Development and Application of a Revit-ANSYS Model Transformation Interface

H Zhou¹,², X He¹, X L Leng³

¹School of Civil Engineering and Architecture, Wuhan Polytechnic University, Wuhan Changqing Park, Southern Rd. 68, Wuhan 430023, China.

²Institute of Rock and Soil Mechanics, Chinese Academy of Sciences, Wuhan 430071, China.

E-mail: 27491739@qq.com

Abstract. To address the difficulties of modeling complex building structures in ANSYS, the Revit-ANSYS model conversion interface was developed. As powerful 3-d modeling software, Revit can establish refined and complex models, which has advantages that ANSYS does not have. It is particularly important to automatically transform BIM complex model into structural analysis model for accurate structural calculation and analysis. Based on the Revit application programming interface (API), the geometry size, coordinates, material attributions, boundary conditions and load information of the existing structural models in Revit, were extracted using C# on the Visual Studio 2017 platform for secondary development. In the interface program, all the required information was processed, material attributes were matched, structural connection was conducted using Boolean operation, and mesh division was prepared in advance, with the purpose of automatic modeling, Boolean operation, and automatic mesh division before finite element analysis using ANSYS. This generated an APDL file that can be recognized by ANSYS for this purpose. The rapid transformation of Revit-ANSYS model was thus realized. The conversion interface sets up the data communication between Revit and ANSYS and is applicable to both superstructure and underground structure. The goal of importing complex structure models of Revit into ANSYS for accurate structural calculation and analysis was achieved. This improved the efficiency of modeling and calculation of complex structures. The resulting Revit-ANSYS model conversion interface can integrate the two different software packages to create BIM and FEM models. Finally, it was applied to the 3-d BIM modeling and structural analysis of a construction project. By using the interface program, the construction was dynamically simulated by real-time updating of BIM model and the mechanical calculation was conducted. The correctness and practicability of the model conversion interface were verified.

1. Introduction

At present, businesses and BIM-users involved in geotechnical engineering are developing new BIM applications, which should not only allow three-dimensional visualization of the model, but should play a more critical role in the engineering. BIM model can only achieve the combination of “viewing” and a project budget, but it is far from perfect in terms of structural calculation and risk prediction. If BIM can be effectively combined with finite element calculation, the specific construction model in BIM software can be quickly converted into finite element calculation models through the finite element software interface, which will have a wide range of applications in engineering.
The concept of BIM was first proposed by Dr Chuck Eastman of Architecture and Computer Science at Georgia Tech. College in the United States, in 1975 [1-2]: “The building information model integrates all geometric model information, functional requirements and component performance to cover all information of the life cycle of the building project, including process information, such as construction progress, maintenance management process, etc., in a single building model.” The International Standards Organization Facilities Information Council defines BIM as: “Building information model is a computable or performability representation of the physical and functional characteristics of facilities and their related project life cycle information under open industrial standards. All information related to the building information model is organized in a series of application program, allowing operations such as acquisition and modification” [3]. The core of BIM technology is building information, and establishing model conveys building information, which shows the important position of modeling software in BIM technology, making Revit, which belongs to Autodesk with AutoCAD, first choice for more engineers. Considering the weakness of Revit in structural analysis, researchers and software development engineers use intermediate software or IFC format files as a conversion platform to develop numerous interfaces, such as Revit and Robot, ETABS, STAAD, YJK, or PKPM, but in terms of its function, these interface packages are mainly used in the design of construction drawings of engineering structures, but fail to solve problems related to their structural analysis [4]. To overcome the deficiencies of Revit in structural analysis, Song Jie et al. [5] studied the conversion interface between Revit and the ANSYS structural model using Revit application programming interface (API) technology, but problems persist in the applicability of its interface program to Revit model and the treatment of structural joints. Chen Shu et al. [6] studied the application of Revit and ANSYS structure calculation, but the degree of automation of its interface program is insufficient: it is also necessary to divide the region manually and then mesh its structure. Zhang Xiaoheng et al. [7] researched the BIM modeling interface of a bridge superstructure based on MIDAS-Revit, and the interface aims to reconstruct a BIM model by using existing MIDAS model data, but cannot use BIM model data for modeling in computing software. Zhao Xiaoyu et al. [8] studied the automatic interface of model conversion between Revit and ANSYS, but its automatic interface needed data provided by the modeling process in Revit and could not be directly converted using the existing model.

Generally, a good data conversion should follow three rules [9]:
1. The openness and transparency of the conversion process, including the selectivity of the conversion content and the record of errors, warnings, and other information in the conversion process should be clear.
2. Availability of the transformation results. The transformed model should not only be consistent with the original model in terms of geometric size, but also the intrinsic properties of components should meet the requirements of subsequent analysis.
3. The stability of the conversion standard (interface) should not be interrupted due to data loss or other reasons in the conversion process.

Based on the above criteria, we use Revit API to extract and process the data of BIM model, and realize re-modeling in ANSYS. The ability of finite element analysis and mesh division ability of ANSYS software are used to conduct finite element analysis of complex models, which provides a reference for design and construction and lays a foundation for the future establishment of an integrated platform of finite element and BIM and the formation of closed-loop model data.

2. The research of model transformation

2.1. Model transformation idea

The model data required by the Revit model transformation interface include model position information, geometric dimensions, material properties, boundary conditions, and load regime. Among them, location information, geometric size, boundary conditions, and loading conditions are extracted and stored according to different components, and material attributes are extracted and stored according to different families. The model data are extracted through Revit API, processed through an
interface program, and written into a text file in the form of command stream according to ANSYS parametric design language APDL. Finally, the required analysis model can be constructed by reading the file in ANSYS.

2.2. Development tools
Revit is not only a powerful parametric 3-d modeling software tool, but also has a rich API that makes it easy for developers to integrate extended applications into the Revit platform. The Revit API is contained in two interface component files (Revit API.dll and RevitApiui.dll) and is available on the RevitApiui.dll.NET environment for development, compatible .NET languages include C#, C++, and so on. Here, the C# language will be used for interface development, and external command mode of secondary development will be used to facilitate developers to make changes [10-11].

2.3. Specific process of interface development
The interface development process is shown in Figure 1. First, a filter program is established to filter the model according to the name of the family class (Element.Category.Name), and then the scanned form of selected elements is determined to obtain its position origin, positioning line, and geometric size information. The location line calculates its terminal coordinates according to its starting point, direction and length, and continues the preliminary processing and storage of these information based on the reconstructed target in ANSYS. For the column and beam in the structure part, the geometric dimension information is mainly used to obtain the width b and length h in the section. When acquiring the information of these structures, the boundary conditions, and load information on their structural planes or points are obtained according to these structures and stored and output. Finally, the information obtained above is output in the APDL file format needed for input to ANSYS. Since the modeling method of elongated body is advantageous when modeling the column with fixed section and generatrix, this method is applied to the modeling of beams, floor-slabs, and columns in ANSYS. After the transformation of the model is completed, the material attributes are extracted and output. Meanwhile, in the process of output, it is also necessary to control the number of points, edges, surfaces, and bodies according to the modeling methods and the types of models.

Figure 1. Development interface flow chart.
2.4. The key technology
After the complete model transformation is exported to ANSYS in the early stage of development, the grid is often unable to be divided normally during Boolean operation, which causes the failure of finite element calculation. Through the analysis of this problem, we found that BIM model itself is a view-based model, which is often accompanied by the processing of intersecting parts at the structural junctions. When processing these intersections, Revit will cause a single structure to suffer omission of a certain part, and the missing part will not be considered during the transformation process. The model converted to ANSYS will have these intersections, which need to be handled by Boolean operation; however, the intersections in the BIM model may be small in size, and some of the intersections are attributed to data transformation needing to meet the necessary precision and value requirements. These intersecting parts will report errors due to their admissible values when Boolean operation is conducted in ANSYS. Even if the admissible values are adjusted to make the Boolean operation successful, the size of the structure of the intersecting parts cut thereby is far smaller than that of the model body, which adversely affects the mesh division and subsequent finite element analysis.
Aiming at the above problems and aiming to decrease the number of Boolean operations in ANSYS, we conduct more detailed processing of the model data in the interface program, so that the output beams, plates, columns, and other structures are all complete and can be glued together into a complete structure, obviating the need for Boolean operation. The steps in its implementation are described below.

2.4.1. Beam set-up in Revit
The model is adjusted in Revit. The intersection mode of the connection is set according to the priority of column, beam and plate, and the start-point and end-point of the beam are set at the center of the column section (Figure 2). The floor is also set according to the contour line of column and beam (Figure 3).

![Figure 2. Beam start-point and end-point settings](image1)
![Figure 3. Board profile setting.](image2)

2.4.2. Beam conversion technology
For beam spanning multiple columns, they will be divided in the interface program respectively, according to its different segmentation situations. To determine the beginning and end of the beam, the interface program will obtain the whole of the beam-positioning line in the direction of the vector, then obtain the beam as an entity on the side of the structure and the structure of these entities, and this
is screened with each beam-positioning line parallel to the directed vector of the edge. Finally, the solid structure is output according to the positioning of the start and end-points of these edges, ensuring that the start and end of each beam in ANSYS are connected to the column contour line (Figures 4 and 5).

Figure 4. Beam Revit model.  

Figure 5. Beam ANSYS model.

2.4.3. Floor conversion technology
For floor slabs, boundaries will be set according to the column and beam contours in Revit to ensure that there is no section of the floor that intersects the columns and beams. The combination of different beams, plates, and columns will produce different types of sections of plates, according to which, screening conditions are set in the interface program. According to the different points on different floor planes, specific output formats are selected to ensure the integrity of the board after output to ANSYS (Figure 6).

Figure 6. Floor sections.

2.4.4. Boolean operation

In the APDL format file, the last part of the processor – Boolean operation and mesh division – the interface program conducts Boolean operation to bond each structure together firstly, and then goes through all the structures to divide the mesh, ensuring continuity of the mesh between each structure.

While outputting the structure, the boundary conditions and load information on each structure are also determined synchronously. The boundary conditions and load information are matched according to the number of the output structure and applied to the corresponding structure one-by-one when output to the APDL format file, completing the process of constraint and loading.

When all the structure position information, geometric information, material attribute information, boundary conditions, and loading information are acquired, the output link is entered. The APDL format file of ANSYS can cover the whole process of ANSYS from modeling to calculation (Figure 7), including the following: data-cleaning before passing calculation and analysis information, geometric parameters and material properties, geometrical modeling processes, giving material properties and the structural unit type, which, through Boolean operation will allow all structure and bonding processes
in meshing, after processing into the solver before defining the type of analysis, constraints, and loading process, solution and finite element analyses are completed.

3. Analysis of examples
To verify the effectiveness of the above interface program in practical engineering application, we combined drawings in Revit from an engineering project for analysis: a BIM model of the project is established through Revit (Figure 8) to analyze the stress of the structure under the constraints of the foundation bottom and the dead-weight load. The structural part of the model can be displayed through the adjustment of the visibility graph (Figure 9).

The interface program is loaded by running the.dll application extension, which is then run in Revit to select the corresponding structural components such as beams, slabs, columns, and foundations according to the program’s filter function. As the interface program runs, it generates an APDL file at the specified path (Figure 10). Finally, the APDL file is run in ANSYS. In this process, ANSYS will follow the instructions of the command stream to complete the material attribute and geometric parameter definition, structure modeling, constraint and load application, then finish the calculation and other post-processing steps.
After ANSYS completes the aforementioned operations, the total displacement and stress intensity nephograms of the model are as shown in Figures 12 and 13, respectively.

4. Conclusion

Based on a BIM model and finite element analysis, an interface program was written through secondary development to achieve information exchange between Revit and ANSYS software, establish the data flow between a BIM model and a finite element model, and realize the real-time conversion of model information between BIM software and finite element software. The key conclusions are as follows:

4.1 The structural model components were selected in Revit, and the model information of structural components, such as position information, geometric size, boundary conditions, load regime, and material attributes, can be automatically obtained according to the data-extraction methods of different components.

4.2 In the interface program, the family class of components and the judgment of the intersection situation were achieved, and the position information and geometric size obtained were automatically processed.

4.3 The obtained BIM model information was output as an APDL format file through the interface program and run in ANSYS, achieving full automation of the finite element analysis of the model.

Acknowledgments
This work was supported by the National Natural Science Foundation of China (No. 52079135), the International Partnership Program of the International Cooperation Bureau of Chinese Academy of Sciences (No. 131551KYSB20180042)

References
[1] HEG Aia 2004 Aec from the ground up: manage aec projects online Cadalyst.
[2] Zhao A 2006 Application of BIM technology in computer aided building design D.Chongqing University.
[3] Zhu L B, Tian Y F 2014 A review of BIM research literature in China's construction industry J. Architectural Design Management.31(02) 33-37.
[4] Wang X X, Huang Y L, Zhao J C and Duan L P 2020. Development and Application of Revit-ABAQUS Model Conversion Interface J. Journal of Shanghai Jiao tong University.54(02) 135-143.
[5] Song J, Zhang Y D, Wang M J, Zhang J H and Fan J Y. 2016. Research on the interface between Revit and ANSYS structural model conversion J. Journal of Civil Engineering and Management (01) 79-84.
[6] Chen S, Cheng Y Z, Sun Y, Liu J J and Zhang M 2019. Application of Ship Lock Main Structure Calculation Based on Revit and ANSYS J. Water Transportation Engineering (06) 140-146.
[7] Zhang X H, Yang J and Wang J L 2020. Research on BIM Modeling Interface of Bridge Superstructure Based on Midas-Revit J. Municipal Technology (04) 70-74.
[8] Zhao X Y, Chen Z H, Yu S L and Bu L 2021. Research on Model Transformation Automation Based on Revit and ANSYS J. Yangtze River Information and Communication 34(01) 119-121.
[9] Yang D H, Su Y and Sun M 2004. Analysis of data exchange in BIM technology structure design based on Revit J. Information Technology in Civil and Architectural Engineering 6(03) 13-18.
[10] Zhong H, Li C, Sun H and Zhang K 2019. Data extraction and application technology for secondary development of BIM model J. Journal of Shenyang Jianzhu University (Natural Science Edition) (03) 560-566.
[11] Li Chang Wang Ziru 2020. Parametric modeling of polyhedrons based on the secondary development of Revit J. Civil Engineering Information Technology (01) 110-116.