Computers and Concrete, *Vol. 11, No. 5 (2013) 475-492*
DOI: http://dx.doi.org/10.12989/cac.2013.11.5.475

Nonlinear finite element analysis of loading transferred from column to socket base

Özgür ANIL* and Burak UYAROĞLU

Civil Eng. Dept., Gazi University, Maltepe, Ankara, Türkiye, 06570

(Received April 23, 2012, Revised November 8, 2012, Accepted November 13, 2012)

Abstract. Since the beginning of the 90 s, depending on the growth of the industrial sector in Turkey, factory constructions have been increased. The cost of precast concrete buildings is lower than the steel ones for this reason the precast structural systems are used more. Precast concrete structural elements are mostly as strong as not to have damage in the earthquake but weakness of connections between elements causes unexpected damages of structure during earthquake. When looking at the previous researches, it can be seen that there is a lack of studies about socket type base connections although there were many experimental and analytical studies about the connections of precast structural elements. The aim of this study is to investigate the stress transfer mechanism between column and the socket base wall with finite element method. For the finite element analysis ANSYS software was used. A finite element model was created which is the simulation of experimental research executed by Canha *et al.* (2009) under vertical and horizontal forces. Results of experimental research and finite element analysis were compared to create a successful simulation of experimental program. After determining the acceptable parameters, models of socket bases were created. Model dimensions were chosen according to square section column sizes 400, 450, 500, 550 and 600 mm which were mostly used in industrial buildings. As a result of this study, stress distribution at center section of the socket base models were observed and it is found that stress distribution affects triangular at the half of socket bottom and top.

Keywords: nonlinear finite element analysis; socket base; contact analysis; ANSYS

1. Introduction

Since the beginning of the 90 s, our country has experienced a rapid growth in the industrial sector. Depending on the growth of the industrial sector, new fields of production have been needed. In parallel, constructions of factory building have been increased. While looking at their structural systems, it can be seen that steel and precast concrete systems are used in the construction of the industrial buildings. Long spans, short duration of construction, manufacturing processing that makes construction process not to be affected by weather conditions and constructing much floor heights can be listed as the main reasons for this.

Comparing with steel and reinforced concrete structural systems in terms of the cost of their frame structure, it can be seen that the construction of steel buildings is 96% expensive than those

*Corresponding author, Ph.D., E-mail: oanil@gazi.edu.tr

http://www.techno-press.org/?journal=cac&subpage=8 ISSN: 1598-8198 (Print), 1598-818X (Online)

of the concrete buildings. In this sense, the investors have tendency to primarily prefer to use precast concrete systems. Precast concrete structures are constructed in the field of construction area by combining building elements such as columns, beams, floors that are produced in the factory. Controlling during the fabrication process, standardization, curing practices, using high-strength concrete compared to conventional concrete systems and passing long spans by pre-stressing technique are important advantages of this system (Fabbrocino *et al.* 2005).

96% of surface area of Turkey is in the seismic zone and 42% of that is located within the boundaries of the 1st seismic zone. Although structural elements which are manufactured with using high-strength concrete and good reinforcement detail are competent to the earthquake-resistant, they can be damaged at the earthquake due to the weaknesses in the junctions of these elements. When investigating the previous studies about junction points of prefabricated structural elements, it is seen that although there have been many experimental and analytical studies regarding the types of connection at the point of beam-column joints, there have been very few studies about column and base joints (Alyavuz and Anıl 2007, Osanai *et al.* 1996, Canha *et al.* 2009, Canha 2004). Studies closely related to column-base joints are briefly presented below.

A study on the mechanism of stress transfer of column-base connection conducted by Osanai *et al.* (1996) is alternating to the load transfer mechanism contained in The German DIN 1045 and the Japanese AIJ Recommendations and it analyses the forces of friction between the socket and the column face within the theoretical formulation. Then, an experimental study was carried out in order to control the formulation. In this experimental study, shear key was utilized for better formation of the friction between the column face and interfaces of the socket and the results were compared. It was seen that there was similarities between the results obtained from the theoretical formulation and those of the experimental study. General conclusions from this study were stated as there is always a good association when embedded length of column is equal to or greater than 1,5 times of the largest dimension of the column; and the column was cracked by shear force while reaching the flexural capacity whether there was a shear key; in other words, the connection did not cause a problem related to the stiffness. They stated that this value could be reduced to 1,25, and if this value was reduced, they suggested to use shear key not to cause a problem.

In a study of Canha *et al.* (2004, 2009) the load transfer mechanism of column socket base interface was explained. For this purpose, an experimental study was conducted, and two different load transfer mechanisms that are compatible with the experimental results were proposed for rough and smooth interfaced socket bases. 7 samples were used in this study, only the first 4 of them were smooth interfaced samples, the remaining 3 types were rough interfaced. Samples were 1/1 scale and the size of the column was selected as 400 x 400 mm dimension which is commonly used in practice. While the size of the 1st, 2nd and 3rd numbered smooth interfaced samples' socket depth were selected as 2 x 400 mm dimension, the size of the 4th one was chosen as 1.6 x 400 mm. While the size of the 1st and 2nd numbered rough interfaced samples' socket depth were selected as 1,6 x 400 mm dimension, the size of the 3rd one was chosen as 1,2 x 400 mm. Experiments were done by applying the normal force and bending moment at the upper ends of column, failure cracks in the walls of the socket, the vertical reinforcement behavior were investigated. After comparing theoretical results with experimental data, a design model that overlapped the results of experimental study was proposed. As a result of the experiments, it was explored that the side walls of the 2nd and 3rd numbered smooth interfaced samples displayed short console behavior whereas the 4th one whose depth was reduced did not show the same result. The 1st and 2nd types of rough interfaced samples represented monolithic behavior and the strength of the 3rd one was decreased at the rate of 23%, comparing other samples. It was also dk explored that the design model proposed for smooth interfaced samples overlapped the results of the 2nd and 3rd numbered samples.

Although there have been different types of practices about column-base connections for prefabricated concrete structures, in our country, socket base is commonly used. In this system, in the field of construction, there are sockets settled on the footing manufactured by cast in place and connected with the footing monolithically. Sockets are open-top, box-shaped concrete structures enclosed by shear walls in 4 sides. Prefabricated columns that are produced in the factory environment and then send to the construction area are placed in the socket and fixed by the help of wooden wedges. Arranging orthogonality and axis misalignment of columns, fixing of the column is provided by pouring of concrete in a void between the socket and column. There is no reinforcement in the void between the socket and the column (Alyavuz and Anıl 2007).

Having occurred, Adana - Ceyhan earthquake (1998) and Kocaeli earthquake (1999) in Turkey, one of the most observed problem regarding precast concrete structures is that beam-column joint regions were damaged as a result of the displacement of the upper end of the columns. These types of columns in buildings are usually fixed at the base and pinned at the upper end. For the prefabricated concrete structures whose floor heights are more than other structures, movements at the point of column-base connection may lead to the large displacements at the upper end of the columns. Therefore, how horizontal seismic forces are transferred from columns to the foundation at the base-column connection and distribution and geometry of stress transmitted to the foundation should be determined correctly (Alyavuz and Anıl 2007, Osanai *et al.* 1996, Canha *et al.* 2009, Canha 2004).

Exploring a limited number of experimental and analytical studies related to socket bases, it can be pointed out that there have been different proposals related to transfer mechanism of the stress transferred from the column to the socket wall. Accordingly, within the scope of this study, it was decided to examine the transfer mechanism of the stress between column and socket wall by means of the establishment of a computer model by using the finite element method. For the achievement of this study, firstly control model was created by using the data of an experimental study already carried out. After comparing the data of experimental study with the results obtained from finite element analysis of this model, finite element model was formed as close to reality. This study has an important role in terms of investigating the reliable and accurate results obtained from the analyses of finite element models whose experiment does not exist.

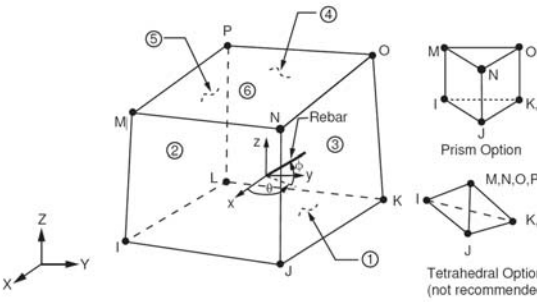
Within the scope of this study, firstly, the finite element model of the experimental study of Canha *et al.* (2004) which explored behavior of the socket base-column connection on the horizontal and vertical loads was created. A behavior between the column and socket wall is composed by load transfer as a result of contact with two different qualities of concrete mass. This is required to solve a problem of a non-linear surface-surface contact. For this purpose, ANSYS which is commonly used non-linear finite element analysis software in the academic fields and contains various types of elements including concrete and surface contact elements is utilized in this study (ANSYS 2009). After the necessary definitions such as contact surface, concrete, reinforcement, and installation features were given, non-linear finite element analysis was conducted and the results obtained from these analyses were compared with experimental results (Saboori and Khalili 2012, Han *et al.* 2011, Drosopoulos *et al.* 2012, Shi *et al.* 2012, Bulut *et al.* 2011, Dogan and Anil 2010). After identifying the appropriate analysis parameters, the analyses of the models generated on the basis of the basic dimensions of the column and base commonly used in practice were conducted. While evaluating the results of the analysis of the models, stress transfer mechanism of the prefabricated reinforced concrete column and socket base connection was interpreted.

2. Finite element model

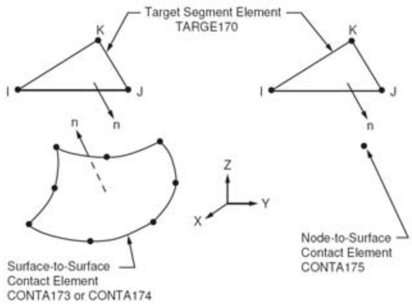
In this study, ANSYS as a finite element software which has been used in many fields of research such as structural mechanics, physics, fluid dynamics, dynamic, and electromagnetic since 1970 is utilized. ANSYS software is frequently used in academic studies due to many reasons. Some of the reasons for the selection of the ANSYS software are as follows: it contains many types of finite element including the behavior of concrete and reinforced concrete; it allows to use non-linear material models; it allows to solve a problem of 3-dimensional surface-surface contact; and results of the analysis are validated with experimental data in many academic studies.

Finite element models of the socket bases are prepared by using elements of Solid65, Targe170, Conta174 element library of the ANSYS. The element of Solid65 is used for identification of behavioral characteristics of reinforced and unreinforced concrete at the parts of the socket base. Conta174 is used as an element of surface contact. Targe170 is used for the model of the contact surface between filling concrete inside the socket and column. Properties used for the types of finite element are given in Fig. 1.

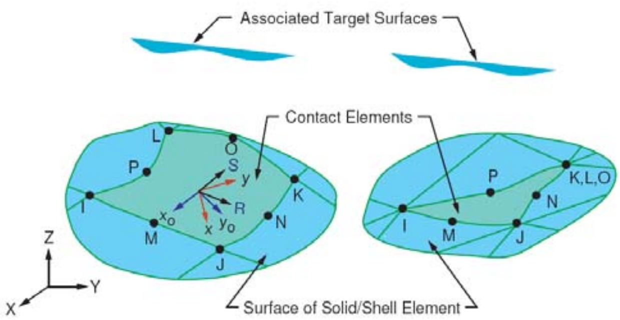
Solid65 element is a 3-dimensional rigid body element which allows the modeling of reinforced or unreinforced concrete. It has cracking and crushing properties under tensile and compressive stresses, respectively. Also, it allows the definition of reinforcement at different rates



(a) Solid65 finite element



(b) Targe170 finite element



(c) Conta174 finite element

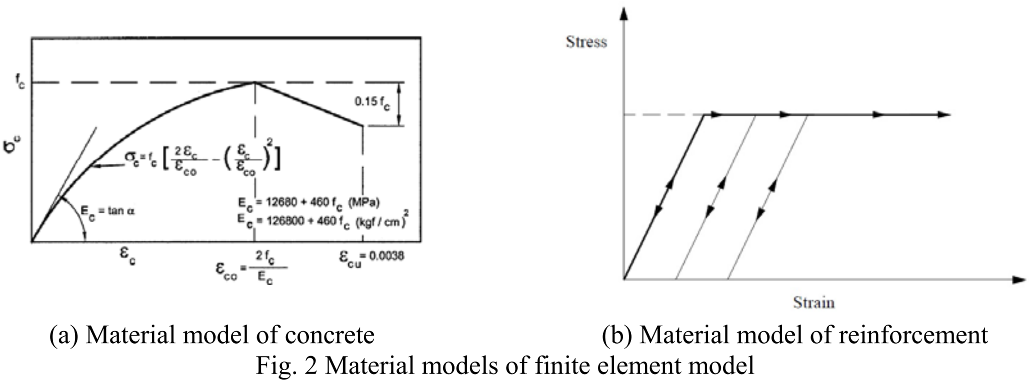
Fig. 1 Finite element types using into model in X, Y and Z directions by using real constant settings in an element. The amount of reinforcement is expressed by the volumetric ratio. Defined reinforcement can be employed with tensile and compressive stress but does not take shear force. This element has 8 nodes and there are 3 degrees of freedom in each of X, Y and Z direction. The most important feature of this element, characteristics of non-linear element are superior to the elements having similar characteristics in the library of the ANSYS. The element of Solid65 offers the possibility to control the behavior of the system for the identification of the parameters of concrete's collapse failure. Software uses William - Warnke failure criterion in calculations.

The element of Targe170 is a target surface element used in the 3-dimensional contact analysis. Contact elements determine the boundaries of the body that may be deformed, while wrapping 3-dimensional elements such as Solid65. This target surface creates a contact pair with Targe170 target surface elements and Conta174 contact surface elements and is governed with a real constant set in which features are controlled together. Forces and moments as well as planar or rotational displacements, the difference in temperature, electric and magnetic forces can be applied to Targe170 element. Complex elements of the target surfaces can be easily created with Targe170 element thanks to its 1, 2, 3, 4, 6 and 8 node point and structure of various geometric shapes. Created substrates may be rigid or flexible.

Element of Conta174, which is an appropriate element for analysis of 3-dimensional and structural surface-to-surface contact, is used to describe the contact and slip between three dimensional surfaces. For this purpose, a surface that may be deformed are created with element Conta174 and placed at the surfaces of 3-dimensional elements with the help of the nodes of the midpoint. An element having 8 nodes has the same geometric features with the surfaces of 3-dimensional elements to which it is connected. When a contact element penetrates into the target surface, a contact is provided. Coulomb and shear friction between the pair of contact can be defined with the help of Conta174.

2.1 Material model using finite element model

In this part of the study, selected models of materials for reinforcement and concrete used in the finite element model are briefly described. In this study, a model of Hognestad with which the σ–ε relationship of unconfined concrete is defined is utilized. $\sigma - \varepsilon$ curve of the material model that is proposed by Hognestad and widely used is given in Fig. 2(a). Reinforcement steel under tensile



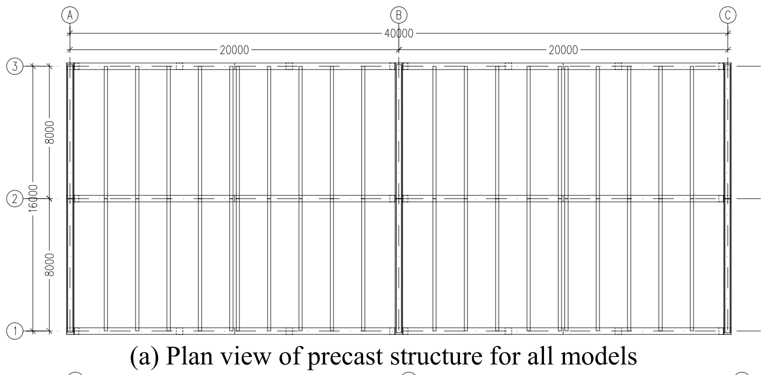
(a) Material model of concrete (b) Material model of reinforcement

Fig. 2 Material models of finite element model and compressive stress strain curves is assumed to be the same. A graph of $\sigma - \varepsilon$ which is obtained under axial tensile or compressive is displayed in Fig. 2(b). This model is generally accepted and widely used model. This model was used in the finite element analysis carried out in this study. As shown in the model, hardening was not taken into account.

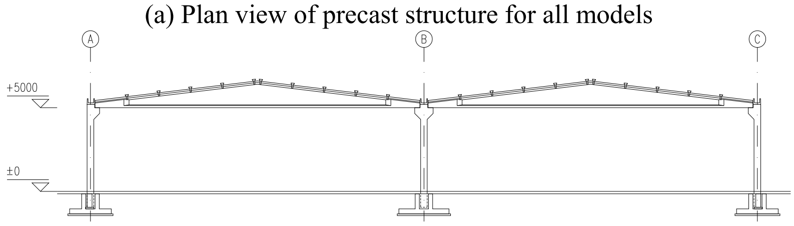
2.2 Loading of finite element model

In the finite element analyses related to the determination of transfer mechanism of stress transferred from the column to the wall of the socket base, the distribution of stress on the socket base is investigated through the implementation of moment and axial load to the upper end of column. In determining vertical and horizontal loads, provisions of TS498 standard (TSE 1997) and the Principles of Construction in Seismic Zones (DBYYHY 2007) as the related regulations of Turkey were taken into account.

The Snow load, roof covering load, and dead load of prefabricated reinforced concrete structural elements were considered in the calculation of the vertical loads acting on the column. Besides, the calculation of horizontal loads was carried out by a method of equivalent seismic load according to the DBYYHY 2007. According to this, axis measurements of the structure in accordance with the length of a trapezoidal roof girder, gutter girder and purlin which are often used in practice were determined. A sample selected from plan and section views used for the calculation of the loadings of the models analyzed in this study is presented in Fig. 3. Reinforcement ratio in the range of $0.01<\rho<0.015$ for each type of column cross-section whose finite element model will be created within the context of the study was prescribed and according to this ratio, normal force-moment curve was drawn. Seismic analysis of the structure was conducted according to the method of equivalent seismic loads by the selection of any height of the column. In addition to that, whether a capacity of column-moment is exceeded or not was



(a) Plan view of precast structure for all models



(b) Section view of precast structure for Model-1

Fig. 3 Plan and section view example of analyzed structures controlled by marking an acquired value of moment in the moment-axial load diagram. The status of achieved moment value which creates a value of limit capacity for the calculated value of axial load was determined as the height of the structure. This allows determining which type of structure the stress transfer mechanism that will be obtained as a result of the finite element analyses has in the applications. The total mass of the structure was initially calculated to investigate equivalent seismic load acting on the selected structure. In terms of the formation of the most unfavorable loading conditions, the 1st seismic zone and the *Z*4 ground class was adopted in the conducted analyses, and accordingly A_o = 0,40 and T_a = 0,20, T_b = 0,90 were taken due to the related regulations. The structural system behavior factor for single-storey buildings in which seismic loads are fully carried by the columns with hinged connections on the top. Building importance factor for the industrial buildings are defined as I = 1,00 (DBYYHY 2007).

The provisions of TS9967 standard (TSE TE9967 2004) used for prefabricated buildings in Turkey were applied in determining the size and reinforcement of the models in the conducted finite element analyses for the aim of the exploration of the stress transfer mechanism transferred from the column to the wall of the socket base. The sizes of the models were assigned on the basis of the size of the columns due to the designation of the size of socket depending on the dimensions of the column according to this standard.

Investigating one storey, hinged at the upper end of the columns, shed-type precast concrete industrial structures, square-section columns were commonly used in these structures. The size of the columns is determined depending on the height of the structure, roof load, and earthquake force. In this sense, the dimensions of the column are selected as 400, 450, 500, 550 and 600 mm in practice. The sizes larger than 600 mm are not preferred except in special circumstances because of the difficulty in the columns' transport. In the context of the study, 5 different analyses were carried out through the creation of the finite element models for each of 5 square-sectioned columns whose edge lengths are 400, 450, 500, 550 and 600 mm dimensions.

The finite element program SAP2000 was used for the finite element analysis of the frame structure within the calculations of the design loads and defined finite element model is presented in Fig. 4. Framed formed columns are designated as fixed at the base and hinged at the connection points of a trapezoidal roof girder. Concrete class and the elastic modulus is taken as *C*30 and *E* = 32000 MPa, respectively.

Precast structure designed by using 400 x 400 mm square-sectioned column is named as "Model 1". A plan and section of the precast concrete industrial structure presented in Figure 3 was used in the calculation of horizontal and vertical loads transferred from the structure for the design of socket. Construction plans of the buildings remained constant and only storey height of the structure was increased for other structures with 450, 500, 550 and 600 mm square-sectioned columns and this cause a variation.

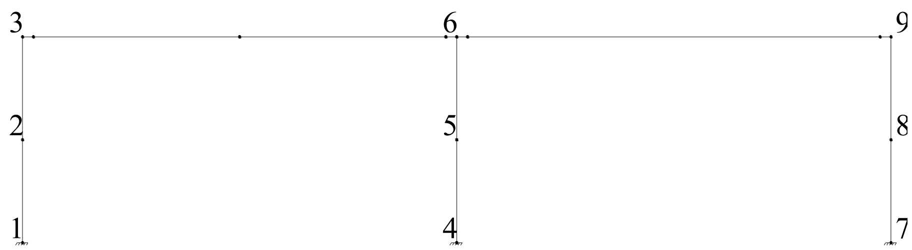


Fig. 4 SAP 2000 finite element model for calculating internal forces

Table 1 Model 1 SAP2000 analysis results

Node number	W_i (N)	H_i (mm)	$W_i x H_i$	F_{fi} (N)	m_i (kgf)	d_{fi} (mm)	$F_{fi} x d_{fi}$	$m_i x d_{fi}^2$
3	114570	5000	572850000	16230	11678,899	13,1	212613	2004215,9
6	213070	5000	1065350000	30180	21719,674	13,1	395358	3727313,2
9	114570	5000	572850000	16230	11678,899	13	210990	1973733,9
2	19600	2500	49000000	1390	1997,9613	3,8	5282	28850,561
5	19600	2500	49000000	1390	1997,9613	4,1	5699	33585,729
8	19600	2500	49000000	1390	1997,9613	4,5	6255	40458,716
			2358050000				836197	7808158

Table 2 Loading using finite element analysis colum-socket base interaction

Model No	N (N)	V (N)	M (N-mm)
1	308830	53270	156750000
2	316720	55850	216380000
3	327310	59370	282910000
4	340960	63930	380940000
5	358030	69640	445190000

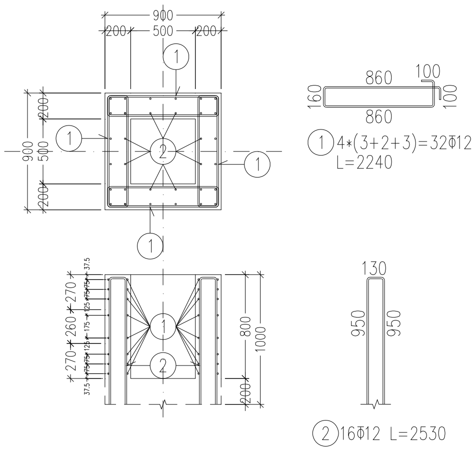


Fig. 5 Model -1 Reinforcement details

Joint displacements were calculated and the modal period was obtained as T=0,60 sec through the analysis of the finite element model of the structure with 400 x 400 mm square-sectioned column shown in Fig. 4 with the help of SAP 2000 software. The results of the analysis of Model 1 are given in Table 1. The same analyses were made for other building and loads acting on the columns were calculated for five models. Joint reactions obtained as a result of analysis carried out by means of the finite element program SAP 2000 are presented in Table 2 to determine the loads used for the analysis of stress transfer acting on column-socket joints.

The provisions of the regulation TS9967 was applied in determining the size of the socket and calculation of the reinforcement (TSE TS9967 2004). Concrete class is obtained as C20 and f_{ck}=20 MPa in the design of reinforcement, the reinforcement is taken into account as the S420 and f_{yk}

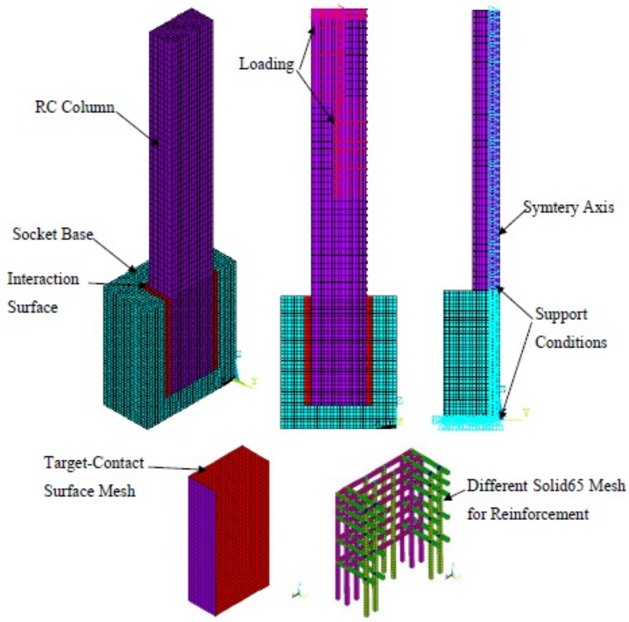


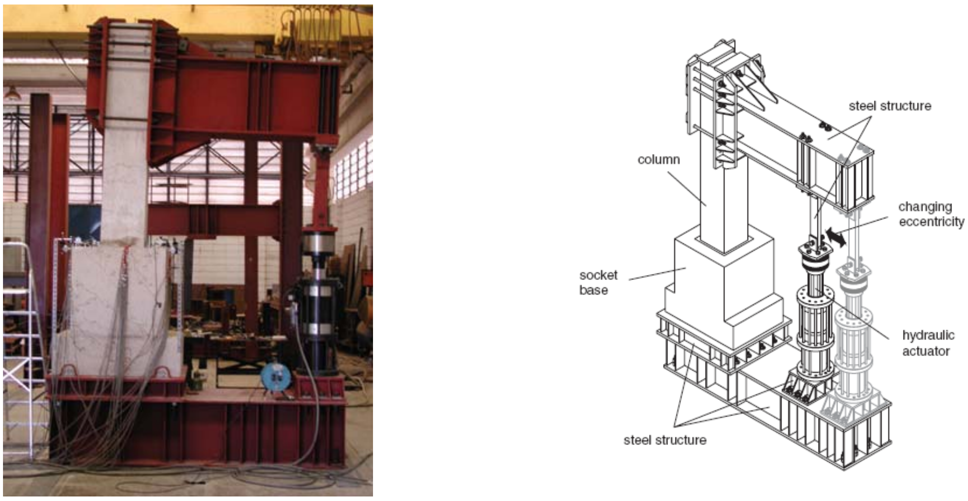
Fig. 6 Finite element model column-socket base connection

Table 3 Finite element number, iteration number and analysis time

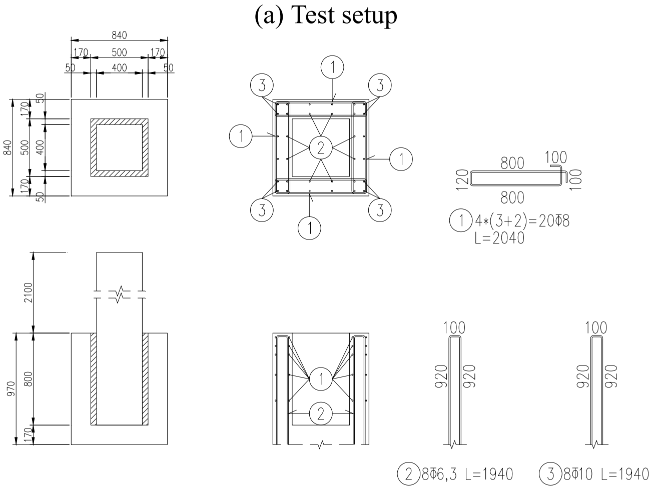
Model No	Finite element number	Iteration number	Analysis time (Hour : Minute)
Canha [4]	30797	882	23:49
1	34609	217	35:50
2	41545	295	39:20
3	51719	265	44:44
4	73860	293	49:43
5	83927	267	51:22

=420 MPa in calculations. Details of the reinforcement of the socket for the Model 1 with 400 x 400 mm squared column are given as an example in Fig. 5. After analyses and calculations outlined above were also made for each of the five models, all the necessary data was determined for modeling the column-socket connection. A half of the column-socket joint was modeled thanks to its symmetry for the aim of making a time profit in designing finite element models on the computer. Created finite element mesh is generated in Fig. 6.

The size and placement of reinforcement for the models, material properties except implemented force, contact surface parameters, and other parameters related to the finite element analyses are the same for the five models. The force was applied in a total of 10 steps to the models and in each step, it was applied by increasing up to 1/10 of the total force. The number of iterations made for convergence in the non-linear analyses is automatically determined by the software. The total numbers of iterations for each model, the total number of elements that make up finite element mesh and analysis time are shown in Table 3.



(a) Test setup



(b) Reinforcement details of specimens

Fig. 7 Properties of experimental study done by Canha *et al.* (2009)

2.3 Verification study of finite element model

In a study carried out by Canha *et al.* (2004, 2009) in the laboratory of Sao Carlos Faculty of Engineering in the University of Sao Paulo, precast reinforced column-socket joint was investigated experimentally and theoretically and the behavior of the socket walls are also discussed.

Within the scope of the experimental program, seven 1/1 scaled samples were tested under the influence of axial force applied with a large eccentricity. Experimental setup used in the experimental study and the details of reinforcement are given in Fig. 7.

In the experiment made by Canha *et al.* (2004, 2009) the displacement were measured at different points of the specimen's outside and stress values acting on the reinforcements were also measured. As the aim of this study is to investigate the behavior of the socket, measurements made only from were taken as the reference. Placement of displacement measurements made from the

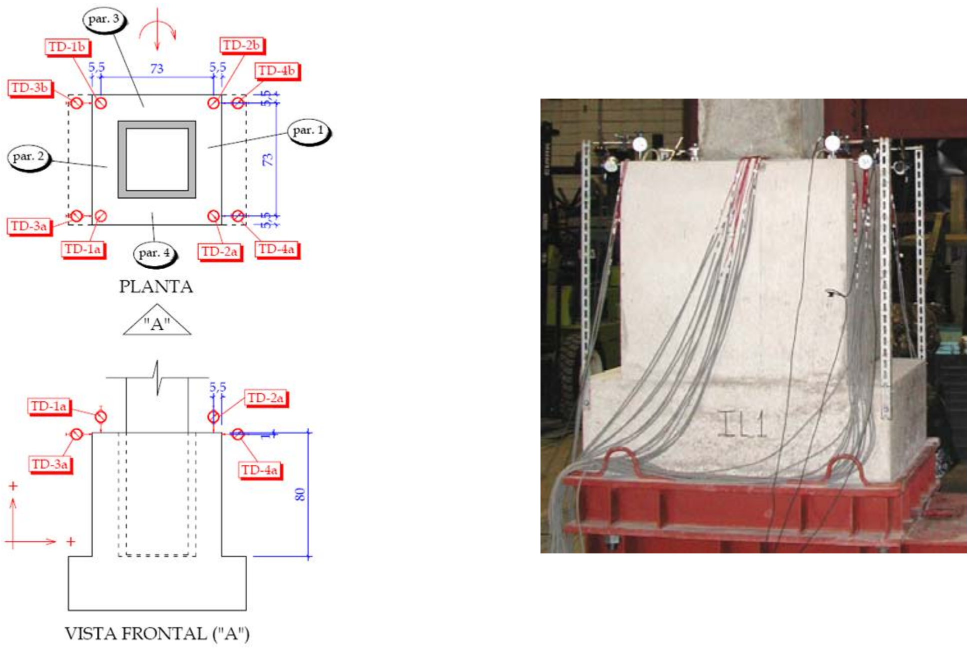


Fig. 8 Instrumentation setup of experimental study (Canha *et al.* 2009)

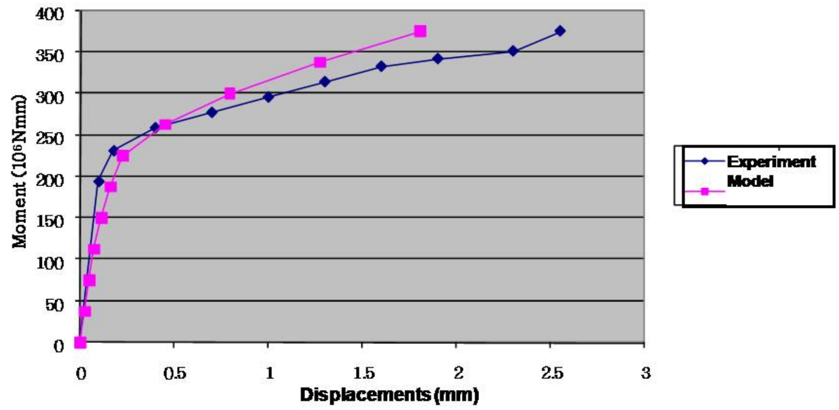
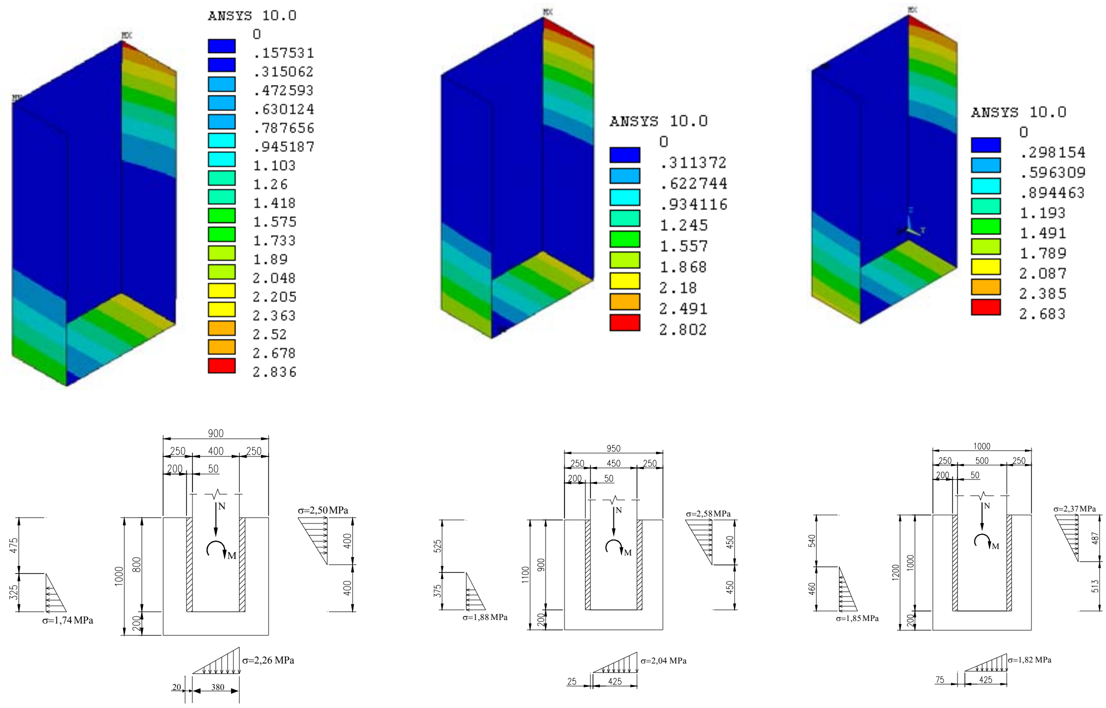
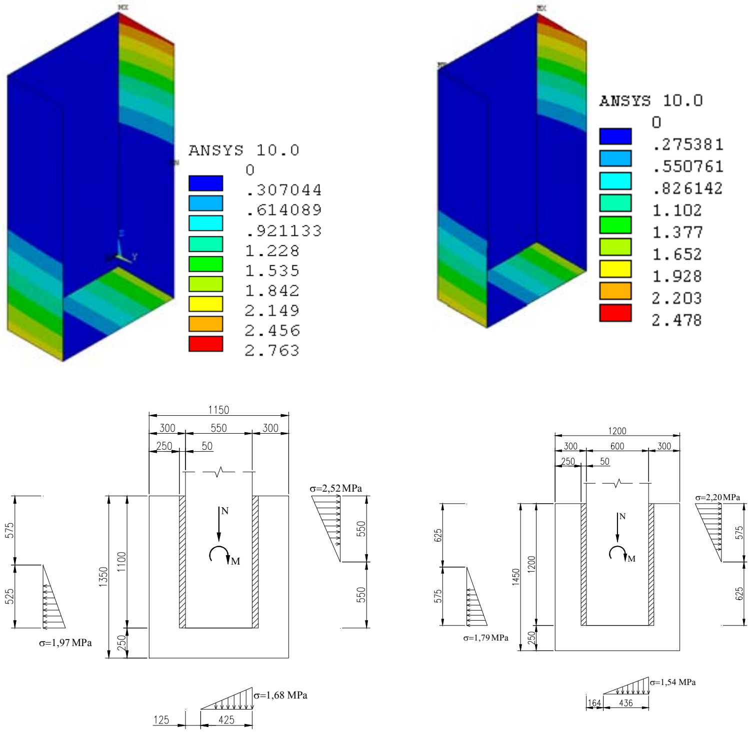


Fig. 9 Comparison of experimental and finite element analysis load-displacement curves

outside of the socket's side walls are displayed in Fig. 8. For the comparison of the results of the finite element analysis with the experimental results, values obtained from measuring points TD-4a and TD-4b in which displacement values of the socket's corner point was measured towards the installation were taken into account as a reference. A graph of the load-displacement obtained from Finite element analysis and a graph obtained from results of the experiment are shown in Fig. 9. It is seen that these load-displacement graphs are quite compatible with each other while exploring the results of both experimental and finite element analysis. The yield point and the displacement were obtained very close to each other. In addition, collapse load and



(a) Model-1 (b) Model -2 (c) Model -3



(d) Model -4 (e) Model -5

Fig. 10 Column-socket base interface stress distribution displacement were obtained very close to the analysis' results. As a result, nonlinear behavior in the finite element analysis has been achieved successfully.

3. Column-base interface stress analysis

The finite element model verified by using experimental results of the study of Canha *et al.* (2009) was used to analyze the stress transfer mechanism formed at the widely used 5 different column-socket joint. The models of materials and contact parameters in the confirmed model were also used within the analysis of the stress transfer mechanism and only the geometric dimensions and reinforcement details indicated a change. Contact stress distributions at the column-socket joint obtained from the finite element analysis and 2-dimensional drawings of these values for the axis of symmetry of the socket are given in Fig. 10.

The largest values of the stress and the distances of the stress' spread which were acquired for the middle section of the socket with the help of the finite element analyses are given in Table 4. After stress distributions obtained from the analysis of all models are examined, the average values of the stress distributions obtained for the middle section of the socket were measured as 0,49 l_b for the upper end of the socket, 0,44 l_b for the lower end of the socket, and 0,86 b for the socket base. According to the results of the non-linear finite element analysis, the horizontal stress distribution transferred from the column to the socket walls is effective about half of the socket's height in

Table 4 Column- socket base interface stress values

Stress values at upper side of socket wall				
Model No	l_b (mm)	Stress height (mm)	Stress_{top} (MPa)	l_b/Stress Height
1	800	400	2,5	0,50
2	900	450	2,58	0,50
3	1000	487	2,37	0,49
4	1100	550	2,52	0,50
5	1200	575	2,2	0,48
Stress values at lower side of socket wall				
Model No	l_b (mm)	Stress height (mm)	Stress_{bot} (MPa)	l_b/Stress height
1	800	325	1,74	0,41
2	900	375	1,88	0,42
3	1000	460	1,85	0,46
4	1100	525	1,97	0,48
5	1200	550	1,79	0,46
Stress values at base of socket				
Model No	b (mm)	Stress length (mm)	Stress_{base}(MPa)	b/Stress length
1	400	400	2,26	1,00
2	450	425	2,04	0,94
3	500	425	1,82	0,85
4	550	425	1,68	0,77
5	600	436	1,54	0,73

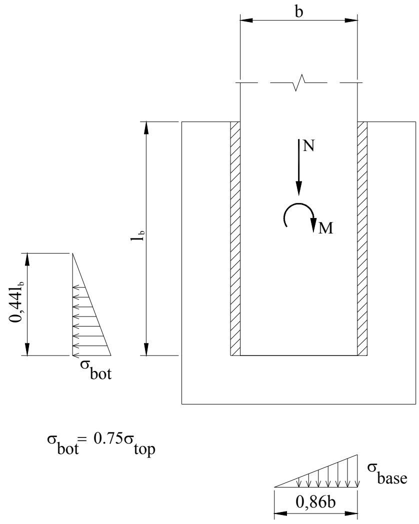


Fig. 11 Proposed stress distribution model colum-socket base interface according to finite element analysis

a triangle spread shaped. It can be stated that this stress can be applied in an equal value and height for both the upper and lower end of the column. A generalized form of the stress distribution is given in Fig. 11. The usage of this distribution in the analyses is thought to be beneficial for the calculation of element force values which are more realistic and closer to the experimental results.

Investigating the literature on the design of the socket bases, it can be noted that a force equation has often been proposed for the design. After the stress values obtained from the conducted finite element analyses was converted into a singular force, they were compared with formulas of the force proposed in the previous studies and regulations. Reinforced concrete design was usually carried out for the design of the upper end of the socket and the reinforcement obtained from it was used as the same for the lower end of the socket. Therefore, only for the upper end of the socket, it is thought to be appropriate for the comparison of the forces.

The force proposed for the design of the socket base in TS9967 standard and acting on the upper end of the socket is expressed by Eq. (1). This expression is also mentioned in German DIN1045 standard (TSE TS9967 2004, DIN 1045 1988). In the regulation, while the force is suggested to be implemented from the upper end of the socket to the below in the rate of 1/6 of the socket height, the stress distribution are displayed as the uniform format in 1/3 area of the height of the socket.

$$V_o = \frac{3}{2}\frac{M}{l_b} + \frac{5}{4}V \tag{1}$$

The force proposed for the design of socket base acting on its upper end in Japanese standard AIJ is presented in Eq. (2). In the Regulation, it is focused on that the force is applied from the upper end of the socket to the below in an amount of d_t and the lower end of the socket is proposed to be uniformly distributed manner in a region with d_c height (AIJ 1999). A value of d_t was determined as 1/6 of the height of the socket, d_c height was implemented as 1/3 of the height of the socket in the calculations.

Table 5 Comparison of regulation equation with ANSYS analysis results

Model No	TS9967	AIJ	Canha	ANSYS
1	360493,8	320541,3	202489,7	200000,0
2	430445,8	388558,3	244559,2	261225,0
3	498577,5	454050,0	285417,8	288547,5
4	599376,1	551428,6	345951,0	381150,0
5	643537,5	591307,5	372019,5	379500,0

$$V_o = \frac{1}{l_b - d_t - \frac{d_c}{2}} \cdot \left[M + \frac{V}{2} \cdot (l_b - d_c) \right] \tag{2}$$

The force obtained as a result of experimental studies carried out by Canha and colleagues (Canha *et al.* 2009) and acting on the upper end of the socket is shown in Eq. (3). Canha *et al.* (2009) suggest that the force is applied from the upper end of the socket to the below y, from the lower end of the socket to the above *y*', and misalignment of the reaction force formed at the column base from column axis is given as e_{nb}. In the calculations, the values of y and y' and the value of e_{nb} were obtained up to 1/6 of the height of the socket and 1/4 of the column size, respectively. Coefficient of friction was taken as $\mu = 0.6$.

$$V_o = \frac{M - N \cdot \left[e_{nb} + \frac{\mu y' - \mu^2 (0,5b + e_{nb})}{1 + \mu^2} \right] + V \cdot \left[l_b + \frac{y' - \mu (0,5b + e_{nb})}{1 + \mu^2} \right]}{l_b - y - y' + \mu b} \tag{3}$$

Comparing with the V_o forces acquired by using equations outlined above, it is seen that the forces obtained by the finite element analyses are very close to the values made by using the formulation proposed by Canha *et al.* (2009). The values calculated according to TS9967 stay on the safe side and the ones calculated according to AIJ are closer to the values created by using the experimental and analytical values. Calculated V_o forces are displayed in Table 5.

Stress distribution obtained by finite element analyses was affective on the half of height of the socket with a triangle shape. Forces obtained at the upper end of the socket were accepted to be used for the design of its lower end. In this sense, the points which the forces are acted on are equal to each other and at a distance from the upper end of the socket to 1/6 of the socket height. By making these assumptions, a simplified V_o force equation which can give the results of finite element analysis created to be used by the designers and displayed in Eq. (4).

$$V_o = \frac{M}{l_b} + \frac{5V}{4} - \frac{N}{5} \tag{4}$$

In Eq. (4), a singular design force V_o is acted on the socket wall, design moment is transferred from M column to the socket base, shear force is transferred from V column to the socket base, normal force is transferred from N column to the socket base, and l_b is defined as the length of the remaining part of the column inside the socket. By the usage of the obtained equation, more realistic internal forces compatible with the results of experiment and 3-dimensional non-linear finite element analysis can be calculated.

4. Conclusions

Within the context of the study, the study based on the load transfer mechanism acting on the connection region of the flat surfaced socket base which is used in the manufacture of prefabricated concrete buildings and column was carried out. This load transfer mechanism which is widely used in prefabricated reinforced concrete buildings is an extremely complex problem of engineering including a three-dimensional contact problem. As there have been very few studies concerning this issue, it has been decided to be realized. Mentioned joint detail widely used in prefabricated reinforced concrete and industrial buildings is an important subject to be investigated because these structures have important effects on the nation's economy. The large height of the floor for prefabricated reinforced concrete buildings result in the constitution of the displacement or rotation that can be formed on the combination region of the lower end of the column with the base by growing at the top of the column. Column-beam connection causes the buildings' damage as it is negatively affected from this behavior, which can be observed from recent earthquakes occurring in Turkey. This is also a significant reason for doing this study.

The first phase of the study, an ANSYS model was created to simulate the behavior of the sample of the experiment conducted by Canha *et al.* (2004, 2009) under the load and the results of finite element analysis of this model was compared with the results of the experimental study. By this way, verification of the finite element model was realized. Besides, it was observed that the results obtained from the finite element model were consistent with the empirical results showing the actual behavior. It was also seen that experimental capacity, displacement, the stress transfer model formed by means of experimental results, and the results of the analysis were compatible with each other.

In the second part of the study, the ANSYS model of the sockets whose sizes and reinforcements was determined according to TS9967 standard for different types of columns was created. Additionally, conducting finite element analyses under the horizontal loads determined according to DBYYHY (2007), the stress distribution of the force transferred from column to the socket wall was tried to be interpreted. It is aimed in this study that the results of this study is intended to be utilized by design engineers through the selection of the commonly used sizes for the column-socket base joint in the applications in Turkey.

To achieve the aim of this study, the ANSYS finite element program was selected and used. ANSYS finite element software is a useful tool for modeling the reinforced concrete building elements and non-linear properties of concrete and it involves the elements which the reinforcement can be put into. Furthermore, it is essential to model a three-dimensional contact problem between 2 continuum structure parts in order to examine the stress transfer in the study. This problem is got involved in the top of the list of non-linear complex analyses within mechanical literature. ANSYS finite element software also contains specific elements and wizards for solving the contact problems. The problem includes non-linear situation of concrete materials due to their nature and it is also a non-linear analysis due to the change in contact problem depending on the time. The analysis is also a quite complex problem that requires more computer time. ANSYS finite element program is extremely favorable software for the realization of the analysis; thus, results of the analyses were obtained in accordance with the experimental results. A model compatible with experimental results was established by means of the studies conducted for the first analysis and the contact parameters were confirmed to give results consistent with experimental results.

The stress values acting on the socket wall as a result of the analyses was varied according to contact surface stiffness factor (FKN). It was seen that while FKN value was increased, stress values were increased. The increase of FKN value also led to an increase in the number of iterations, which caused extending the time period for the analysis. Having examined the stress distribution is transferred from the column to the socket base, the stress distribution situated at the middle section of the socket seems to be effective in the triangle form in a region approximately with the half of the height of the socket' depth at the top and bottom end of the column. It was seen from the result of the analyses that the load and stress values proposed by the regulations were much higher values staying on the safe side.

German DIN 1045 and TS9967 regulation gave the same values for the stress and load transferred from the column to the socket. The stress and load values in Japanese regulation AIJ are much closer to the values obtained by the experimental and ANSYS finite element analyses. The load transfer model proposed by Canha *et al.* (2009) and used for the verification of ANSYS finite element model is approved to be very consistent with the results of ANSYS finite element analysis and to give successful results.

Within the result of the ANSYS finite element analysis, it was seen that the stress distribution transmitted from the column to the wall of the socket base was a triangular distribution of spread and it was effective at the half of the height of the socket. A design force value which is much more realistic and consistent with the experimental results is thought to be achieved by using this model. For this purpose, an equation which engineers can use for the design of reinforced concrete socket base was investigated. Reinforcing the upper end of the socket wall by means of the design force calculated using this equation and using the same reinforcement for its lower end are suggested in terms of ease of implementation and accountability.

References

ANSYS Inc. (2009), *Element reference. ANSYS Inc*., Canonsburg, 565, 1034, 1037, 1089.

Alyavuz, B. and Anıl, Ö. (2007), “Nonlinear finite element analysis of loading transferred from column to socket base”, *J. Fac. Eng. Arch. Gazi Univ*., **22**(3), 471-479.

Architectural Institute of Japan (1999), “AIJ standard for structural calculation of reinforced concrete structures”.

Bayındırlık ve İskan Bakanlığı (2007), “Deprem bölgelerinde yapılacak binalar hakkında yönetmelik”, *Resmi Gazete*, 26454. 10-19 (In Turkish).

Bulut, N., Anil, Ö. and Belgin, Ç.M. (2011), “Nonlinear finite element analysis of RC beams strengthened with CFRP strip against shear”, *Comput. Concrete*, 8(6), 717-733.

Canha, R.M.F., El Debs, M.K., Jaguaribe Jr, K.B. and El Debs, A.L.H. (2009), “Behavior of socket base connections emphasizing pedestal walls”, *ACI Struct. J*., **106**(3), 268-278.

Canha, R.M.F. (2004), “Estudo teórico-experimental da ligação pilar-fundação por meio de cálice em estruturas de concreto pré-moldado”, Phd Thesis, Escola de Engenharia de São Carlos da Universidade de São Paulo, São Carlos – SP, 143-159, 168-169, 173, 215.

DIN 1045 (1988) “Reinforced concrete structures – design and construction”, Beuth Verlag, Berlin.

Dogan, A.B. and Anil, Ö. (2010), “Nonlinear finite element analysis of effective CFRP bonding length and strain distribution along concrete-CFRP interface”, *Comput. Concrete*, 7(5), 437-453.

Drosopoulos, G.A., Stavroulakis, G.E. and Abdalla, K.M. (2012), “3D finite element analysis of end - plate steel joints”, *Steel Compos. Struct*., 12(2), 93-115.

Fabbrocino, B., Pecce, M.R. and Di Sarno, L. (2005), "Traditional versus innovative composite base column connections", *Adv. Steel Struct.*, **1**, 793-798.

Han, S.C., Nukulchai, W.K. and Lee, W.H. (2011), "A refined finite element for first-order plate and shell analysis", *Struct. Eng. Mech.*, **40**(2), 191-213.

Osanai, Y., Watanabe, F. and Okamoto, S. (1996), "Stress transfer mechanism of socket base connections with precast concrete columns", *ACI Struct. J.*, **93**(3), 266-276.

Saboori, B and Khalili, S.M.R. (2012), "Free vibration analysis of tapered FRP transmission poles with flexible joint by finite element method", *Struct. Eng. Mech.*, **42**(3), 409-424.

Shi, G., Liu, Z., Ban, H.Y., Zhang, Y., Shi, Y.J. and Wang, Y.Q. (2012), "Tests and finite element analysis on the local buckling of 420 MPa steel equal angle columns under axial compression", *Steel Compos. Struct.*, **12**(1), 31-51.

Türk Standardları Enstitüsü (1997), "Yapı elemanlarının boyutlandırılmasında alınacak yüklerin hesap değerleri", TSE TS-498, Ankara, 7 (In Turkish).

Türk Standardları Enstitüsü (2004), "Yapı elemanları, taşıyıcı sistemler ve yapılar - Prefabrike betonarme ve öngerilmeli betondan – Hesap esasları ile imalat ve montaj kuralları", TSE TS-9967, Ankara, 44-46, 101-102 (In Turkish).

CC