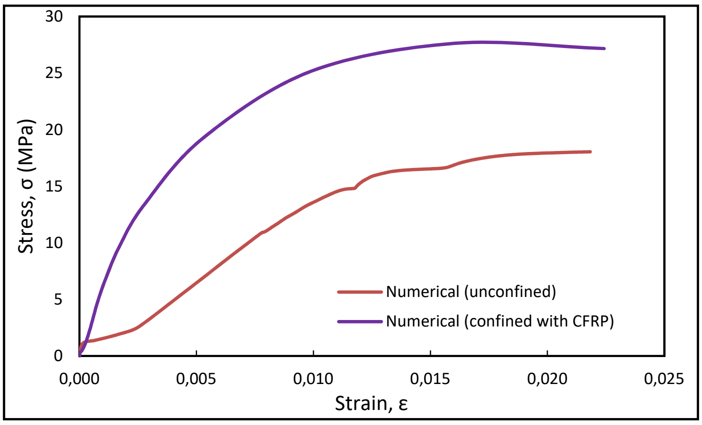

FIGURE 6. Predicted stress-strain relationship of circular hollow unconfined and confined RC column with CFRP under full axial load.

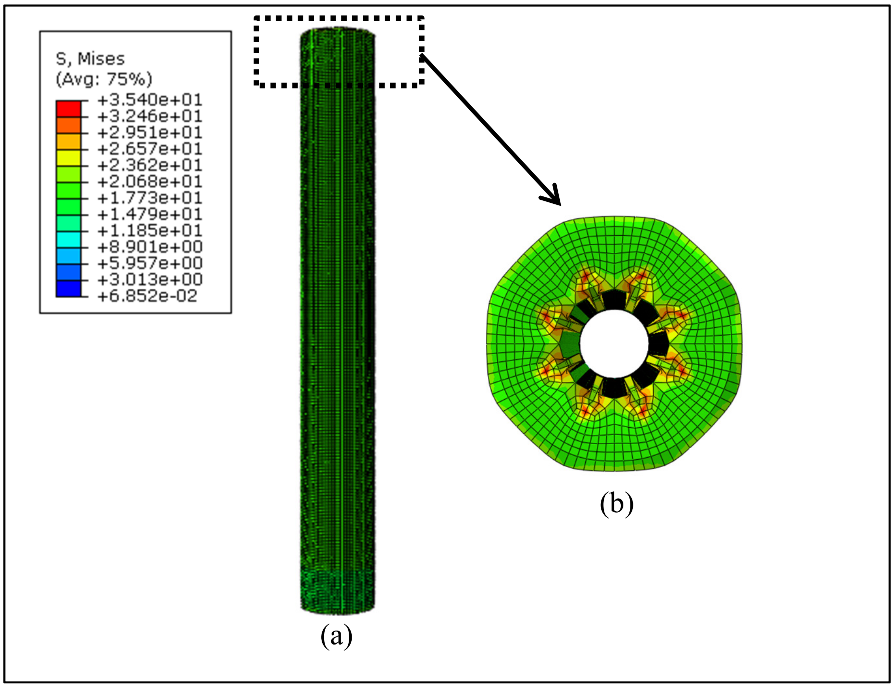

FIGURE 7. Predicted von Mises stress of circular hollow unconfined RC column (at the end of simulation analysis); (a) Front view (b) Upper plan view showing failure behavior on top of the column.

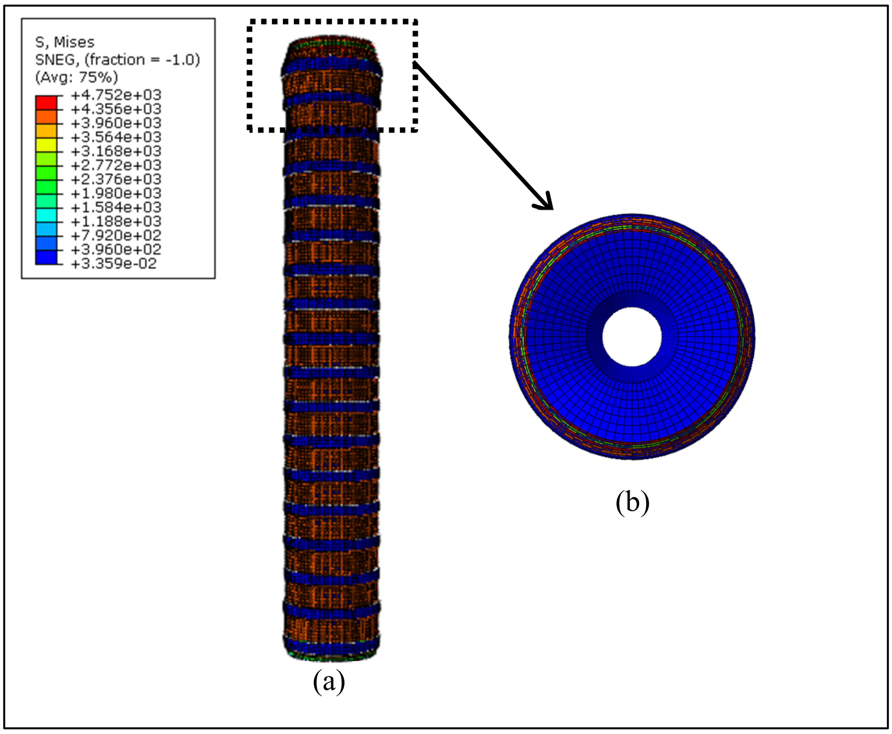

Figure 8. Predicted von Mises stress of circular hollow confined RC column strengthen with partial CFRP sheet (at the end of simulation analysis); (a) Front view (b) Upper plan view showing failure behavior on top of the column.

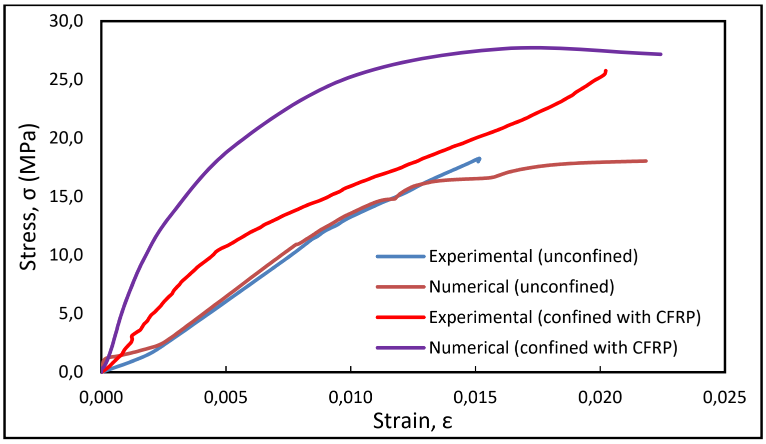

Figure 9. Validation process stress-strain relationship of circular hollow unconfined and confined RC column with CFRP under full axial load.

4. Discussions

From the experimental and numerical results shown in the previous chapter, the validation process shown in Figure 9. For the circular hollow unconfined RC column, the stress-strain behavior agrees well between both experimental and numerical approaches. However, numerical results outcome of circular hollow confined RC column with CFRP sheet exhibit over predicted stress-strain value up to strain value of 0.02. It can be seen that ABAQUS FEA can consistently able to perform and verify the real experimental results, but there are some results display variances between the values calculated by the numerical analysis and experimental output. The reason may be due to:

- Finite element simulation is the best tool to predict any structural behavior due to axial load. However, there are several constitutive concrete models available that may correspond with the RC column model. The different constitutive models may contribute to different failure behavior to the RC column. Therefore, the most appropriate constitutive concrete model needs to be selected to achieve the most reliable failure mode of the RC column at the end of numerical simulation.
- Apart from that, the connection and bonding between the CFRP sheet and the RC column model also need to be taken into account in the numerical simulation. The relationship between both material also may contribute to the failure mode of the RC column, as shown in the previous validation process.
- Other important factors also may affect the structural behavior of the RC column, namely convergence of finite element simulation, the global meshing size of the RC column model, parameter values inserted in material properties of finite element simulation, and loading conditions applied onto the RC column.

5. Conclusions

This research paper represents the verification process between the actual experimental results and predicted numerical analysis. Applying ABAQUS FEA was able to verify the actual experimental results of the unconfined circular hollow RC column. However, when adopting additional material of the CFRP sheet onto the RC column, the predicted behavior was over anticipated as compared to experimental results. Therefore, ABAQUS FEA proves to be the most reliable method in predicting the structural behavior of the RC column by selecting the most appropriate constitutive concrete model.

Acknowledgements

This work fully sponsored by the Universiti Putra Malaysia (UPM) research grant under the Putra Grant initiative (Grant Reference No: 9439300). The authors would like to express their most profound appreciation to UPM for providing all facilities and human resources during the laboratory work phase. Full gratitude dedicated to Universiti Teknologi MARA, Cawangan Pulau Pinang, Kampus Permatang Pauh, Pulau Pinang.

References

[1] L. N. K. Sai. Madupu, K. S. Sai Ram. Strengthening of Axially Loaded Reinforced Concrete Rectangular Columns with GFRP Strips. *Materials Today: Proceedings* **18**:3758-65, 2019. https://doi.org/10.1016/j.matpr.2019.07.311.

[2] R. Ismail, R. S. M. Rashid, F. A. A. Zakwan, et al. Experimental study of circular hollow reinforced concrete column strengthened with partial Carbon Fibre Reinforced Polymer (CFRP) confinement. *IOP Conference Series: Materials Science and Engineering* **615**(1), 2019. https://doi.org/10.1088/1757-899x/615/1/012075.

[3] G. M. Al-Mekhlafi, M. A. Al-Osta, A. M. Sharif. Behavior of eccentrically loaded concrete-filled stainless steel tubular stub columns confined by CFRP composites. *Engineering Structures* **205**, 2020. https://doi.org/10.1016/j.engstruct.2019.110113.

[4] M. Miralami, M. R. Esfahani, M. Tavakkolizadeh. Strengthening of circular RC column-foundation connections with GFRP/SMA bars and CFRP wraps. *Composites Part B: Engineering* **172**:161-72, 2019. https://doi.org/10.1016/j.compositesb.2019.05.063.

[5] S. Peng, C. Xu, M. Lu, et al. Experimental research and finite element analysis on seismic behavior of CFRP-strengthened seismic-damaged composite steel-concrete frame columns. *Engineering Structures* **155**:50-60, 2018. https://doi.org/10.1016/j.engstruct.2017.10.065.

[6] M. Lezgy-Nazargah, P. Vidal, O. Polit. A 1D nonlinear finite element model for analysis of composite foam-insulated concrete sandwich panels. *Composite Structures* **210**:663-75, 2019. https://doi.org/10.1016/j.compstruct.2018.11.087.

[7] Q. Huang, E. Hamed. Nonlinear finite element analysis of composite precast concrete sandwich panels made with diagonal FRP bar connectors. *Composite Structures* **212**:304-16, 2019. https://doi.org/10.1016/j.compstruct.2019.01.019.

[8] Q. Huang, E. Hamed. Nonlinear finite element analysis of composite precast concrete sandwich panels made with diagonal FRP bar connectors. *Composite Structures* **212**:304-16, 2019. https://doi.org/10.1016/j.compstruct.2019.01.019.

[9] J. D. Nzabonimpa, W.-K. Hong, J. Kim. Nonlinear finite element model for the novel mechanical beam-column joints of precast concrete-based frames. *Computers Structures* **189**:31-48, 2017. https://doi.org/10.1016/j.compstruc.2017.04.016.

[10] W. Zhou and W. Yan, Refined nonlinear finite element modelling towards ultimate bending moment calculation for concrete composite beams under negative moment, Thin Walled Struct., vol. 116, no. February, pp. 201-211, 2017.

[11] M. R. Javanmardi, M. R. Maheri. Extended finite element method and anisotropic damage plasticity for modelling crack propagation in concrete. *Finite Elements in Analysis and Design* **165**:1-20, 2019. https://doi.org/10.1016/j.finel.2019.07.004.