Numerical Investigation of Precast Grouted Sleeve Connection under Cyclic Loading Using ABAQUS

A.Hemamathi1,*, Sanjith Murugavel2, Binu Sukumar3, M.Usha Rani4

R.M.K. Engineering College, Department of Civil Engineering, 601206, Tamil Nadu, India.

Abstract. This paper deals with numerical investigation using finite element software, ABAQUS CAE, for the analysis of precast reinforced concrete (RC) column connected to the foundation containing protruding bar in grouted sleeve subjected to axial, and lateral loads, including the cyclic loading response. The monolithic connection of column and foundation is designed and the same has been used for the design of precast concrete column and foundation. The model focusses on the length of the protruding bar provided in the foundation that is connected to the column through the grouted sleeve. As there is lack of Indian standard codes in the design of precast concrete connections, an attempt has been made by varying the length of protruding bars. The model was subjected to quasi-static loading condition and parameters such as ultimate load carrying capacity, load-displacement hysteresis, ductility and plastic strain were studied and were compared with that of monolithic connection. The results proved that when the length of protruding bars provided was equal to development length of the bar provided, the results were comparable with that of monolithic connection.

1. Introduction

Construction industry is experiencing a boom as the use of precast technology is increasing in the recent decades. This is because of the several facts that the precast construction is rapid and less time consuming when compared with conventional construction practices. Also as the precast concrete elements are manufactured in controlled factory conditions, there is no compromise on the quality of the materials used. Therefore the erection speed is high and it can be used for any number of storeys at a faster pace. Pre-stressing in precast construction may reduce the size and number of the elements used. All the advantages of precast concrete technology leads to durability in the structure as a whole.

In any structure, connections are the integral part in transferring the load from one element to another. The connections must be properly designed to transfer these loads. In precast structure, the type of connections must be selected such that the connections influence the transfer of forces under vertical and horizontal loads. To achieve a satisfactory design, the designer must understand the behaviour and characteristics of the connections. The connections are of two types: wet connection and dry connection. In wet connection, the elements are connected using insitu concrete, in case of dry connections, mechanical connectors are used for transferring the load. So well detailed and constructed joint plays an important role in the integrity of the building.

The purpose of Connection and Joint is: 1) To transmit forces between structural components 2) To provide overall stability 3) To provide strength to the structure and 4) To prevent from external leakages.
The precast concrete technology also has its own disadvantages, because the failure in precast structure is attributed to the failure of connections and joints between the elements. Some of the reasons for failure of precast connections may be due to:

a. Improper Detailing of reinforcement in elements.
b. Inadequate lapping of steel in connection.
c. Poor quality grout used for grouting.
d. Improper erection methods.
e. Unskilled labours for executing work
f. Improper selection of type of connection.

2. Objective

• To compare the column-to-footing joint for cast-in-situ and precast RCC structure under cyclic loading using Finite element (FE) models.

• To study the effect of varying length of protruding bar in grouted sleeve precast column – foundation connection.

• To design an effective connection for precast column-to-footing using finite element method in ABAQUS 6.14.

2.1. Finite element model

Finite element method is a numerical technique used to simulate the given physical phenomenon reducing the physical prototypes and optimize components in their design phase. Using finite element analysis, it is possible to analyse structural system subjected to combination of any loading condition that can result in complex internal stress distributions. In case of structural elements, a combination of concrete, steel and grout, the complexity in analysis becomes quite challenging. Concrete shows non-linear behavior that increases the level of difficulty in analyzing the model. Also the influence of steel when reinforced with concrete provides more complexity. The collective complexity in the behavior can be well understood by the technique of finite element analysis (FEA). In this study, loading such as axial forces, bending moments, shear forces and torsional moments are involved and FEA will be able to provide more converging solution. In this study finite element software ABAQUS/CAE is used to model the precast column foundation connection.

2.2. Application of FEM

Finite element methods finds its application in many of the engineering problems including aerospace, mechanical and civil engineering. It finds its wide application in fluid-structure interaction, thermo-mechanical, Thermo-chemical, thermo-chemo-mechanical problems, biomechanics, biomedical engineering, piezoelectric, ferroelectric, and electromagnetics. It is also used to determine the stress analysis, slope stability analysis, soil structure interactions, seepage of fluids in soils and rocks, analysis of dams, tunnels, bore holes, propagation of stress waves and dynamic soil structure interaction.

Finite element methods have taken off in problems which involve large volume of calculations and where manual calculations have become more cumbersome. So there is need to use software to solve such simultaneous equations and provide the internal responses in a more convincing manner. Finite element software ABAQUS can analyse more complex problems and provide load-response based on the geometry and material properties. Once a model is properly calibrated to experimental results it can be used to provide further insight into those physical experiments or to artificially expand upon their test matrix without the need for costly physical specimens.
3. Methodology

![Methodology Diagram]

**Figure 1.** 1 Methodology of finite element analysis of precast column foundation connection

Fig. 1 explains the methodology of numerical study carried out on the grouted sleeve precast column foundation connection.

3.1. Selection of problem statement

Use of precast structure in place of monolithic structure will face many complications regarding capability of ultimate strength, connecting one element to another, detailing of reinforcement in elements, and providing a procedure for erection of elements at site.

A perfect detailing of precast structure connection is important, it is important to transmit the forces between the structural components and also provide overall stability and strength to the structure. Column-foundation connection act as base, in which entire structure is withstanding on it. Hence, we consider connection between column and foundation in precast structure for our thesis. To make an effective connection, proper detailing of reinforcement should be considered for enhance good ultimate strength capability and high energy dissipation capacity.

3.2. Design of column and footing

Before modelling in ABAQUS, design of reinforcement detailing of column and footing manually has to be done. We considered one of the foundations of two-story building (G+1) for design. Data such as material characteristics of steel and concrete, characteristics of soil (safe bearing capacity) were
assumed. Both monolithic and precast structure were designed according to Indian standard code. Analysis and design of column were made in ETABS (three-dimensional analysis of building system).

Dimension and reinforcement details of column are
Size of the column = 230 x 230 mm
Height of the column = 2000mm
Main reinforcing bars = 4 # 16mm diameter Fe 500 bar
Lateral ties = 8mm diameter Fe 500 bar@ 200mm center to center
Development length Ld. = 500mm
Dimension and reinforcement details of isolated square footing are
Size of the footing = 1350 x 1350 mm
Depth of the footing = 300 mm
10 mm diameter bar @ 125 mm c/c in both directions were provided.

Fig. 2 provides the detailing of column and foundation block

![Detailing of monolithic column foundation connection](image)

**Figure 2.** Detailing of monolithic column foundation connection

3.3 Modelling methodology

In this study a two-dimensional solid model of the column-footing and its reinforcement was modelled using the software Autocad. This model was parametrically defined so as to allow for easy development of modified geometry for future studies. This two dimensional model was then exported to ABAQUS/CAE software using top-down approach. Meshing of the components was done in finite element software.

3.3.1 Model components.

In finite element model, the column-footing connection was represented using a total of 34 parts. These can be categorized into seven groups and are identified as such by different colours in Figure.3
3.3.2 Mesh elements

In ABAQUS, while meshing the elements, three types are used; viz, line bodies, solid bodies, and shell bodies. Beam and/or truss elements are used to represent the line bodies, a wide variety of solid elements are used to represent solid bodies which range from four noded tetrahedral elements to polyhedral shapes with 14 or more faces, and shell elements are used to mesh shell bodies.

3.3.3 Eight node brick element

In this study, all deformable solids are modelled using eight noded brick element. Deformable solid in the study includes concrete and grout used inside the sleeve and that used for connecting the column at the bottom. A linear brick element C3D8R element is used in this model with reduced integration. The term “reduced integration” refers to the element having only a single integration point, used to evaluate the material response, as compared to having eight integration points with the C3D8 linear brick element with full integration.

3.3.4 Two Node Beam Elements

The reinforcing bars that are provided to resist bending, shear and relative rotation between adjacent beam elements are modelled using beam element. A two noded beam element is used to mesh the main longitudinal reinforcement and transverse spiral reinforcement in column.

3.3.5 Discrete Rigid Elements

To model the grout a special type of shell element was used in this study. Grout is used in the sleeve of the column and also at the bottom of the column is defined as a rigid body. These elements are defined in ABAQUS as non-deformable quadrilateral and triangular shell elements, R3D4 and R3D3.
respectively. Since the grout is modelled as discrete rigid element the need to mesh the internal volume of the lofted cap is eliminated. Also there is a reduction in the computational demand as the nodal values related to development of stiffness matrix is not stored.

3.3.6 Reactionary Boundary Conditions.

To prevent rigid body motion, the lower face of the footing were defined with constraints. The motion in all six degrees of freedom were defined as zero at the applicable nodes.

3.3.7 Load and displacement boundary conditions

The top of the column was defined as discrete rigid body and the model was loaded through the top geometry surface. Firstly, vertical axial load (31kN) was applied on the top of column in the form of pressure (0.586 N/mm²) or otherwise loading can be made through the reference point at Centre of the top of the column. In this Finite element modal, fixed axial load was applied at the step of automatic increment. In order to determine the energy dissipation capacity, load vs displacement characteristics (displacement control) was required. Hence cyclic displacement (15 mm) was providing at top edge of geometry along x-axis in the form of line load. Displacement was applied in the form of response spectrum as shown in figure 5

![Loading protocol of displacement](image)

**Figure 4.** Loading protocol for each cycle of displacement

4. Finite element modelling

4.1. Interactions and Constraints between Components.

To ensure strain compatibility between the various components, kinematic relationship between the components are to be defined within the finite element model. To define the interaction between the concrete and the steel reinforcement, embedded constraints was used. To define the interactions between the master surfaces and the slave surface representing the bottom of column and the top of footing, tie constraints was used.

4.2 Concrete.

In ABAQUS, concrete is modelled as Concrete Damaged Plasticity (CDP) model. It provides a general capability for modeling concrete and other quasi-brittle materials in all types of structures (beams, trusses, shells, and solids) and uses concepts of isotropic damaged elasticity in combination with isotropic tensile and compressive plasticity to represent the inelastic behavior of concrete. The present study makes use of the model proposed by of Lubliner et. al. (1989), with the modifications proposed
by Lee and Fenves (1998) to account for different evolution of strength under tension and compression. The evolution of the yield surface is controlled by the hardening variables. In terms of effective stresses, the yield function takes the form

$$F(\bar{\sigma}, \overline{\epsilon}^p) = \frac{1}{\kappa} \left[ q - 3\alpha\bar{p} + \beta(\overline{\epsilon}^p)\overline{\sigma} + \gamma(-\overline{\sigma}_{\max}) - \overline{\sigma}_c(\overline{\epsilon}^p) \right] \leq 0$$

The term $\gamma$ is a function of the constant $K_c$

$$\gamma = \frac{3(K_c - 1)}{2K_c - 1}$$

It is necessary to define values for $\psi$, $\epsilon$, $f_{b0}/f_{c0}$, $\kappa$, and $\nu$ in the model before performing the analysis. The ratio $f_{b0}/f_{c0}$ must be in the range of 1.10 and 1.16, here in this study the value assigned was 1.25. A default value of 0.1 was adopted for eccentricity parameter $\epsilon$. Constant $K_c$, described above, was used to determine $\gamma$ for cases of triaxial compression and it was provided with a value of 0.7. The value of $\psi$ assumed in the present study was 36° and the viscosity parameter should be assumed a very minimal value or set at zero. Here it was considered as 0.002 to improve stability in the model.

4.3 Compression stress-strain relationship.

The compression stress-strain relationship used in this study is based on Hognestad relationship (Hognestad, 1951). The compression stress is described by a parabola and the constitutive relationship is given by the Equation

$$f_c = f_c^0 + \left[ f_c^0 \left( \frac{\varepsilon_c}{\varepsilon_0} - \left( \frac{\varepsilon_c}{\varepsilon_0} \right)^2 \right) \right]$$

Where, $\varepsilon_c$ is defined as the strain and $f_c$ is defined as the stress at a given point in the relationship. The term $f_{c0}$ is the strain at the apex of the parabola and is taken as 1.8 times $f_c^0$ divided by the modulus of elasticity of concrete. In order to account for differences between cylinder strength and member strength, $f_c^0$ is taken as a reduction of the compressive strength of concrete cylinders $f_c^0$, generally on the order of 85% to 90% of $f_c^0$. In this study, $f_c^0$ was taken as such as $f_c^0$ in the material inputs. The value of the concrete elastic modulus was considered based on the equation given in IS 456-2000.

$$E_c = 5000 \sqrt{f_c^0}$$

4.4 Tensile stress-strain relationship.

In this study, the tensile stress-strain relationship was simplified greatly to reduce the computation demands. The magnitude of the tensile forces in the concrete are anticipated to be negligible relative to the tensile forces in the longitudinal reinforcement. Therefore a more simplified results were used in the modelling. The tensile stress increases linearly to a value of 1.0% of $f_c^0$, then increases linearly to a value of 1.1% of $f_c^0$ over the next 10 micro strain, and finally remains at a constant 1.1% of $f_c^0$.

4.5 Steel Reinforcement.

Classical metal plasticity was used in modelling the steel reinforcing bars in this study. The classical metal plasticity model in ABAQUS uses the Mises yield surface, which provides an isotropic yield surface. While modelling in ABAQUS, stress vs. plastic strain were provided to define both the initial yield point and the hardening behavior. A yield strength of 500Mpa was utilized in the development of the material inputs, despite the higher values shown in Table.1.
Table 1. Plasticity strain of steel Fe500

| Yield strength (N/mm²) | Plastic strain |
|------------------------|----------------|
| 360                    | 0              |
| 470                    | 0.2            |

In this study, the reinforcing steel behaviour was simplified as an effectively elastic-perfectly plastic behaviour. The inputs are based on a linear elastic response up to yielding and a constant stress from the point of yielding to the ultimate strain, with the exception of modifications made at the elastic-to-plastic transition point to assist convergence.

4.6 Grout.

It is made up of cement mortar with high compressive strength(60Mpa-110Mpa). In the FE model, grout was modeled using mass density, mechanical properties such as elasticity and plasticity in Abaqus. Plasticity of grout was expressed by Nominal Concrete Damaged Plasticity (CDP) as already discussed.

Table 2. Material parameter for grout

| Mechanical property     | Values            |
|-------------------------|-------------------|
| Mass density            | 19.4E-6 Kg/mm³    |
| Elastic modulus         | 21000 N/mm²       |
| Ultimate strength       | 60 N/mm²          |
| Poisson’s ratio         | 0.2               |

4.7 Corrugated steel sleeve.

The corrugated steel sleeve was modelled using mass density, mechanical properties such as elasticity and plasticity in ABAQUS.

Table 3. Material parameter of corrugated steel

| Mechanical property     | Values            |
|-------------------------|-------------------|
| Mass density            | 0.955E-6 Kg/mm³   |
| Elastic modulus         | 900 N/mm²         |
| Ultimate strength       | 25 N/mm²          |
| Poisson’s ratio         | 0.3               |

4.8 Solution settings

In order to obtain more convergent results in ABAQUS it is required to modify the convergence settings from their defaults. Our study encompasses geometric nonlinearities, material nonlinearities, and interactions. This step of modifying the settings should be done cautiously as these settings vary between the various loading steps. Solver default were adopted for the first and second step where the axial load was applied and the boundary conditions were set. In the subsequent steps the values of I0 and IR are increased from 4 to 8 and from 8 to 10, respectively. I0 represents the number of equilibrium iterations.
allowed. Once the residuals were not increasing, in next consecutive steps the solver checks were ensured. In this study, the model is subjected to multiple interactions of nonlinear materials, the convergence is expected to be non-mono- tonic. This requires an increase in \( I_0 \). After this the rate of convergence check is performed by increasing the number of equilibrium iterations \( IR \). As concrete is modelled as asymmetric, an unsymmetrical equation solver is required.

### Table 4. Step Incrementation Settings

| Step | End Point | Step Length | Initial Increment | Max Increment | Min Increment |
|------|-----------|-------------|-------------------|---------------|---------------|
| 1    | Initial State* | N/A* | N/A* | N/A* | N/A* |
| 2    | Axial Load | 1 | 0.001 | 0.002 | 1.0E-05 |
| 2    | CP-1 | 1 | 0.001 | 0.002 | 1.0E-10 |
| 2    | Cp-2 | 1 | 0.005 | 0.005 | 1.0E-15 |

5. ANALYSIS AND RESULTS

A. Finite Element Analysis Overview

This section describes the finite element analysis performed in this study. Before the analysing of FE model, data’s check should be performed in the Abaqus. If error will occur, again modelling should be done with proper correction as shown in error file. Otherwise analysis should be performed. First the base model is analysed (monolithic connection) and is detailed in section 5.1, followed by a more in-depth look at individual aspects of the model. Section 5.2. Section B briefly describes the analysis result of precast column-foundation connection with varying protruding bar in grouted sleeve respectively. Finally, the comparative studies were made with analysis report.

5.1 Analysis and Result of Base Model

Monolithic reinforced concrete (RC) column-footing structure acts as base model in this study and its analysis was done first. Cyclic displacement of column with respect to time are shown in figure 5. From the Input of lateral displacement, it induces the combination of both reaction force and stress at the top of column and reinforcement at connection respectively.

Due to cyclic lateral displacement, column will experience Hysteresis effect. The displacement of elastic materials due to a varying load lags behind the fields. This effect is called hysteresis, and the term is used to describe any system in whose response depends not only on its current state, but also upon its past history. As result of varying load, load vs displacement curve extends in the form of hysteresis loop as shown in figure 5 and 6.
5.2 Analysis and result of precast model

Reinforcement detailing of three different type of precast models were done similar to that of monolithic connection. Analysis procedure were followed as same as for all three FE models but only difference in varying length of protruding bar in grouted sleeve. The grouted sleeve length provided was taken for three models as L_d, L_d +10d, and L_d -10d, where L_d is the development length of the bar and d is the diameter of the bar.

Due to cyclic loading, connection between column and footing experience plastic strain (PEMAG) as shown in figure 8. As displacement increases, sudden failure will occur due to strain at connection.
B. COMPARATIVE STUDY ON PRECAST FE MODELS.

This section provides a brief description of the comparative studies made on different precast model through finite element analysis performed in ABAQUS. The parameters considered for the study were ultimate load capacity of element under dynamic condition, load-displacement hysteretic loops, displacement ductility and plastic strain of the specimen.

1. Ultimate load capacity. It is the Ability of element to withstand the external load at certain limit. The maximum load resisted by each of the precast model is shown in figure 9 and is compared with three base model (Monolithic connection). It is found that the load carrying capacity for a displacement of 20 mm was high in the monolithic connection when compared to precast models. The ultimate load of base model is 29 kN. The models Ld and Ld +10 was able to resist load in par with base model and it was 25.7 kN and 24.7 kN respectively. The model Ld -10 was able to resist a load of 13.4 kN only.

Figure 8. Plastic strain analysis of different precast FE models

Figure 9: Varying ultimate load for different precast models.
From the graph above, precast column-footing connection with protruding bar of length (Ld) 725 mm have more load carrying capacity (25722N for 20 mm) compare to other two models (Ld+10d and Ld-10d). On other hand, precast column with protruding bar of length 565mm exhibits low load carrying capacity compare to other two models. Load carrying capacity of precast structure are minimum compare to monolithic RC structure.

2. Load-displacement hysteretic loops:

Load-displacement hysteresis loop help s to study the behaviour of connection under cyclic loading and to observe its pinching effect. Also the hysteresis loops gives a knowledge on yield capacity and ultimate load capacity of the models. Figure 10 a to figure 10 d, gives the load-displacement hysteretic behaviour of all three precast models and base model.

![Hysteretic loop](image1)

**Figure 10. a** Load vs Displacement curve of base model

![Hysteretic loop](image2)

**Figure 10.b** Load vs Displacement curve of Ld FE model

![Hysteretic loop](image3)

**Figure 10.c** Load vs Displacement curve of Ld + 10 d FE model

![Hysteretic loop](image4)

**Figure 10.d** Load vs Displacement curve of Ld - 10d FE model

3. Ductility. Ductility is known as the ability of material to deform beyond its elastic yield limit into the plastic zone. Ductility is important to achieve the inelastic zone and in inelastic energy dissipation. If the material is not ductile then it does not have the stamina to go into hundreds of inelastic cycles in case of an earthquake. Ductility of structure is determined by their energy dissipation capacity.

Generally, energy dissipation capacity is calculated by area of closed hysteresis loop. So, obtained hysteresis loop is adjust to closed loop. Detailed study on energy dissipation capacity of different finite element models. It concluded that Precast Ld model (having protruding bar length 725mm) have more energy dissipation capacity compare than monolithic structure. Hence precast Ld
model have more ductility than other models. Figure 11 and 12 explains the ductility and energy dissipation capacity of all models.

![Figure 11. Energy dissipation of all models upto displacement of 20 mm](image1)

![Figure 12. Cumulative Energy dissipated by all models](image2)

4. **Magnitude of plastic strain.** Strain is defined as the relative change in shape or size of an object with externally applied load. From figure 13, it shows development of plastic strain at connection due to increasing lateral force on it. From this studies, it can be seen that the precast column with protruding bar of 565 mm experience high plastic strain at base of column and stress at reinforcement bar compare to other two precast models.
4. Conclusion

The model had mixed successes when attempting to simulate the response of the precast column-foundation in this study. The failure of column-footing connection can’t be predicted completely in this model study. But a comparison helps to understand the projection of protruding bars to be provided in grouted sleeve precast column and foundation connection.

1. By comparing the ultimate load capacity of precast models, column detailed with protruding bar of length 725mm in grouted sleeve has showed maximum value.
2. From the ductility, precast model with 725mm length protruding bar have more energy dissipation capacity compare with other models. If length of protruding bar increases, ductility of column will gradually decrease.
3. Through studies made on stress-strain development by lateral load in precast models, when length of protruding bar decreases, it will induce more stress at footing starter bar and strain at base of column.

References

[1]. Adam Christopher Morgan. (2018). Nonlinear finite analysis of repaired column. Department of civil engineering, Missouri institute of science and technology. https://scholarsmine.mst.edu/masters_theses/7771.
[2]. Amin Einea. (1993). Grout-filled pipe splices for precast concrete construction. Prestress concrete institute, Page number: 82-93. https://www.pci.org/PCI_Docs.
[3]. Dr.J.N.Reddy. (2004). An introduction to nonlinear finite element analysis. Oxford University Press. https://soaneemrana.org.
[4]. Dr.K.P.Jaya.(2016) Connections in precast structure. CSIR - CBRI Roorkee. https://cbri.res.in/wp-content/uploads/2016/03/Dr.KP-Jaya.
[5]. Dr. Milad Hafezolghorani, Farzad Hejazi and Mohd Saleh Bin Jaafar (2017). Simplified damage plasticity model for concrete. Scientific Paper Structural Engineering International. EP- 78. https://www.tandfonline.com/doi/abs/10.2749/101686616X1081.
[6]. Dr.P.Seshu.(2004). Finite element analysis. Prentice-Hall of India Pvt.Ltd; 1st edition. http://www.sasurieengg.com/e-course-material.
[7]. Dr.R.K.Bansal.(2009). Strength of material. Laxmi Publications; Sixth edition (2018).
[8]. Eliya henin and George morocus (2015). Introduce of reinforcing bar splice sleeve. 14th International Conference Structural & Geotechnical Engineering 2015 (ICSGE14), At Cairo-Egypt.

[9]. Elsayed Mashaly (2010). Finite element analysis of footing-to-column joints in steel frame under cyclic loading. Techno-Press Publishers of international journals and conference proceedings. http:// www.elsevier.com/locate/ doi: 10.1016/j.aej.2011.01.012.

[10]. Hubert Batchmann and Alfred Steinle.(2012). Precast concrete structures. Wiley India Pvt. Ltd. (2018). https://onlinelibrary.wiley.com/doi/book/10.1002/9783433600962.

[11]. Hussam Ali Mohammed (2018). Analysis and design of precast concrete structures. Al-Furat Al-Awsat Technical University. https://www.researchgate.net/publication/328391948.

[12]. Hyungoo Kang and J Inkoo Kim (2017). Response of a column-footing connection subjected to load. Structural Engineering & Mechanics. EP - 136. DO - 10.12989/sem.2017.63.1.125.

[13]. IS: 456-2000. Indian Standard code of practice for plain and reinforced concrete, Bureau of Indian Standards, New Delhi

[14]. N.Subramanian (2014). Design of reinforced concrete structure. Oxford University Press (26 December 2014).

[15]. SP 16 (1978). Design aids to code of practice for plain and reinforced concrete. Bureau of Indian Standards, New Delhi