

Parameterized FEA Model Generation and Simulation of Underground Structures Based on BIM: A Case Study

Sheng-Yu Song¹, Jia-Rui Lin², Yu-Cheng Zhou³, Wen-Qi Ding⁴

¹Graduate Student, Department of Civil Engineering, Tsinghua University, Beijing 100084, China; e-mail: scy20@mails.tsinghua.edu.cn

²Assistant Professor, Department of Civil Engineering, Tsinghua University, Beijing 100084, China; e-mail: lin611@tsinghua.edu.cn, **corresponding author**

³Graduate Student, Department of Civil Engineering, Tsinghua University, Beijing 100084, China; e-mail: zhouyc19@mails.tsinghua.edu.cn

⁴Professor, Department of Geotechnical Engineering, College of Civil Engineering, Tongji University, Shanghai 200092, China; e-mail: dingwq@tongji.edu.cn

ABSTRACT

Building information modeling (BIM) and finite element analysis (FEA) are widely used technologies for underground structure design; however, they failed to coordinate work well. Currently, existing BIM-to-FEA model conversion methods have a coarse conversion effect and a narrow scope of application. They do not consider the variational working conditions in the construction of underground structures, and thus can hardly be used in the simulations. To address this problem, this research proposes a semi-automatic BIM-to-FEA conversion method for construction simulation of underground structures based on Revit Dynamo and OpenSees software. The method is realized in three steps. Firstly, typical subway stations are selected as the research objects, and the parameterized BIM models are established. Secondly, based on Dynamo visual programming and tool command language (TCL) scripts, a BIM-to-FEA semi-automatic conversion method is formed to carry out FEA calculation. Finally, the method is extended for construction simulation. Based on the BIM model of the station's enveloping structure, an FEA model of diaphragm wall with multiple working conditions is automatically generated, and the structural simulation of the open-cut excavation process is conducted. This method provides an automatic BIM-to-FEA conversion way that can efficiently generate FEA construction simulation models with various working conditions, and realizes a time-saving construction simulation framework based on BIM technology.

INTRODUCTION

As is widely used in civil engineering design and construction, Building Information Modeling (BIM) provides a multi-disciplinary three-dimensional collaborative design and information management platform, greatly improves the efficiency of design and management (Lin & Zhou 2020). Finite element analysis (FEA) is a numerical calculation method widely used in the performance analysis of engineering structures, especially underground buildings. The integration of BIM and FEA is expected to realize the automation of underground structure modeling and analysis, which can improve the design efficiency with promising application prospects. Currently, commonly used BIM software platforms such as Revit integrate no FEA module suitable for underground structures. Therefore, BIM models of underground structures cannot be directly used for FEA structural analysis. Instead, they highly rely on methods such as

BIM-to-FEA model conversion, which means the BIM model is converted to an FEA model, and then the FEA process of the model is conducted in FEA software.

However, the existing BIM-to-FEA model conversion methods have disadvantages such as high development complexity, low conversion accuracy, low degree of automation, and they are difficult to use in underground structures. For example, Song et al. (2016) use C# for secondary development based on Revit API, and convert the Revit model into ANSYS analysis model. Li (2012) used a similar method to realize the conversion of the Revit model to SAP2000 model. These existing model conversion methods are mostly based on Revit API combined with the commercial FEA software such as ANSYS and Abaqus, which are close-source and inconvenient for secondary development. Additionally, they implement the FEA of statics models, and rarely consider the dynamic construction process, which is important in analysis of underground structure.

OpenSees (Open System for Earthquake Engineering Simulation) and Dynamo have brought new possibilities in this field. OpenSees is a new generation of structural analysis software, with advanced tool command language (TCL) modeling functions. As an open-source software, it is convenient for secondary development and collaborative work (Lu et al. 2015; Mazzoni et al., 2010). Its parallel algorithms are applicable for the calculation of large-scale models (J. Lin 2004; McKenna & Fenves 2008; Usmani et al. 2012). The Dynamo tool of the Revit platform is a visual programming component, which is conducive for fast programming, and is often used for quick parametric design and model information extraction. For instance, Nezamaldin (2019) used Dynamo for parametric design, conversion, and analysis of CAD model to BIM model. One recent report (Lin et al. 2020) combined Dynamo tool with OpenSees and realized a preliminary automatic assessment of steel frame structure reliability.

Therefore, aiming at the dilemma of BIM-to-FEA model conversion of underground structures, this research selects OpenSees and Dynamo for secondary development, establishes parametric BIM models of typical subway stations as the research objects for BIM-to-FEA conversion. Then, conversion and structural calculation examples of several subway station BIM models are implemented. The contributions of this research are: 1) breaking the limit of traditional conversion methods, and proposing a semi-automatic BIM-to-FEA conversion method for underground structures by visual programs; 2) extending the BIM-to-FEA conversion method to dynamic construction simulation based on the excavation process of the subway station; 3) integrating the BIM-to-FEA conversion and simulation process into a BIM based structure simulation framework.

SEMI-AUTOMATIC BIM-TO-FEA CONVERSION METHOD FOR UNDERGROUND MODEL

First of all, this research first established Revit models based on typical subway station design cases as the data model for BIM-to-FEA transformation. Then Dynamo visual programming tools and TCL programming are adopted for secondary development to form a BIM-to-FEA semi-automatic conversion method for underground structures. Finally, the FEA simulation case of the main structure is realized, and the multi-stage excavation displacement is simulated in the construction case. In general, the whole framework is divided into the following steps: 1) Starting from the typical subway station structure, BIM models of the main and the envelope structure are established by parametric modeling method based on Dynamo. 2) Dynamo programs are written to extract geometric parameters of BIM models. The pre-processing

parameters are converted into the corresponding FEA pre-processing TCL files. By pre-processing files, Dynamo programs have formed a semi-automatic BIM-to-FEA conversion method. 3) Aiming at the example application of the semi-automation BIM-to-FEA conversion method for underground structure, the model conversion and FEA simulation of the main structure of the two types of subway stations and the envelope structure cases are carried out as the case study. Combining the above three steps, a simulation and visualization method framework of underground structure, which integrates BIM modeling, BIM-to-FEA model conversion, and FEA structure calculation and analysis, can be obtained as shown in Figure 1.

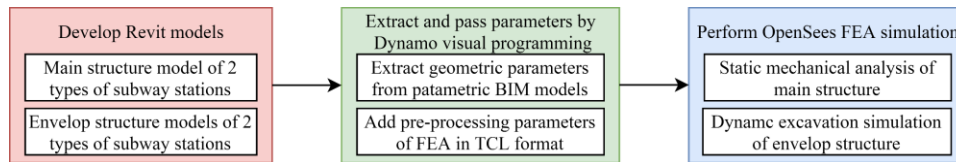


Figure 1. An overview of the research methodology.

Establishment of parametric BIM model cases for subway stations. Parametric modeling is a method of real-time generation and adjustment of BIM models using parameters and parameter equations. With the help of Revit Dynamo, the parametric modeling process can be implemented in the form of a visual script. In detail, for beams or unidirectional slab elements, lines are used to indicate their position, which is abstracted as the relationship between different points; for column or wall elements, points and elevations are used to represent their location; For all components with a specific shape of cross-section, the dimensional parameters of the section are used to indicate the cross-section. In addition, considering the requirements of BIM-to-FEA model conversion and FEA calculation, elastic modulus and gravity parameters can be defined to generate different materials, and the load parameters can be determined to apply the load.

Taking into account the model conversion requirements of OpenSees, the establishment of parametric Revit models can be simplified appropriately. Ignoring irrelevant details, the Dynamo parametric modeling of the subway station cases can be divided into several parts: grids, levels, walls, columns, and slabs. When creating Dynamo script, grids are created through "Grid.ByStartPointEndPoint" node, levels are created through "Level.ByElevation" node, columns, and walls are generated through "FamilyInstance.ByPointAndLevel" node, and slabs are generated through "StructuralFraming.BeamByCurve" node. Through the above steps, parameterized BIM models of the main structure are established. The two types of subway station cases are shown in Figures 2(a) and Figure 2(b). Correspondingly, a parametric model of the envelope structure is produced, as shown in Figure 2(c). Each model represents a kind of structure, and the parameters can be modified to adapt to different structure candidates. In this research, the three types of BIM models are considered to be transferred into FEA models.

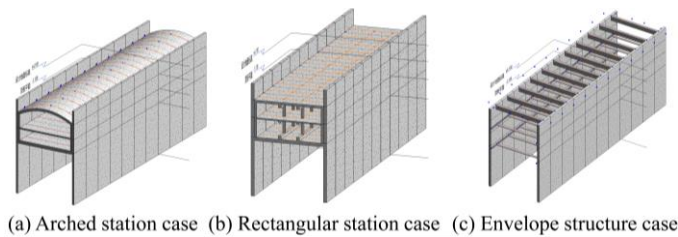


Figure 2. The parameterized BIM model of the subway station cases

Semi-automatic conversion of parametric BIM model to OpenSees model. In general, the workflow of the semi-automatic model conversion is as follows. Firstly, based on the parametric BIM model of the station cases, the load information and pre-processing information that are not available in the BIM model are independently summarized in the form of a TCL parameterized pre-processing script, which is necessary to build the parametric two-dimensional elastic frame FEA models of OpenSees. The pre-processing information is also formed in the form of variables through TCL parametric programming. Secondly, a Dynamo conversion program is developed through IronPython scripts, in which the parameters of BIM models of the station cases are imported into a Dynamo parameter transfer node based on the Python Script node. The python program in the node combines BIM model information and TCL pre-processing information into a complete FEA model file in TCL format, completing the semi-automatic conversion of the Revit model to OpenSees FEA model.

The main difficulty of this method includes the implementation of parameter passing by the Python Script node and the pre-processing by parametric TCL programming. The idea of the conversion Python Script node is as follows: 1) extracting the model node coordinates, calculating the spacing of center lines of columns and walls, elevation of center lines of slabs, and thickness of slabs and walls, etc.; 2) inputting them into the Python Node as an “IN” list; 3) using numerical calculation operation, string operations and file operation of IronPython, to convert coordinates and assign the values of related parameters to the variables of the TCL file to achieve the parameter transfer.

The idea of pre-processing TCL program is as follows: realizing variable parametric pre-processing calculation directly through OpenSees TCL language script. After the Python Script node generates a parameter assignment command and completes the parameter transfer, the FEA modeling parameters can be directly pre-processed through TCL commands to obtain commands for unit generation, constraint generation, and load application to form an FEA model.

CASE STUDY

Basic information of the research cases. As a typical underground engineering project, subway stations are widely adopted in China. At present, the structure of subway stations is mainly rectangular frame structure, and some subway stations also adopt arch frame structure. Based on the site situation, the column-free and large-span design scheme adopted by the Shanghai Wuzhong Road Station was selected as prototype, and a parametric Revit model of the arched station was established. Then based on the same geometric size information as the Wuzhong Road Station, a design scheme of a two-story, two-column and three-span subway station is also selected as another prototype for establishing a parametric Revit model of the rectangular station.

In terms of engineering features of the arched station case, based on the scheme of Wuzhong Road station, the internal net width is 20m, and the thickness of the top slab, middle slab, and bottom slab are respectively 1000mm, 700mm, and 1300mm. The thickness of the inner wall and underground continuous wall are 600mm and 800mm respectively. The elevation of the design ground, the top of the roof vault, the junction of the top slab and the inner walls, the top of the middle slab, and the top of the bottom are respectively +4.100m, +2.000m, -1.139m, -6.793m, -13.063m. For the convenience of comparison, the span, elevation, and wall thickness of the rectangular station case are the same as those of the arched station case, and the thickness of the slab can be appropriately lowered. The thickness of the top, middle, and bottom slabs of the rectangular station case is 800mm, 400mm, and 1100mm, respectively.

When simulating the construction process, the elevations of the top and bottom of the underground continuous wall are +4.100m and -32.000m. Only three supports are considered, the first one is 800mm×1200mm rectangular-section reinforced concrete support. The second and third support is steel pipe support with 800mm outer diameter and 16mm thickness. The elevations of centerlines of the three supports are +3.200m, -4.900m, and -14.200m respectively.

FEA simulation of the main structure of subway station cases. Firstly, take the parametric BIM model of rectangular station case as an example to verify the BIM-to-FEA model semi-automatic method. The following steps are needed: parameter transfer, parametric pre-processing programming, load simplification, model simplification, model generation, and calculation.

For parameter transfer, the following steps are adopted: 1) compiling the parameter transfer script in the Python Script node, and inputting the spacing and size parameters of the slabs, walls, and columns in the way of Dynamo connection as the “IN” list of Dynamo Python Node, and 2) recording the input model parameters in the form of variables, and setting the auxiliary parameters that need to be used in pre-processing such as the longitudinal spacing of the center column, the elastic modulus, and the stiffness of the soil spring as variables, and 3) writing the value into the TCL variable assignment command to get the TCL parameter definition file, as shown in Figure 3. In the TCL pre-processing script, loop and other procedures are used to compile parametric calculation programs to perform the pre-processing of the model, as is shown in Figure 3, directly generate FEA node and unit commands, etc., and form the pre-processing files. In terms of load value, the various loads on the component are appropriately simplified, and the value is set as a variable parameter, which is directly defined by the TCL command. Specifically, the top slab load is simplified as the combination of earth pressure, self-weight, and ground overload; the middle slab load is simplified as a combination of live load, equipment load, and self-weight; bottom plate load is simplified as live load, equipment load, self-weight, and earth pressure; The earth pressure on the sidewall is a trapezoidal load.

As regards to model simplification, the framework is simplified to elasticBeamColumn frame unit, and the soil spring is simplified to truss unit. Considering the relationship between the bottom slab, the underground continuous wall, and the soil, soil spring nodes are added to divide the continuous wall and bottom slab into more sections of frame elements. By the Dynamo script, the obtained TCL file is imported into OpenSees for calculation. OSLite software is used for the visualization of the calculation results. The FEA model analysis results of the two types of subway station cases can be obtained as shown in Figure 4.

```
#####
for (set i 1) {$i<=2} {incr i} {
  element elasticBeamColumn [expr 6+6*($i-1)+1] [expr 6*($i-1)+1] [expr 6*$i+1] Sadi Ses1 Sizdi 1
  element elasticBeamColumn [expr 6+6*($i-1)+2] [expr 6*($i-1)+2] [expr 6*$i+2] Saside Ses1 Sizside 1
  element elasticBeamColumn [expr 6+6*($i-1)+3] [expr 6*($i-1)+3] [expr 6*$i+3] Sacolumn Ses1 Sizcolumn 1
  element elasticBeamColumn [expr 6+6*($i-1)+4] [expr 6*($i-1)+4] [expr 6*$i+4] Sacolumn Ses1 Sizcolumn 1
  element elasticBeamColumn [expr 6+6*($i-1)+5] [expr 6*($i-1)+5] [expr 6*$i+5] Saside Ses1 Sizside 1
  element elasticBeamColumn [expr 6+6*($i-1)+6] [expr 6*($i-1)+6] [expr 6*$i+6] Sadi Ses1 Sizdi 1
}
#####
for (set i 3) {$i<=3} {incr i} {
  element elasticBeamColumn [expr 6+6*($i-1)+1] [expr 6*($i-1)+1] [expr 6*$i+1] Sadi Ses1 Sizdi 1
  element truss [expr 6+6*($i-1)+2] [expr 6*($i-1)+2] [expr 6*$i+2] 1.0 1
  element truss [expr 6+6*($i-1)+3] [expr 6*($i-1)+3] [expr 6*$i+3] 1.0 1
  element truss [expr 6+6*($i-1)+4] [expr 6*($i-1)+4] [expr 6*$i+4] 1.0 1
  element truss [expr 6+6*($i-1)+5] [expr 6*($i-1)+5] [expr 6*$i+5] 1.0 1
  element elasticBeamColumn [expr 6+6*($i-1)+6] [expr 6*($i-1)+6] [expr 6*$i+6] Sadi Ses1 Sizdi 1
}
}
```

Figure 3. An example of the scripts of the TCL file.

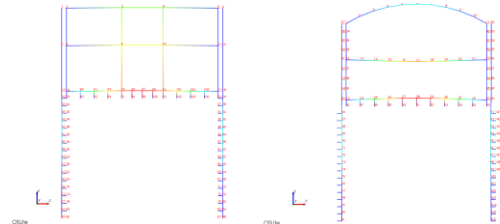


Figure 4. FEA results of the rectangular (left) and arched (right) station cases (Deformation magnification factor:10).

Excavation simulation of envelope structure of subway station cases. The open-cut method is the most widely used in the construction methods of subway stations, so its excavation process is simulated by the semi-automatic BIM-to-FEA conversion method based on the parameterized

BIM model of the envelope structure of the station cases. The parameter transfer and pre-processing of construction simulation are the leading process to this example.

The main difficulty is how to consider the excavation process of the open-cut construction method, and prepare the pre-processing file of the parameterized TCL model that meets the requirements of construction simulation. Concretely, the foundation pit is excavated in three steps: 1) excavating to +1.900m, pouring the first reinforced concrete support; 2) excavating to -5.400m, erecting the middle steel support; 3) excavating to -14.7m, erecting the bottom Steel support, completing the excavation process. The calculation of construction simulation contains the working conditions corresponding to the above steps.

In terms of model simplification, the underground continuous wall is selected for conversion per linear meter, and the axial deformation is ignored. The continuous wall is simplified to a two-dimensional load structure model. Correspondingly, the soil spring element is simplified to the truss element, and the underground continuous wall is simplified to the elasticBeamColumn element, of which the element length is taken as 0.1m. For load simplification, considering multiple soil layers as an equivalent soil layer, the active earth pressure on the left side of the continuous wall is simplified and applied with triangular loads. The load value is defined as a parameter in the pre-processing file, which is convenient to adjust.

In order to simulate the variations in each stage of the excavation construction, the load application is divided into three working conditions according to the excavation situation. Each working condition includes two steps of excavation and adding support, so the construction constraints are adjusted in 6 steps. After running the Dynamo program, importing the generated TCL file into OpenSees for FEA calculation, and using OSLite software to realize the visual output of the calculation results, the deformation simulation results at each stage of the excavation are figured out, as shown in Figure 5 (left). In the same way, we can freely adjust the model to suit different situations. For example, if both the first support and the second support fail, the simulation results of the three stages of structural deformation during excavation are shown in Figure 5 (middle). If only the second support fails, the simulation result of the three stages of structural deformation during excavation are also shown in Figure 5 (right).

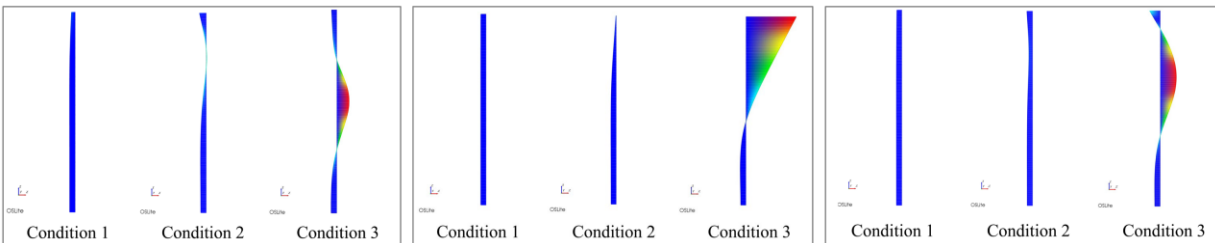


Figure 5. The deformation at each stage of the excavation while no support fail (left), the first and second support fail (middle), and only the second support fails (right) (Deformation magnification factor: 30,5,5).

Generation and simulation of different design candidates. In this research, BIM model establishment, BIM-to-FEA conversion, and FEA calculation are all based on parametric methods. They can be easily integrated to obtain a parametric model establishment, conversion, and structural calculation framework. The framework can solve series of problems in batches of which the working conditions are similar to the case study. It is also convenient to compare and

optimize different schemes of the same structure type by BIM model and FEA simulation at the initial stage of designation of structural schemes.

For example, the Dynamo modeling script and conversion nodes of the arched station are integrated, and then the modeling parameters are adjusted to generate different results with different combinations of parameters in batches, as shown in Figure 6. In this way, design parameters with satisfactory structural performance can be directly obtained.

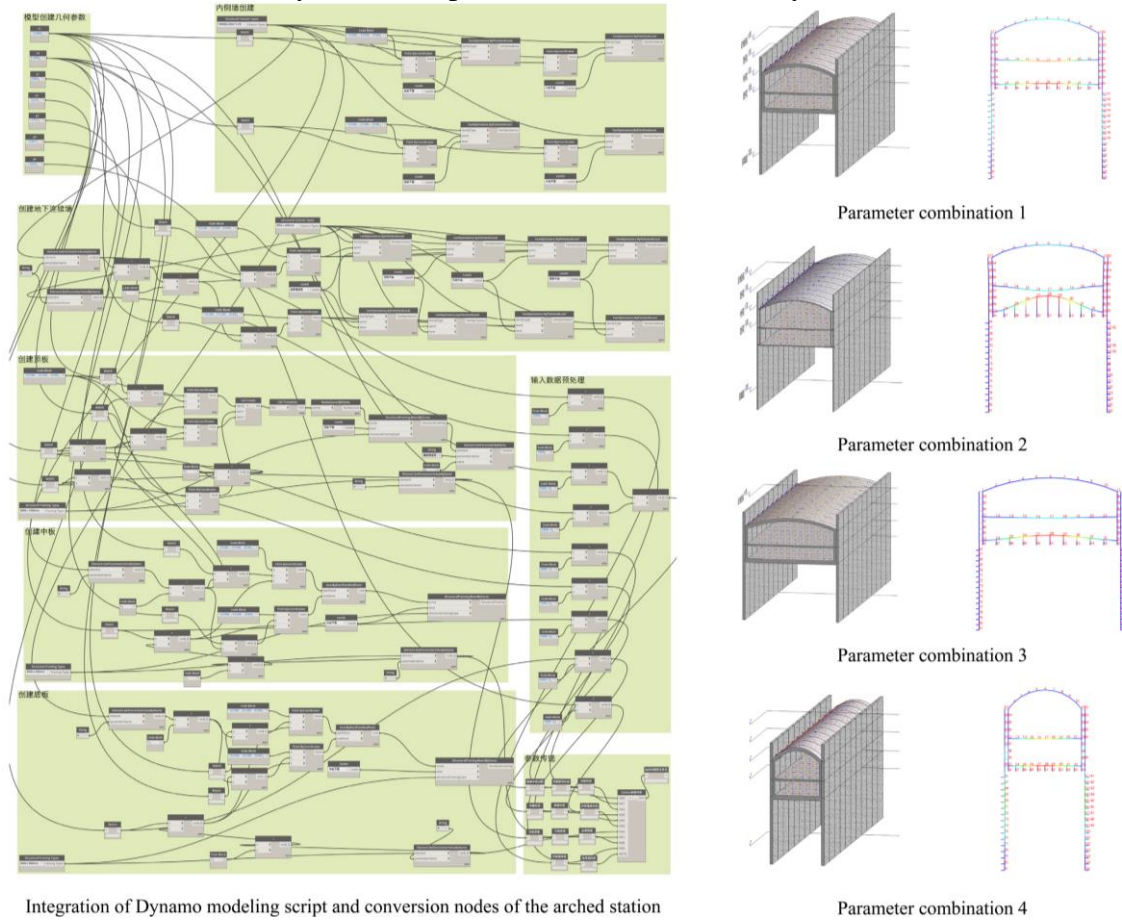


Figure 6. An example of the simulation framework based on the arched station case.

DISCUSSION AND CONCLUSION

In summary, the contributions of this study are: 1) Through Dynamo visual programming and secondary development of Python and TCL, limitations of traditional BIM-to-FEA conversion methods are broken, and an effective semi-automatic BIM-to-FEA conversion method for underground structures is proposed, which is simple to develop and has a high degree of automation. The method is implemented by examples of the main structures of two types of mainstream subway stations. 2) The BIM-to-FEA conversion method is extended to multistage simulation of the excavation process based on the envelope structure model, by which enhance the practicality of BIM-to-FEA conversion. 3) Research results are integrated into a complete BIM-based structural simulation framework, by which BIM models of similar structures can be generated and simulated in batches to determine an appropriate structure scheme.

Some limitations are identified in the proposed method which need to be improved in future researches. 1) The model has a high degree of geometric simplification, and only realizes the FEA calculation of the two-dimensional frame, which cannot be used for simulation of complex three-dimensional structures. 2) The conversion method is realized in the form of scripts, rather than a complete software or a plug-in of Revit, which is inconvenient for promotion. Thus, it can only be used in subway stations rather than other types of underground structures. 3) As the load-structure model is used in FEA, the stratigraphic details are discounted. Thus, the simulation of case study is not fine enough for real engineering application.

The method and the corresponding software platform have great potential for secondary development. Subsequent research is expected to form a graphical plug-in, achieve semi-automatic BIM-to-FEA conversion of 3D structure and a more automatic conversion process, and promote the integration of BIM and FEA software.

ACKNOWLEDGEMENT

The authors are grateful for the financial support received from the National Natural Science Foundation of China (No. 51908323, No. 72091512), and the Tsinghua University-Glodon Joint Research Center for Building Information Model (RCBIM).

REFERENCES

- Li, Y. (2012). *Research of the Structure Model of Architecture Based on BIM* [PhD Thesis, Xi'an University of Architecture and Technology].
- Lin, J. (2004). *Application and preliminary development of nonlinear analysis for reinforced concrete structures in object-oriented open program OpenSees* [PhD Thesis, Chongqing University].
- Lin, J.-R., Xiao, J., & Zhang, Y. (2020). *A Framework to Automate Reliability-based Structural Optimization based on Visual Programming and OpenSees*. 8th International Conference on Construction Engineering and Project Management, Hong Kong, China.
- Lin, J.-R., & Zhou, Y.-C. (2020). Semantic classification and hash code accelerated detection of design changes in BIM models. *Automation in Construction*, 115, 103212.
- Lu, X., Xie, L., Guan, H., Huang, Y., & Lu, X. (2015). A shear wall element for nonlinear seismic analysis of super-tall buildings using OpenSees. *Finite Elements in Analysis and Design*, 98, 14–25.
- Mazzoni, S., McKenna, F., & Fenves, G. L. (2010). The OpenSees getting started manual. *Online Manual*. <http://opensees.berkeley.edu>
- McKenna, F., & Fenves, G. L. (2008). *Using the OpenSees interpreter on parallel computers*. Network for Earthquake Engineering Simulations, Berkeley, CA.
- Nezamaldin, D. (2019). *Parametric design with Visual programming in Dynamo with Revit: The conversion from CAD models to BIM and the design of analytical applications* [Master Thesis, KTH Royal Institute of Technology].
- Song, J., Zhang, Y., Wang, M., Zhang, J., & Fan, J. (2016). Research of Model Exchange Interface Between Revit and ANSYS. *Journal of Civil Engineering and Management*, 1, 79–84.
- Usmani, A., Zhang, J., Jiang, J., Jiang, Y., & May, I. (2012). Using OpenSees for structures in fire. *Journal of Structural Fire Engineering*, 3, 57-70.