International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 07 Issue: 02 | Feb 2020 www.irjet.net p-ISSN: 2395-0072

A REVIEW ON FINITE ELEMENT MODELLING AND NUMERICAL STUDY OF PRECAST STRUCTURAL ELEMENTS

S. Parkavi[1], Dr. K. Ramadevi[2] and V.Sindhu[3]

[1]P.G Scholar, M.E. Structural Engineering, Kumaraguru College of Technology, Coimbatore,
[2]Professor, Civil Engineering Department, Kumaraguru College of Technology, Coimbatore
[3]P.G Scholar, M.E. Structural Engineering, Kumaraguru College of Technology, Coimbatore.

Abstract - *Precast structures have a major advantage for its rapid construction in order to maintain it simple connections with minimum width should be maintained for the least material and labor usage during the installation stage. Analytically resolving displacements and stresses in precast concrete material was tedious task and engineers had to depend on empirical formulas since concrete consists of heterogeneous material and creep and shrinkage influenced deformations in it. Due to these difficulties engineers in past had been facing complications in handling such problems, but with the enhancement of digital computerization and modern numerical methods for analysis such as finite element method, these issues can be addressed in a very effective way. There are two ways to carry the nonlinear finite element problems using ABAQUS and ANSYS software. In the analysis, material and geometric non linearities of concrete and steel are considered. Finite Element Modelling software is used to simulate the damage plasticity model provided by the concrete material behavior. Finite element analyses and own experimental results are compared to verify the finite element models. In this paper review on the process of finite element study involving modelling of elements, mesh sensitivity study and crack modelling is reviewed.*

***Key Words*: Mesh sensitivity, ABAQUS, ANSYS, Finite Element Modelling, Precast Crack Modelling.**

1. INTRODUCTION

Precast buildings are highly beneficial in the construction industry for its speed in construction and in providing high strength. Precast is an integrated building system based on components and connections. The durability of these buildings is enhanced by the type of connection adopted. They carry the predefined loads in different structures and resist climatic loads. Installing Industrially produced precast connections are fast and easy. According to local building conditions and restrictions, and loads on the structure demands for precast connection types may vary. Connecting loops, ties and fixing sockets are done for the precast walls. Loop connections connect the vertical joints of precast wall panels one another. Connecting loops are easy to install by opening the wall cover, bending the loops to the operating position and connecting them to reinforcement. As like loop connections various connections are used in the precast structures.

2. LITERATURE REVIEW

Literature based on the modelling of precast connections, mesh sensitivity study and crack modelling of specimens. From the detailed literature review, inference is studied.

1. Tang Lei et al (2015) investigated on wall connections which are subjected to in-plane lateral force. Multi linear isotropic strengthening model and William Warnker failure criterion is adopted for modelling the concrete by using ANSYS software. Loading is applied as both load control and displacement control. Cracks occurred at the bottom and wall interface. Load vs Lateral displacement has been plotted.

2. Hafez Taheri et al (2016) investigated Contemporary connection and proposed connection which were compared to find the rotational loading capacity. The interactions provided here is surface to surface interactions. Six boundary conditions were included Urx, Ury, Urz, Mx, My, Mz. Meshing provided here is structured and sweep, Concrete is meshed as brick element as C3D8R, Steel is meshed as wire element T3D2 in ABAQUS software. It was found that the displacement of the panel in the proposed connection is less than the corresponding value in the loop connection.

3. Ramin Vagheia et al (2013) investigated on analysis of wall panel connections. Analysis adopted here is Dynamic explicit nonlinear analysis. The wall connections are subjected to in-plane lateral ground movement. Meshing adopted for concrete and steel are C3D8R and T3D2 in ABAQUS software. Loading given are self-weight of wall as gravity load and lateral load as 0-10 mm/ 2 sec for 20 seconds. Investigation is done for maximum principal stress, total deformation and absolute plastic strain

4. S. Sudhakar, et al (2019) investigated on bolted wall to wall connections. Cover plates and bolt and nut are modelled separately as per the mesh sensitivity study five types of meshes are adopted for the whole wall to wall connections. The wall connections are subjected to both in-plane and out plane lateral load including dead load, live load and wind load. Analysis is done in ANSYS software. Meshing adopted here is Hexa mesh for each mesh size equivalent stresses of every element is individually studied.

5. Vidjeapriya. R, et al (2013) investigated on Beam column connections. Beam column connections is done by using a J bolt and cleat angle. Nonlinear Analysis is done by ANSYS software. Elements included were solid and contact. Modeling is done in 3 types as discrete model, Smeared model and embedded model. Concrete is modelled as Solid 65 and steel is modelled as Link 8 elements. Analysis is done in ANSYS software. The wall connections are subjected to reversal loading. The load displacement relationship, ductility and energy dissipation are studied.

6. L.Dahmani, et al (2010) investigated on beam subjected to flexural loading. Reinforced concrete beam is been developed with Three-dimensional nonlinear finite element model. No**n**linear Analysis is done by ANSYS software. SOLID65 solid elements - concrete. Smeared reinforcement is used. Hand calculated results has been compared for the concrete beam. The ability of the model to capture the critical crack regions, loads and deflections for several types of loadings has been illustrated in the reinforced concrete beam.

7. Chong-fang Sun, et al (2017) investigated on unbonded wall to wall connections by using rabbets. The new connection proposed here is the rabbet-unbonded horizontal connection, which is composed of rabbets and unbonded rebar segments. Three specimens were tested with different parameters under cyclic quasi-static loading. A parameter analysis using a finite element model proved that the ductility and energy dissipation of a shear wall with the rabbet-unbonded horizontal connection increased with the unbonded length and level. Meshing adopted for concrete and steel are C3D8R and T3D2 in ABAQUS software.

8. Zenagebriel Gebremedhn, et al (2018) investigated on u shaped box culvert. Its components consisted of two symmetrical u-shaped structures joined together using the tip at the end of the bell. The finite element method has been chosen are the standard three dimensional solid and wire elements. Model adopted is Concrete damage plasticity model. Bonded rebars are embedded in the concrete with no slippage. The unbonded length proportionally increased the ductility and energy dissipation. The unbonded length depends on axial compression ratio and the unbonded level.

9. Mazizah Ezdiani Mohamad, et al (2018) investigated on interface modelling of precast concrete slab and toppings. The interface shear strength between precast concrete slab and cast-in place concrete topping. Finite Element Modeling package ABAQUS 6.12 was used to model the interface bond of concrete-to-concrete layers. The push-off test specimens featured segments of precast concrete slabs with a variety of surface textures. Failure of the bonded interfaces was modeled with cohesive zone model (CZM) approach. The parameters used in the analysis include interface shear strength, fracture energy and elastic shear stiffness. The study shows that the difference between the model and experimental results is relatively small.

10. Carlos Coronado, et al (2015) investigated on mesh sensitivity study for concrete panels (i.e., slabs and walls) used in nuclear power plant construction. Shell and solid elements in the ANSYS are used. The accuracy of the finite element solutions is benchmarked against results from refined solid element models. The use of element nodal forces vs. shell element forces taken at the center of gravity of the elements is discussed, given that it is common practice to use the element centroid results for ease of design. It shows that significant error or underestimation of element design results may be introduced when using element centroid results.

11. Mostafiz Emtiaz(2017) investigated on the vulnerability of the flat slab-column connection subjected to lateral loading. It is checked through the analyses of finite element analysis. The finite element analyses have been conducted with ABAQUS software. Elastoplastic CDP model is used a material modelling for the reinforced concrete. The concrete damage plasticity model and linear elastic plastic model are used in this study. Mesh size adopted here is 10 mm. It is found that thickness of the slab, reinforcement ratio, usage of bent bars, high strength materials are the important factors in punching shear failure in the slab-column connection.

12. Uwe Starossek(2008) investigated on the interaction between steel tube and concrete core under three types of loading. Two and three-dimensional nonlinear finite element models are developed in ABAQUS. The damage plasticity model provided by ABAQUS is used to simulate the concrete material behavior. Comparisons between the finite element analyses and own experimental results are made. Parametric studies using the numerical models are performed to investigate the effects of diameter-to-thickness ratio, uniaxial compressive strength of concrete, length of shear connectors, and the tensile strength of shear connectors.

13. Neha S. Badiger(2014) investigated on four point bending analysis is carried out in reinforced concrete beam. The results of the beam with respect to mesh density, varying depths, use of steel cushions for support and loading points, effect of shear reinforcement on flexure behavior, impact of tension reinforcement on behavior of the beam are analyzed and discussed. Finite element software ANSYS 13.0 is used for modeling and analysis by conducting nonlinear static analysis.

14. Ali. Laftah. Abbas (2017) evaluated on masonry wall modeling using the representation techniques adopted in literature and use the appropriate technique to represent masonry room by using the ABAQUS software under the seismic load. Three interface modeling such as micro, simplified micro and macro are adopted. The concrete cracking behavior is studied by concrete damage plasticity model. Tie constraints are used for connection modeling loading applied here are gravity load, lateral load.it is also analyzed under time history analysis.

15. Xu, L., Shen, X. and Shen, J.(2014) investigated experimental observation and FEM simulation to analyze the hybrid shear wall system. Three sets of full-scale shear walls including two hybrid shear walls and one cast-in-place shear wall were tested subjected to cyclic horizontal. A series of 3D non-linear finite element models created in ABAQUS to simulate the experiments. The pushover analysis method was employed. The response of the structure was computed both at macro and micro levels. A comprehensive parametric study was conducted to determine seismic design parameters, and to explore damage mechanism of the hybrid shear wall system.

3. CONCLUSIONS

1. Different types of connection modelling have been studied.
2. Finite element modelling in both ANSYS and ABAQUS finite element software has been reviewed.
3. Type of loading condition, meshing techniques and the various numerical analysis adopted in various papers is studied.

REFERENCES

1. Tang Lei" Nonlinear Finite Element Analysis of New Precast Concrete Shear Wall" International Conference on Civil Engineering and Transportation (ICCET 2015), Pp. 322-326, (2015)
2. Ramin Vagheia, Farzad Hejazia, Hafez Taheria, Mohd Saleh Jaafarb and Abang Abdullah Abang Alic "Evaluate Performance of Precast Concrete wall to wall Connection" International Conference on Chemical, Biological and Environmental Engineering (ICBEE 2013), APCBEE Procedia 9, Pp 285 – 290,2014.
3. S. Sudhakar, G. Srinivasan "Behavior of Bolted Connection in the Vertical Joint of Precast Wall Panel ", International Research Journal of Engineering and Technology IRJET, ISSN: 2395-0072, Volume 06, Issue 05, Pp 1883-1888, May 2019.
4. Mostafiz Emtiaz a, A.S.M. Alauddin Al Azad a, H. M. Shahin b and Sultan Al Shafian "Numerical Analysis of a Reinforced Concrete Slab-Column Connection Subjected to Lateral & Vertical Loading" Proceedings of the International Multiconference of Engineers and Computer Scientists , IMECS , Volume 2,Pp 2651-66, March 2017.
5. Vidjeapriya. R, Bahurudeen. A, Jaya. K.P "Nonlinear analysis of exterior precast beam-column J-Bolt and cleat angle connections" International Journal of Civil and Structural Engineering(IJCSER),Volume 4,Pp 324-368, 2013.
6. L. Dahmani, a A. Khennane, and S. Kacia "Crack Identification In Reinforced Concrete Beams using ANSYS Software "Strength of Materials, Volume 42, Issue No. 2, 2010.
7. Chong-fang Sun, Shu-ting Liang, Xiao-jun Zhu4, Hu Li, Jian-min Guo2, Gang Li, Ya-min Song3and Dong-yue Wu" Experimental Study and Numerical Simulation of Precast Shear Wall with Rabbet-Unbonded Horizontal Connection" International Journal of Concrete Structures and Materials, Volume No 4, Issue No: 6,2020.
8. Zenagebriel Gebremedhn, Guofu Qiao, Jilong Li" Finite Element Modeling and Analysis of Precast Reinforced Concrete U-Shaped Box Culvert Using ABAQUS" American Journal of Civil Engineering. Vol. 6, No. 5, 2018.
9. Mazizah Ezdiani Mohamad, Izni Syahrizal Ibrahim, Redzuan Abdullah" Finite Element Modeling of the Interfacial Behavior at Surface Roughness Concrete Without the Projecting Steel" International Journal of Advances in Mechanical and Civil Engineering, ISSN: 2394-2827 Volume-4, Issue-3, Jun.-2017.
10. Carlos Coronado, Damon Reigles, Sungjin Bae and Javeed Munshi "Finite Element Mesh Sensitivity Study Using Ansys For Analysis And Design Of Nuclear Concrete Structures' SMiRT, Volume 21, Pages 6-11,2011.
11. Mostafiz Emtiaz a, A.S.M. Alauddin Al Azad a, H. M. Shahin b and Sultan Al Shafian "Numerical Analysis of a Reinforced Concrete Slab-Column Connection Subjected to Lateral & Vertical Loading",International Multi Conference of Engineers and Computer Scientists , IMECS, Vol II, 2017.
12. Uwe Starossek, Nabil Falah and Thomas Löhning" Numerical Analyses of the Force Transfer in Concrete-Filled Steel Tube Columns" Structural Engineering & Mechanics Pp 2651 -2666, May 2010.
13. Neha S. Badiger, Kiran M. Malipati "Parametric Study on Reinforced Concrete Beam using ANSYS"Civil and Environmental Research ,ISSN 2225-0514 ,Vol.6, No.8, Pp 88-93, 2014.
14. Ali Laftah. Abbas and Maan. H. Saeed, "Representation of the Masonry Walls Techniques by Using FEM" Australian Journal of Basic and Applied Sciences, 11(13) November 2017, Pages: 39-48, November 2017.
15. Xu, L., Shen, X. and Shen, J. "Seismic Study of Hybrid Shear Wall'", Proceedings of the 10th National Conference in Earthquake Engineering, Earthquake Engineering Research Institute, 2014.