# Predicting Nonlinear Behavior and Stress-Strain Relationship of Rectangular Confined Reinforced Concrete Columns with ANSYS

**Tavio, T.[1] and Tata, A.[2]**

**Abstract**: This paper presents a nonlinear finite element modeling and analysis of rectangular normal-strength reinforced concrete columns confined with transverse steel under axial compressive loading. In this study, the columns were modeled as discrete elements using ANSYS nonlinear finite element software. Concrete was modeled with 8-noded SOLID65 elements that can translate either in the $x$-, $y$-, or $z$-axis directions from ANSYS element library. Longitudinal and transverse steels were modeled as discrete elements using 3D-LINK8 bar elements available in the ANSYS element library. The nonlinear constitutive law of each material was also implemented in the model. The results indicate that the stress-strain relationships obtained from the analytical model using ANSYS are in good agreement with the experimental data. This has been confirmed with the insignificant difference between the analytical and experimental, i.e. 5.65 and 2.80 percent for the peak stress and the strain at the peak stress, respectively. The comparison shows that the ANSYS nonlinear finite element program is capable of modeling and predicting the actual nonlinear behavior of confined concrete column under axial loading. The actual stress-strain relationship, the strength gain and ductility improvement have also been confirmed to be satisfactorily.

**Keywords**: ANSYS; confinement; ductility; stress-strain relationship; nonlinear finite element analysis; nonlinear behavior; reinforced concrete columns; strength.

## Introduction

One of several reasons that cause the collapse of a multi-story building or bridge structure is the failure of the supporting members to withstand the earthquake loading. The failure of these members is mostly due to the lack of shear-resisting capacity and insufficient ductility provided by little amount of transverse steel. It is well known that the ductility of a reinforced concrete column plays a very important role in preventing such a failure. That is why the study on the ductility of a reinforced concrete column has been developing at a fast pace in the last two decades in many countries worldwide. One of the effective ways to improve the ductility of a column is by introducing sufficient transverse steel as confining steel for concrete core in a column. This effort is primarily intended to delay the sudden collapse of a column and force it further to fail in a ductile manner.

[1] Department of Civil Engineering, Sepuluh Nopember Institute of Technology (ITS), Surabaya, Indonesia
Email: tavio@its.ac.id
[2] Department of Civil Engineering, Khairun University, Ternate, Maluku Utara, Indonesia
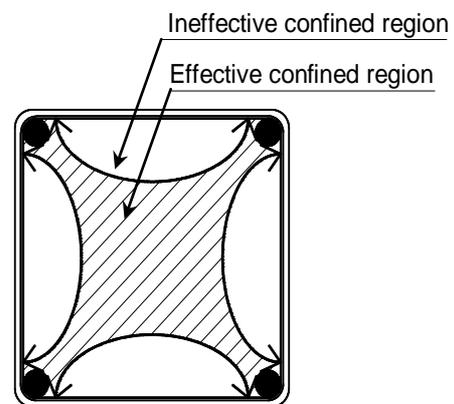
**Figur 1.** Effective and ineffective confined regions in a concrete core of a square column cross section produced by the arching action due to the existence of rectilinear confinement

The effectiveness of confinement depends on the uniformity of the stress occurred around the perimeter interface between the confining steel and concrete core. In rectilinear column section, the most effective confined regions are only located at the four corners of the confining steel as shown in Fig. 1. Fig. 1 also shows the ineffective and effective regions produced by the arching action due to the existence of rectilinear confining steel in a concrete core of a square column section. The effect of confinement in a reinforced concrete column can be considerably increased if:
(1) the spacing of transverse steel is denser;

(2) more number and well distributed longitudinal steel is used around the perimeter of the column section; and

(3) more number and well-distributed crossties are provided in the concrete core.

Numerous researches have been conducted earlier to study the effectiveness of confinement in improving the ductility of reinforced concrete columns. Some experimental tests carried out by several previous researchers include the studies conducted by Cusson and Paultre [1], Saatcioglu and Razvi [2], and Assa, Nishiyama, and Watanabe [3]. Cusson and Paultre [1] carried through the experimental tests on square short columns with high-strength concrete and proposed a stress-strain model of ductile confined concrete. Saatcioglu and Razvi [2] tested and observed the ductile behavior of confined concrete columns with the strength up to 120 MPa. Assa, Nishiyama, and Watanabe [3] also conducted similar tests on confined circular and square short columns and examined their ductile behaviors, and there are still many more studies conducted by others [4-11]. The numerical approaches conducted by previous researchers were mostly developed on empirical basis. This is due to the complex parameters involved in deriving the constitutive law of confined concrete. Though, some researchers had made many attempts to come up with an accurate analytical stress-strain model of confined concrete, they always ended up with a fine-tuning measure in matching up the analytical results with the experimental data obtained from their tests.

The authors fully realize that the experimental program is one of the best ways to adjust a model in order to achieve an acceptable accuracy for practical usage. This sort of effort, however, is often very costly and time consuming; besides it still depends on the availability and accuracy of the test apparatus and instrument. In addition, the use of the proposed model is often limited to a certain extent of the test data where they are calibrated with.

In this paper, the authors propose to use ANSYS [12], which is capable of modeling the nonlinear behavior of reinforced concrete beams [13-15], for predicting the actual stress-strain relationship of both unconfined and confined rectangular concrete columns with various spacings of transverse steel under axial concentric loading. However, none of the works [13-15] conducted previously using ANSYS includes reinforced concrete columns confined by transverse steel or stirrups. The proposed procedure has been verified with four column specimens confined by various spacing of stirrups representing light to heavy confinement tested by Hoshikuma [16]. The analytical stress-strain curves obtained

from the proposed procedure are shown to be in close agreement with the experimental data from Hoshikuma [16].

## Research Significance

Modeling the constitutive law of confined reinforced concrete columns based on the empirical approach can sometimes be inaccurate or limited to a narrow range of available experimental data. The tests are also very expensive and sometimes time consuming. The applicability of the test data mainly depends on the accuracy of the test apparatus and the supporting instruments implemented during the test. Hence, it is deemed necessary to have another option of modeling the stress-strain relationship of confined concrete without deploying an empirical approach in the modeling. One of the suitable software that can be utilized to describe the actual nonlinear behavior of confined concrete columns under axial loading is ANSYS [12]. This is because ANSYS is capable of analyzing the nonlinear behavior of a combination between 3D SOLID and LINK elements in a structure based on the finite element procedure. With this option, researchers or design engineers can confidently predict in advance the actual behavior of various confined concrete columns not only in the linear-elastic region, but furthermore also in the nonlinear post-elastic region. The authors wish that this economical procedure can be used to provide an alternative tool for researchers or structural engineers in investigating various types of structural concrete elements in the future.

## Finite Element Procedure

The finite element procedure implemented in this study is developed using the available element types from ANSYS element library [12]. The concrete is modeled using SOLID65 element type, whereas the steel for longitudinal and transverse reinforcements is modeled with LINK8 element type. SOLID65 is selected because this concrete material model can predict the failure of brittle materials by adopting the constitutive model of concrete [12]. Both cracking and crushing failure modes can be accounted for [12]. LINK8 in ANSYS is selected because this steel material model can take into account the complete stress-strain relations of materials [12]. Both yielding and strain-hardening failure modes can be accounted for [12]. The concepts used are directly applicable to 3D SOLID elements [12]. By adopting and combining these two element types, the reinforced concrete column model was developed.

The column model was subjected to an axial compressive loading on their top face simulating the actual loading applied in the tests [16], while the

bottom side was restrained. The loading procedure can be elaborated in the following sequence: (1) for the ascending branch (up to peak stress): the column model is subjected to a step-by-step incremental axial pressure on its entire top surface; then (2) for the descending portion (beyond the peak stress): the loading was then switched into the displacement-mode control by applying a step-by-step incremental displacement on its top surface.

To obtain an efficient solution, the column was modeled in a quarter following the symmetrical lines of its cross section as shown in Fig. 2. The two sides along the symmetrical lines of a quarter were restrained to simulate the actual behavior of the full-size column, and thus, maintaining the accuracy of analysis of the model.
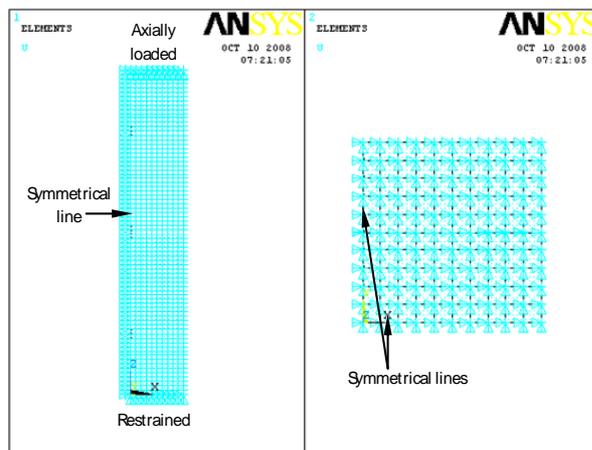


**Figur 2.** Boundary conditions of a symmetrical quarter of a column model: (a) elevation; (b) cross section

## Determination of Model

The analytical models were constructed according to the actual column specimens in literature [16] as shown in Fig. 3. The column models had a typical cross section of 500 mm × 500 mm with the height of 1500 mm as listed in Table 1. The concrete cover was 20 mm. The first column specimen was made from plain concrete, namely specimen LS0 (Fig. 3). The three remaining column specimens had various spacings and diameters of transverse steel, i.e. specimens LS1, LS2, and LS3 (see Fig. 3). The mechanical properties of each specimen used for validation in this study were adopted in developing the analytical models to better reflect the actual behavior of each column specimen.

## Material Properties

The constitutive laws used in the proposed analytical model were developed for two materials of column specimens, namely concrete and steel. Since the proposed procedure is intended as an alternative way for predicting the actual nonlinear behavior of both unconfined and confined reinforced concrete columns prior to conducting the experimental program, the following analytical models are selected in lieu of the actual experimental data. The experimental data is assumed unavailable at this stage, even though it might produce better prediction to the actual behavior of the column specimens. The analytical model proposed by Popovics [17,18] to represent the stress-strain relationship of concrete was adopted in this study. For reinforcing steel, the analytical model was that proposed by Park and Paulay [19]. The element type used to model each material is those from the ANSYS element library [12] and summarized in Table 2. The concrete is modeled using SOLID65 element, whereas the steel reinforcement is modeled with LINK8 element.

To develop the proposed analytical model in the ANSYS software, the following data is required to be prepared for the input data prior to the analysis. The material properties of each element type can be elaborated in the following details to reflect the

**Table 1.** Summary of geometrical and mechanical properties of the column specimens

| Column ID | Cross Section (mm) | Height (mm) | $f_c'$ (MPa) | $\rho$ (%) | $f_{y\ell}$ (MPa) | $f_{yh}$ (MPa) | Spacing,s (mm) | Volumetric ratio (%) |
|---|---|---|---|---|---|---|---|---|
| (1) | (2) | (3) | (4) | (5) | (6) | (7) | (8) | (9) |
| LS0 | 500 × 500 | 1500 | 24.3 | — | — | — | — | — |
| LS1 | 500 × 500 | 1500 | 24.3 | 0.95 | 295 | 235 | 60 | 1.73 |
| LS2 | 500 × 500 | 1500 | 24.3 | 0.95 | 295 | 235 | 75 | 2.19 |
| LS3 | 500 × 500 | 1500 | 24.3 | 0.95 | 295 | 235 | 40 | 2.60 |

Note: $f_c'$ = compressive strength of concrete; $\rho$ = ratio of longitudinal steel; $f_{y\ell}$ = yield strength of longitudinal steel; $f_{yh}$ = yield strength of transverse steel.

**Table 2.** Material types for modeling the column specimens

| Material | ANSYS Elemen Type |
|---|---|
| Concrete | Solid65 |
| Steel Reinforcement | Link8 |
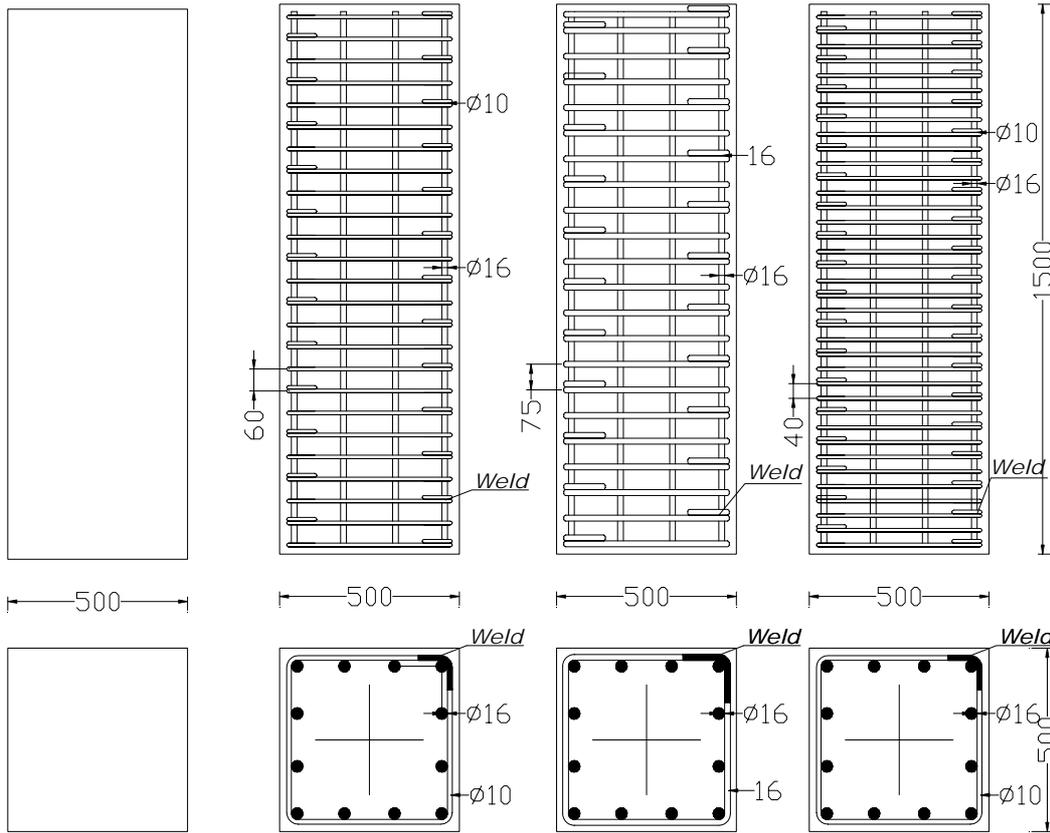
**Figure 3.** Geometrical properties of column specimens LS0, LS1, LS2, and S3 used for model validation

actual mechanical and physical properties of the column specimens. Following is the summary of the concrete properties required for input data:

1) stress-strain relationship of concrete ($\sigma_c$–$\varepsilon_c$);
2) modulus of elasticity of concrete ($E_c$);
3) specified compressive strength of concrete ($f_c' = 24.3$ MPa);
4) modulus of rupture of concrete ($f_r$);
5) poisson ratio of concrete ($\nu_c = 0.2$) ;
6) concrete density ($\gamma_c$);

and for the reinforcing steel, it can also be summarized as follows:

1) stress-strain relationship of reinforcing steel ($\sigma_s$–$\varepsilon_s$);
2) specified yield strength of longitudinal steel ($f_{y\ell} = 295$ MPa);
3) specified yield strength of transverse steel ($f_{yh} = 235$ MPa);
4) modulus of elasticity of reinforcing steel ($E_s$);
5) Poisson ration of reinforcing steel ($\nu_s = 0.3$);
6) steel density ($\gamma_s$).

The stress-strain relationship of concrete proposed by Popovics [17,18] as a part of the constitutive laws adopted in the proposed model can be described by the following equations:
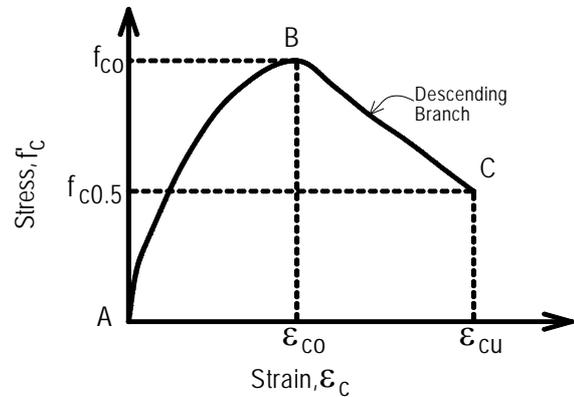


**Figure 4.** Stress-strain relationship of concrete proposed by Popovics [17,18]

$$f_c = \frac{n f_{co}\left(\dfrac{\varepsilon_c}{\varepsilon_{co}}\right)}{n-1+\left(\dfrac{\varepsilon_c}{\varepsilon_{co}}\right)^{nk}} \tag{1}$$

For region AB ($0 \le \varepsilon_c \le \varepsilon_{co}$),

$$k = 1 \quad \text{if} \quad \frac{\varepsilon_c}{\varepsilon_{co}} \le 1 \tag{2}$$

For region BC ($\varepsilon_c > \varepsilon_{co}$),

$$k = 0.67 + \frac{f_c'}{62} \text{ MPa} \quad \text{if } \frac{\varepsilon_c}{\varepsilon_{co}} > 1 \tag{3}$$

$$E_c = 3320\sqrt{f_c'} + 6900 \text{ MPa} \tag{4}$$

$$\varepsilon_o = \frac{f_c'}{E_c}\frac{n}{n-1} \tag{5}$$

$$n = 0.8 + \frac{f_c'}{17} \text{ MPa} \tag{6}$$
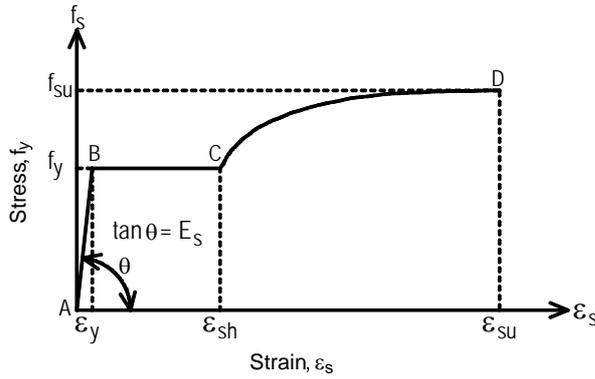


**Figure 5.** Stress-strain relationship for reinforcing steel proposed by Park and Paulay [19]

For reinforcing steel, the adopted stress-strain relationship in the proposed model is that proposed by Park and Paulay [19]. The related equations used to develop the constitutive laws in the model are as follows:

For region AB ($0 \leq \varepsilon_s \leq \varepsilon_y$),

$$f_s = \varepsilon_s E_s \tag{7}$$

$$\varepsilon_y = \frac{f_y}{E_s} \tag{8}$$

For region BC ($\varepsilon_y \leq \varepsilon_s \leq \varepsilon_{sh}$),

$$f_s = f_y \tag{9}$$

$$\varepsilon_{sh} = 16\varepsilon_y \tag{10}$$

For region CD ($\varepsilon_{sh} \leq \varepsilon_s \leq \varepsilon_{su}$),

$$f_s = f_y \left[ \frac{m(\varepsilon_s - \varepsilon_{sh}) + 2}{60(\varepsilon_s - \varepsilon_{sh}) + 2} + \frac{(\varepsilon_s - \varepsilon_{sh})(60 - m)}{2(30r + 1)^2} \right] \tag{11}$$

where:

$$m = \frac{(f_{su}/f_y)(30r + 1)^2 + 60r - 1}{15r^2} \tag{12}$$

$$r = \varepsilon_{su} - \varepsilon_{sh} \tag{13}$$

## Element Meshing

After preparing all the input data of material and geometrical properties, the column models were divided into small cubical elements. The meshing results of all column specimens used for model validation are shown in Fig. 6. Column specimen

LS0 was also meshed with similar pattern as three other column specimens shown in Fig. 6. For columns reinforced with steel rebar, it is worthwhile to notice that the meshing was created according to the locations of reinforcing bars, either the longitudinal or transverse reinforcement, as well as the column specimen cross-sectional perimeter. By using sharing nodes option in ANSYS [12], SOLID65 and LINK8 elements can be interconnected one to another forming a single solid column model which capable of simulating the actual behavior of reinforced concrete column.

## Loading Procedure

To apply the axial load on the top of the column specimen, an axial pressure was implemented over the entire top surface of the column model in the ANSYS software. The axial pressure can be simulated using the ANSYS load step option [12]. Load step option may be used when the incremental loading is considered. The number of load steps depends on the user's definition. In this case, load steps were defined according to the actual load steps applied during the test. A solution was obtained by solving several sub-steps in each load step to attain convergence. In each sub-step, an iteration procedure was carried out until providing a convergent solution before moving to the next sub-step. The number of the sub-steps taken in the analysis may improve the accuracy of the solution. It will, however, sometimes be very time-consuming when too many sub-steps are taken. To avoid the problem, ANSYS offers an alternate automatic time step option [12] to reduce the computational time required in the analysis. Due to this advantage, this option is selected. When the automatic time step option is selected, it will automatically resize the number of the sub-steps in each load step when it fails to reach a convergent solution. This process keeps repeating until it provides a convergence value.

When the load has reached its peak value, the load control mode was switched into the displacement control mode. The displacement control mode was set into several displacement steps corresponding to the experimental data. Using the automatic time steps, the column specimen was displaced until failure. The objective of using this kind of mode is to obtain the descending branch of the stress-strain curve of the column specimens under axial loading. The incremental nonlinear equation can be written as follows:

$$K(u)\Delta u = \Delta P \tag{14}$$

where $\Delta u$ and $\Delta P$ describe the unknown incremental displacement and the given incremental applied load vectors, respectively.

To solve a nonlinear problem, ANSYS uses the Newton-Raphson (N-R) method [12] involving an iterative procedure. This method starts with a trial assumption: $u = u_i$, to define the incremental of the next steps, $\Delta u_i = K^{-1}(u_i)\Delta P$, and the load vector exists beyond the equilibrium, $\Delta R_i = \Delta P - K(u_i)\Delta u_i$. There will always be a discrepancy between the applied load and the load evaluated based on the assumption. To satisfy the state of equilibrium, the load vector exists beyond the equilibrium should be zero. Since the solution requires an iterative procedure, a tolerance value should be determined such that a convergent solution can be obtained. In each iteration step, N-R method calculates the load vector exists beyond the equilibrium and always checks if the convergent solution under specified tolerance is obtained. If the value is still greater than the tolerance value, then the initial assumed value is updated with the incremental displacement, $u_{i+1} = u_i + \Delta u_i$. The next incremental solution vector is determined with $\Delta u_{i+1} = K^{-1}(u_{i+1})\Delta P$, providing a new load vector exists beyond the equilibrium $\Delta R_{i+1} = \Delta P - K(u_{i+1})\Delta u_{i+1}$. This procedure is repeated until the convergent solution is obtained.

## Numerical Impelementation

The quantitative implementation of the finite element procedure used in the ANSYS software [12] is based on the principles of virtual work or the postulation of minimum potential energy in the assembly of the elements as formulated the following equilibrium equation:

$$[K]\{d\}+\{F\}_p+\{F\}_g+\{F\}_{\varepsilon_0}+\{F_{\sigma_0}\}-\{R\}=0 \quad (15)$$

The stiffness matrix $[K]$,

$$[K]=\sum\int[B]^T[D][B]dv \quad (16)$$

The nodal force due to the surface load,

$$\{F\}_p=-\sum_{ele}\int[N]^T\{p\}dV \quad (17)$$

The nodal force due to the body load,

$$\{F\}_g=-\sum_{ele}\int[N]^T\{g\}dV \quad (18)$$

The nodal force due to the initial strain,

$$\{F\}_{\varepsilon_0}=-\sum_{ele}\int[B]^T[D]\{\varepsilon_0\}dV \quad (20)$$

The nodal force due to the initial stress,

$$\{F\}_{\sigma_0}=-\sum_{ele}\int[B]^T[D]\{\sigma_0\}dV \quad (21)$$
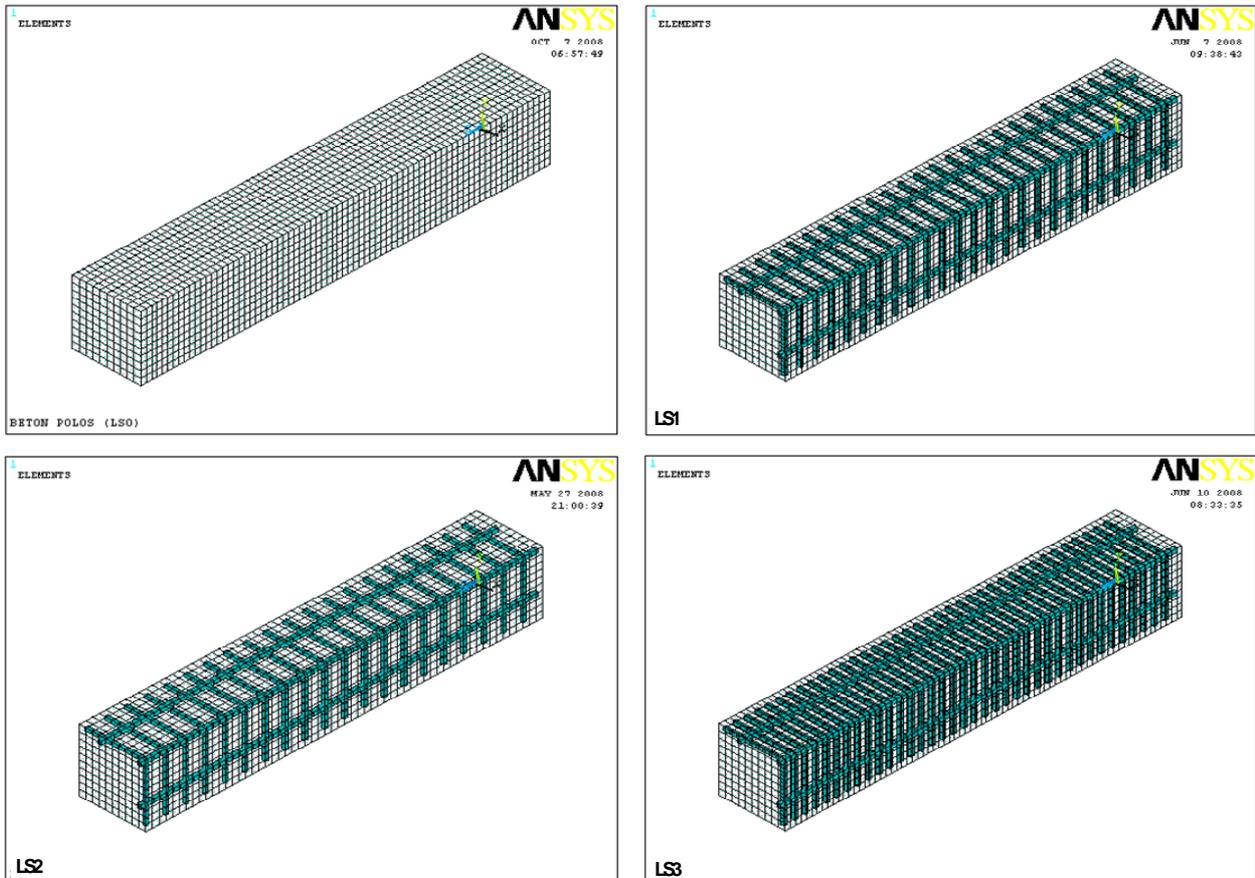


**Figure 6.** Element meshing of quarter column specimens LS0, LS1, LS2, and LS3

where [$N$] is the shape function; {$d$} is the vector of nodal displacement; {$R$} is the vector of applied nodal force; {$p$} is the vector of surface load; and {$g$} is the vector of body load. The ANSYS software uses Newton-Raphson (N-R) method [12] to obtain the convergent solution of the nonlinear equilibrium iterative equation to develop the stiffness matrix of the column model.

## Results and Discussions

### Stress Distribution

The axial stress distributions of column specimens LS0, LS1, LS2, and LS3 obtained from the ANSYS solution are shown in Fig. 7. As can be seen in the figure, for column specimens LS1, LS2, and LS3, the axial stress contours over mid-height cross sections of the column specimens indicate similar axial stress distributions with various intensities of stress concentrations. The axial stress concentrations around the longitudinal reinforcement also indicate similar axial stress distributions with the axial stress distribution in the actual column specimens. Column specimen LS0 has different axial stress contour since it does not contain any reinforcing bars

(plain concrete). Higher axial stress concentration occurs over the center region of the column cross section. This phenomenon describes a correct mechanism of a plain concrete column specimen subjected to axial loading.

### Stress-Strain Relationship

The axial stress-strain curves obtained from the ANSYS solution are confirmed by the experimental results [16]. From the comparisons shown in Fig. 8, it shows that the predictions are in close agreement with the experimental curves. This indicates that the actual behavior of column specimens confined by various volumetric ratios of transverse steel under axial compressive loading can be accurately predicted by the FEM approach.

The accuracy of the proposed procedure is also confirmed by the close values of peak stress, strain at the peak stress as well as strain when the stress drops to 85 percent of the peak stress obtained from the FEM analysis and the experimental test. From the comparison values listed in Table 3, it can be seen that the largest differences of all column specimens considered in the study are only 2.80,
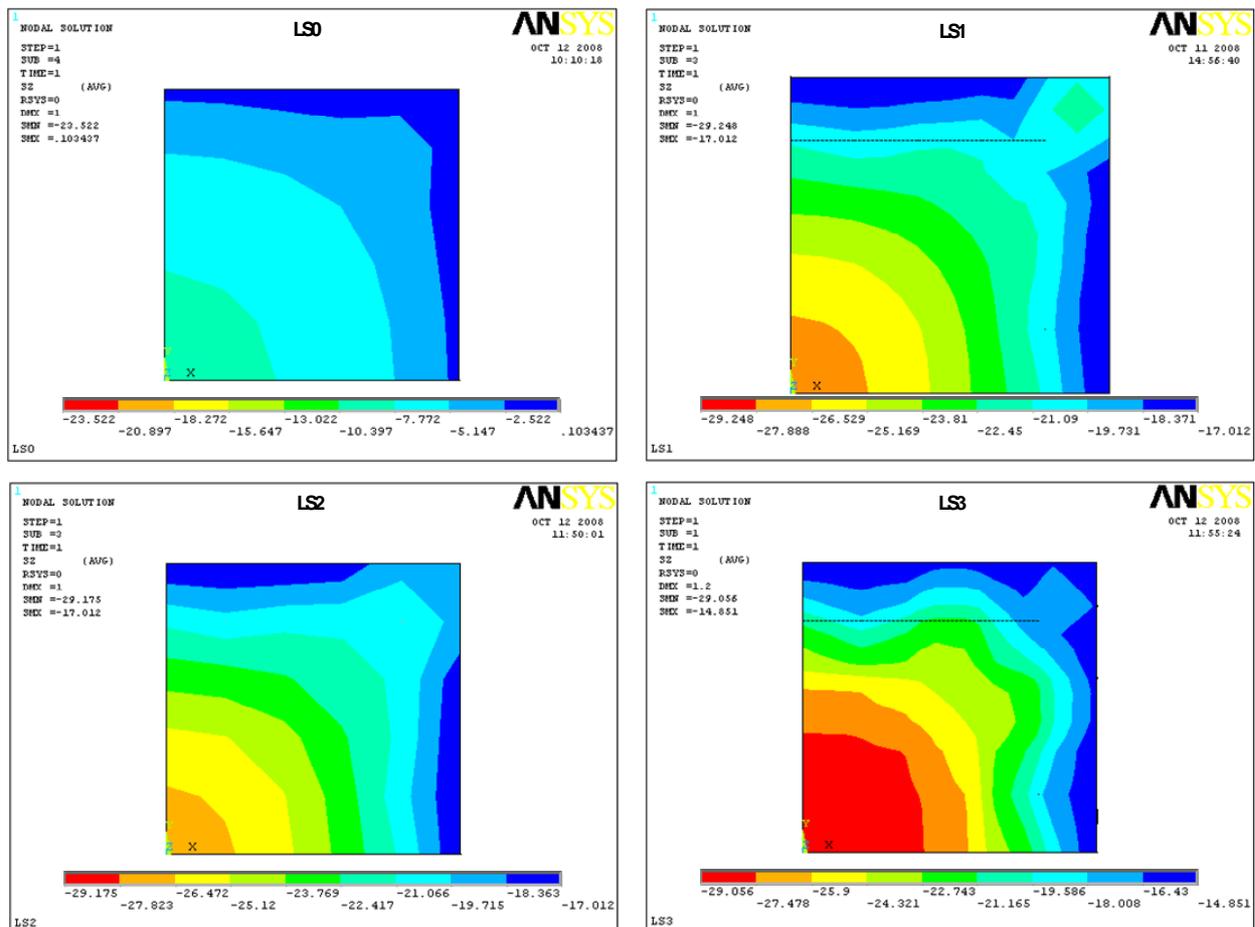


**Figure 7.** Axial stress distributions over the mid-height cross section of quarter column specimens LS0, LS1, LS2, and LS3
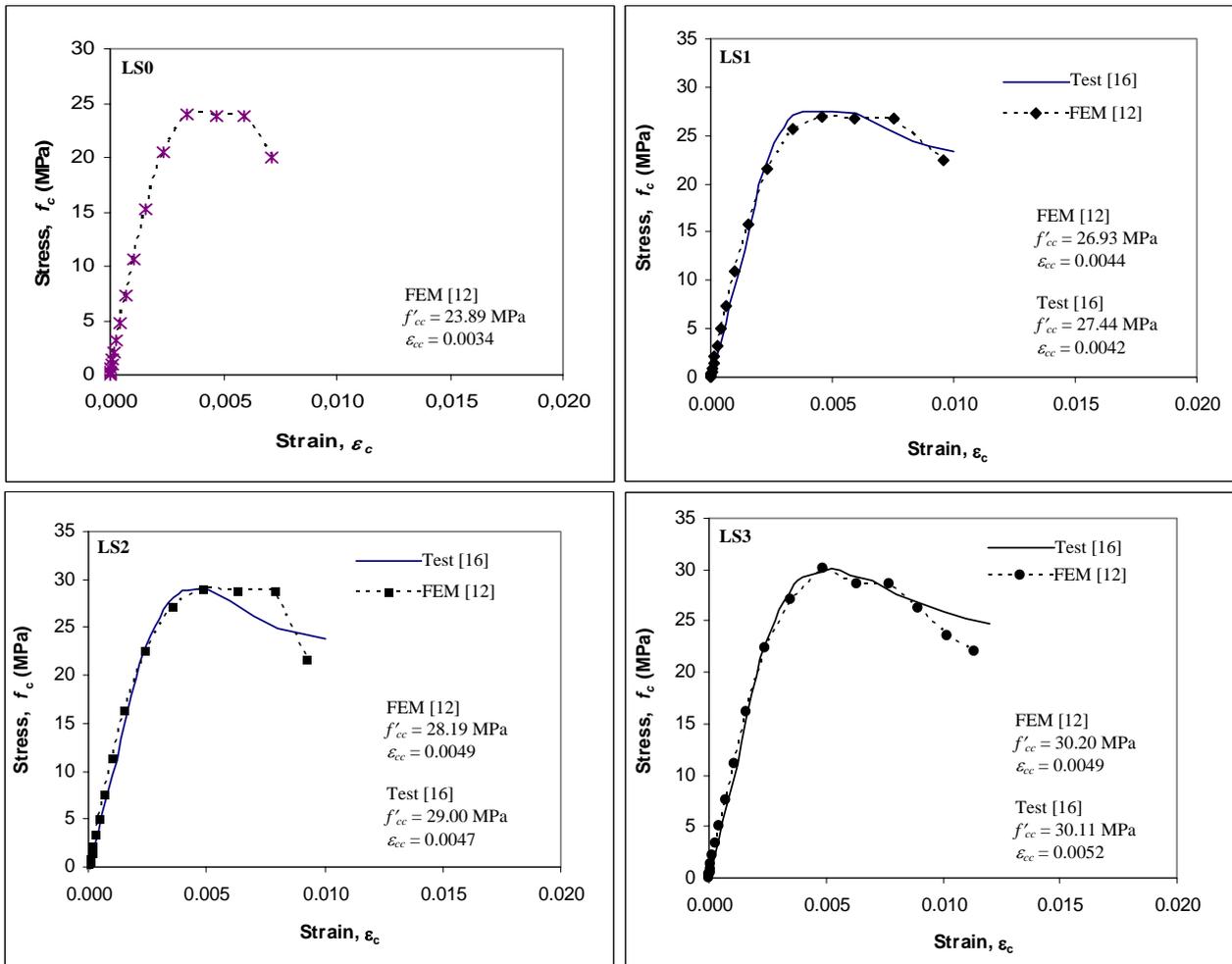
**Figure 8.** Stress-strain curves of column specimens LS0, LS1, LS2, and LS3

**Table 3.** Comparison between the peak stress and strain at the peak stress obtained from FEM analysis and experimental test

| Column Specimen ID | $f'_{cc}$ (MPa) | | | $\varepsilon_{cc}$ (%) | | | $\varepsilon_{cc85}$ (%) | | |
|---|---|---|---|---|---|---|---|---|---|
| | FEM | Test | Diff. (%) | FEM | Test | Diff. (%) | FEM | Test | Diff. (%) |
| LS0 | 23.89 | - | - | 0.34 | - | - | - | - | - |
| LS1 | 26.93 | 27.44 | 1.84 | 0.44 | 0.42 | 3.88 | 0.86 | 0.91 | 5.38 |
| LS2 | 28.19 | 29.00 | 2.80 | 0.49 | 0.47 | 2.58 | 0.84 | 0.85 | 1.17 |
| LS3 | 30.20 | 30.11 | 0.29 | 0.49 | 0.52 | 5.65 | 1.00 | 1.06 | 5.66 |

5.65, and 5.38 percents for the peak stress, the strain at the peak stress, and the strain when the stress drops to 85 percent of the peak stress, respectively. These values confirmed the accuracy of the proposed procedure in predicting the actual nonlinear behavior of the columns under axial loading shown in Fig. 8.

## Conclusions

Based on the FEM analysis and discussion above, the following conclusions can be drawn:

1. ANSYS software is capable of predicting the actual stress-strain relationships of both uncon-fined and confined reinforced concrete column specimens subjected to axial loading.

2. From the axial stress contours obtained from the FEM analysis, it can be concluded that the axial stress concentrations are in the center regions of the column cross sections, particularly in the confined areas.

3. The accuracy of the proposed procedure has been well confirmed by the close values of peak stress, strain at the peak stress as well as strain when the stress drops to 85 percent of the peak stress obtained from the FEM analysis and the experimental test.

## References

1. Cusson, D. and Paultre, P., High-Strength Concrete Columns Confined by Rectangular Ties, *Journal of Structural Engineering*, ASCE, V. 120, No. 3, Mar. 1994, pp. 783-804.

2. Saatcioglu, M. and Razvi, S., High-Strength Concrete Columns with Square Section under Concentric Compression, *Journal of Structural Engineering*, ASCE, V. 124, No. 12, Dec. 1998, pp. 1438-1447.

3. Assa, B., Nishiyama, M., and Watanabe, F., New Approach for Modeling Confined Concrete Rectangular Columns, *Journal of Structural Engineering*, ASCE, V. 127, No. 7, July 2001, pp. 751-757.

4. Fafitis, A. and Shah, P. S., Lateral Reinforcement for High-Strength Concrete Columns, *ACI Special Publication*, SP-87, Detroit, USA, 1985, pp. 213-232.

5. Nagashima, T., Sugano, S., Kimura, H., and Ichikawa, A., Monotonic Axial Compression Test on Ultra High Strength Concrete Tied Columns, *Proceedings of the 10th World Conference on Earthquake Engineering*, Madrid, Spain, July 19-24, 1992, V. 5, pp. 2983-2988.

6. Cusson, D. and Paultre, P., Stress-Strain Model for Confined High-Strength Concrete, *Journal of Structural Engineering*, ASCE, V. 121, No. 3, Mar. 1995, pp. 468-477.

7. Sheikh, S. A. and Uzumeri, S. M., Analytical Model for Concrete Confinement in Tied Columns, *Journal of the Structural Division*, ASCE, V. 108, ST12, Dec. 1982, pp. 2703-2722.

8. Sheikh, S. A. and Uzumeri, S. M., Strength and Ductility of Tied Concrete Columns, *Journal of the Structural Division*, ASCE, V. 106, ST5, May 1980, pp.1079-1102.

9. Scott, B. D., Park, R., and Priestley, M. J. N., Stress-Strain Behavior of Concrete Confined by Overlapping Hoops at Low and High Strain Rates, *ACI Structural Journal*, V. 79, No. 1, Jan.-Feb. 1982, pp. 13-27.

10. Nishiyama, M., Fukushima, I., Watanabe, F., and Muguruma, H., Axial Loading Test on High-Strength Concrete Prisms Confined by Ordinary and High-Strength Steel, *Proc. Symposium on High-Strength Concrete*, 1993, pp. 322-329.

11. Razvi, S. and Saatcioglu, M., *Test of High-Strength Concrete Columns under Concentric Loading, Rep. No. OCEERC 96-03*, Ottawa Carleton Earthquake Engineering Research Centre, Ottawa, ON, Canada, 1996.

12. ANSYS, *ANSYS User's Manual Revision 5.6, ANSYS, Inc.*, Canonsburg, Pennsylvania, 1999.

13. Santhakumar, R., Dhanaraj, R., and Chandrasekaran, E., Behaviour of Retrofitted Reinforced Concrete Beams under Combined Bending and Torsion: A Numerical Study, *Electronic Journal of Structural Engineering*, V. 7, 2007, pp. 1-7.

14. Wolanski, A. J., *Flexural Behavior of Reinforced and Prestressed Concrete Beams using Finite Element Analysis, MSc. Thesis*, Faculty of the Graduate School, Marquette University, Milwaukee, Wisconsin, May 2004.

15. Barbosa, A. F., and Ribeiro, G. O., Analysis of Reinforced Concrete Structures using ANSYS Nonlinear Concrete Model, *Computational Mechanics*, CIMNE, Barcelona, Spain, 1998, pp. 1-7.

16. Hoshikuma, J., Kawashima, K., Nagaya, K., and Taylor, A. W., Stress-Strain Model for Confined Reinforced Concrete in Bridge, *Journal of Structural Engineering*, ASCE, V. 123, No. 5, May 1997, pp. 624-633.

17. Popovics, S., A Numerical Approach to the Complete Stress-Strain Curve of Concrete, *Cement and Concrete Research*, V. 3, No. 5, May 1973, pp. 583-599.

18. Collins, M. P., Mitchell, D., and MacGregor, J. G., Structural Design Consideration for High-Strength Concrete, *Concrete International*, ACI, V. 15, No. 5, May 1993, pp. 27-34.

19. Park, R. and Paulay, T., *Reinforced Concrete Structures*, John Wiley and Sons, Inc., Canada, July 1975.