Determination of Member Connection Stiffness For Semi-Precast High-Rise Building

Tran Thi Thuy Van 1, Nguyen Trung Kien 2, Do Tien Thinh 2, Hoang T.L. Quyen 3

1 Hanoi Architectural University (HAU), Vietnam
2 Vietnam Institute for Building Science and Technology (IBST), Vietnam
3 National Research Moscow State University of Civil Engineering (NTU MGSU), Russia

ttthvan.hau@gmail.com

Abstract. Currently, in Vietnam, the semi-precast high-rise buildings are popular construction structures because of material saving and low construction cost, meeting the requirements of a part of residents with medium and low incomes. However, in the assembly process of these structures, the technical requirements for the connections between prefabricated components as well as between prefabricated components and monolithic components are very important and required a proper analysis. Accurate modeling is very important in structural analysis, as well as analysis of behavior for high-rise semi-precast buildings, especially when they are in the possible earthquake areas. In order to be able to model the real structure, the value of connection stiffness between the components needs to be determined relative correctly. The paper presents how to determine the member connection stiffness of semi-precast high-rise buildings using the finite element method. Verifying the correctness of the member connection stiffness value is carried out through dynamic parameters such as natural frequency and the damping ratio obtained by the experiments of the miniature model of semi-precast high-rise buildings on shaking table test. From there, some comments and recommendations on the suitable model for semi-precast high-rise buildings will be proposed for analysis of these structures more accurately.

1. Introduction

In [1], the concepts of member connection in semi-precast high-rise buildings are presented. To simplify in structural analysis for this type of building, it is acceptable to assume the connection between main members as following types: fixed connection, semi-rigid connections and pinned connections (figure 1). However, in the case of considering of fixed connections, the connection still has a certain elasticity due to deformation of the beam-column, beam-floor connections and local deformation of the joints. The assumption of fixed connection has neglected the elasticity of the connection, the displacement and the natural frequency of the structure are smaller than the actual. On the other hand, in the case of considering of pinned connection, the maximum internal forces will appear in the middle of the spans and the value of bending moment at the end span is zero, but in fact, under loading the internal forces appear in all sections. Similarly, the displacement and natural frequency of the system with pinned member connection will be different from the system with specific member connection stiffness. Thus, it can be seen that to consider the stiffness of member connections has significant meaning in structural analysis of semi-precast high-rise buildings, the member connection stiffness affects the redistribution of internal forces and displacement in the system,
as well as natural frequency of the system. However, in structural analysis for this type of works, the member connection stiffness has not been mentioned much and studied thoroughly. The model, taking into account the member connection stiffness is difficult to apply in practice if lack of appropriate methods and calculation tools for modeling. Also, the determination of member connection stiffness is difficult to carried out if only manual calculation methods are used. With the development of information technology today, especially the application of computational programming softwares for structural analysis, the model, approaching to the actual semi-precast high-rise buildings can be applied without difficulties because of determination of member connection stiffness can be carried out by programming softwares. These softwares of structural analysis based on using the finite element method.

2. Determination of member connection stiffness using finite element method
The finite element method is the most popular numerical method in static and dynamic structural analysis. Programming software Etabs is specialized software used to analyze for high-rise buildings. The member connection stiffness can be determined by Etabs software using iteration and the method of gradual test with reference of specific natural frequency obtained by experimental method. The determination of member connection stiffness by Etabs software can be handled as follows. The member connection is consider as partial fixity. The way to release a specific percentage of moments and shear at the supports is to provide the reduced stiffness of the members. In Etabs software there are options for this by access the following sets: Assign>Frame Lines>Frame Releases/Partial Fixity. There are many sets of combination in various checkboxes of this form for release. One for start point of the member and the other for end point. Here, note that when select the option either “start” or “end”, the boxes for the stiffness values will be enabled, by default the values in these boxes are zero means the stiffness is reduced to zero, so making the connection as pinned connection.

In order to make partial frame releases, it will put the “frame partial fixity springs” values in the start and end boxes. Firstly, the value put in partial spring need to calculate the stiffness of the fully fixed support and this is calculated as \(k=4EI/L, 2EI/L\) based on the support conditions. For example a fixed-fixed end beam having uniformly distributed load will have support stiffness of 2EI/L in one end, 4EI/L on the other end, where \(k\) - fully fixed stiffness of the connection, \(E\) - Modulus of elasticity of the member, \(I\) - Moment of inertia in the direction of analysis, \(L\)=length of the member between supports.

Secondly, after calculating the actual stiffness value of the connection, it will be need to multiply it by the reduction factor by which need to reduce the moment, shear etc. The reduction factor is determined as ratio \(n/(1-n)\), where \(n\) is the percentage to reduce. For example if reduce by 25% get
reduction factor in one end point: 0.25/(1-0.25) = 0.33; in the other end point the reduction factor: (1-0.25)/0.25 = 3

Figure 2. Partial fixity in Etabs software

It will need to multiply 0.33 with 4EI/L and 2EI/L with 3 to get the final spring stiffness value and put it in ETABS as shown in Figure 2. It may not get the moment values reduced by that percentage by which it applied the reduction factor. So this process is iterative. It have to change the stiffness values based on many iterations unless it get the desired results. This is because the remaining moment should be redistributed to the other elements of the structure.

3. Experiment on shaking table test
In order to verify the value of stiffness of member connection by finite element method, it is possible to verify experimentally through dynamic parameters, such as natural frequency, displacement and deformation of structures. In order to implement experiments for semi-precast high-rise buildings, one of the methods that are commonly used and suitable for the conditions in Vietnam is to conduct experiments on shaking table test. However, one of the important input parameters of this experiment is to select an appropriate artificial acceleration diagram to obtain reasonable results, assessing the actual work of the project.

In this paper, an experiment on shaking table was conducted at the earthquake laboratory of Vietnam Institute for Building Science and Technology (IBST). The shaking table test has the following technical parameters:
- Surface size of shaking table: 3x3m;
- Motion with 2 degrees of freedom X and Y;
- Model weight: <10 ton;
- Center of gravity of model: <3m
- Maximum acceleration is 1.1g if model weight is 10 ton;
- Maximum acceleration is 2.6g if no load;
- Maximum test sample mass: 10 tons;
- Maximum moment torque: 30 tm;
- Horizontal mass eccentricity: 1m
- 2 hydraulic jacks in 2 X and Y directions with capacity of each size ±250kN.

4. Example
The experiment is carried out on small scale specimen of 12 storey semi-precast high-rise buildings of social housing projects “Experimental housing for workers” by Vinaconex of Vietnam Construction and Import-Export Joint Stock Corporation, Vinaconex company is investor and Vinaconex Xuan Mai Design Consultant Joint Stock Company is the design unit. The project is built in the area of Kim Chung commune, Dong Anh district, Hanoi and has the plan as shown in figure 3.
Small scale specimen is chosen by similitude theory for seismic testing of structures [2].

The main parameters of small scale specimen as following:
- Surface size of small scale specimen of 12-storey semi-precast building is 2.0x2.1m;
- Earthquake load $a_{gR}=0.1 g$ (iterative circle 475 years and 975 years); ground class D determined in TCVN 9386: 2012 [4].
- 1st floor is 450mm high;
- Floor 2 to 12 is 275mm high;
- Overall height of building is 3475mm

The bearing load structure of the building includes: the middle hard core is made of reinforced concrete constructed in place. Pre-fabricated beams and columns and floors are assembled, floor are compensated in place.

The process of structural analysis is conducted using Etabs software presented in section 2. The calculation model is shown in figure 4.
The calculation results of analysis are shown in Table 1.

**Table 1. Natural frequency in accordance with member connection stiffness**

| Beam-column connection | Value of bending moment at end/Value of bending moment at middle span (%) | Natural frequency (Hz) |
|-------------------------|--------------------------------------------------------------------------------|------------------------|
|                         |                                                                              | In axis X | In axis Y |
| Fixed connection        |                                                                              | 6.65      | 4.85      |
| Pinned connection       |                                                                              | 5.17      | 3.06      |
| Stiffness of connection | - k = 8                                                                       | 18        | 5.98      | 3.85      |
|                         | - k = 10                                                                      | 22        | 6.29      | 3.87      |
|                         | - k = 12                                                                      | 25        | 6.35      | 3.91      |
|                         | - k = 15                                                                      | 29        | 6.41      | 3.97      |

The experiment conducted on shaking table test according to [3] illustrated in figure 5 and figure 6.

**Figure 5.** Small scale specimen of 12-storey semi-precast building [3]
TCVN 9386: 2012 [4] requires the testing process to be carried out with three different acceleration diagrams. This is a fairly new experiment in Vietnam, so the research team at the Vietnam Institute for Building Science and Technology (IBST) has studied many experiments in the world such as Chinese International University, NCREE Earthquake Research Laboratory in Taiwan,... and decided to use the following 3 types of schema:

- (1) Acceleration diagram of the El Centro 1940 earthquake;
- (2) Acceleration diagram of Pasadena earthquake in 1952
- (3) Artificial acceleration diagram. This diagram is established in accordance with the elastic reaction spectrum in TCVN 9386: 2012 according to type D ground of Hanoi location. The first and second types of diagrams are the diagrams recorded from real earthquakes, but the intensity is different from the experimental target of level 7. The research team used the reduction method (acceleration value) corresponds to the acceleration values of the 95-year, 475-year and 2475-year of iterative cycles according to TCVN 9386: 2012. With the using of third diagram (3), the artificial acceleration diagram is presented in [1] with adjusting the amplitude corresponding to iterative cycles obtained from the first and the second above mentioned acceleration diagrams. The artificial acceleration diagram is determined by performing same steps for different iterative cycles: 95 years, 475 years.
**Figure 7.** Artificial acceleration diagram for corresponding experimental model with 95-year of iterative cycles [3]

**Figure 8.** Artificial spectra and target spectra of 95-year of iterative cycles [3]

**Figure 9.** Artificial acceleration diagram for corresponding experimental model with 475-year of iterative cycles

**Figure 10.** Artificial spectra and target spectra of 475-year of iterative cycles [3]
Table 2. Values of natural frequency and damping ratio obtained from experiment of small-scale specimen of 12-storey building

|                      | Before loading                                      | After loading with iterative cycle 95 years | After loading with iterative cycle 475 years |
|----------------------|-----------------------------------------------------|-------------------------------------------|-------------------------------------------|
|                      | Natural frequency $f_m$ (Hz)                        | Natural frequency (Hz)                    | Natural frequency $f_m$ (Hz)              |
|                      | 5.86                                                | 5.86                                      | 5.84                                      |
|                      | Damping ratio (%)                                   | Damping ratio (%)                         | Damping ratio (%)                         |
|                      | 6.96                                                | 7.02%                                     | 7.52%                                     |
| Vibration mode       | Displacement in axis X                              | Displacement in axis X                    | Displacement in axis X                    |
|                      | Displacement in axis Y                              | Displacement in axis Y                    | Displacement in axis Y                    |

5. Discussions

Thus, it can be seen that the calculation results of determination of dynamic characteristics (natural frequency) shown in Table 1 and obtained from structural analysis using finite element method conducted by Etabs software is similar to the results shown in Table 2 and obtained from conducting experiments with the same construction structure. Through the calculation results, it was found that with the stiffness of the beam-column connection $k = 8kN.m$, corresponding to that the value of bending moment in end section gains 18% of value of bending moment in mid-span section, the calculated natural frequency will be consistent with the natural frequency obtained from the experimental model. This result reflects the work of the member connection is semi-rigid, with specific stiffness as mentioned above, consistent with previous studies.

6. Conclusions

Through structural analysis using finite element method, in particular by Etabs software and experimental results through specific natural frequency of small-scale specimen of 12-storey semi-precast high-rise building, the article proposed method and procedures for calculating the value of member connection stiffness. From the determination of member connection stiffness the design and calculation process is more reasonable. The obtained acceleration and displacement values of the experimental model are quite consistent with the theoretical calculation results by finite element method. This proves that the proposed method for determination of values of member connection stiffness is relatively appropriate.

References

[1] Hoang Manh. Doctoral thesis: Experimental study of the overall model of semi-precast high-rise building subjected to earthquake load in Vietnam, IBST, 2018.
[2] TCVN 9386: 2012 Design of earthquake-resistant structures, 2012.
[3] Nguyen Trung Kien. Study on a number of dynamic parameters of semi-precast high-rise building, 2018.
[4] Nguyen Vo Thong, Do Tien Thinh, Nguyen Trung Kien. Establishment of artificial accelerogram for shaking table test. The international conference on advances in computational mechanic, ACOME 2017, 2017.
[5] Nguyen Vo Thong, Vo Van Thao, Do Tien Thinh, Ngo Manh Toan, Nguyen Trung Kien and Hoang Manh. Establishment of similitude standards in the problem of modelling the structure of dynamic impact. Scientific Conference celebrates 50 years of establishment of IBST, 2013