A CAD System Embed Finite Element Modeling Method

Chenglong Li, Xuyong Chen, and Zhifeng Xu*

School of Civil Engineering and Architecture, Wuhan Institute of Technology, Wuhan 430073, China
20120102@wit. edu. cn

Abstract: Currently, most finite element analysis applications are relatively weak at modeling, resulting in finite element analyses of complex structural systems being trivial and troublesome. On the other hand, though computer-aided design (CAD) applications are powerful at modeling, models built in CAD applications are usually not applicable for finite element analysis. To this end, this research proposes a CAD system embed finite element modeling method that takes advantage of the interactive environment and useful modeling functions provided by CAD applications. To demonstrate the validity and practicality of the proposed method, this research further developed a Rhinoceros plugin for finite element modeling named DirectMesh and applied it to the finite element analysis of Wuhan Station. The result shows that the proposed method can not only create valid finite element models but also considerably increase the usability and efficiency of finite element modeling, benefiting the intelligent level of mechanical analysis-related jobs in mechanical engineering, civil engineering, and geo-engineering.

keywords: Computer-Aided Design; Finite Element Modeling; Mesh; Software

1 Introduction

With the rapid development of urban transportation construction, my country is working hard to build a number of modern, three-dimensional, large-scale, comprehensive transportation hubs formed by a variety of transportation nodes. As the functional requirements for large comprehensive transportation hubs are getting higher and higher, their structural forms are becoming increasingly complex and diverse. How to accurately and efficiently establish finite element models has become a hot issue in the field of structural design [1]. Zhao Yu et al. quickly established the finite element model of Kunming South Station based on MATLAB's method of quickly establishing an ANSYS model of large railway station structures [2]. Zhu Huiqing et al. analyzed the internal model data conversion logic between SAP2000 and ABAQUS software and used MATLAB software to compile an STA program to complete the automatic conversion of model data [3]. Zhang Yueqiang et al. used the model conversion software MTR-1.0 to quickly import the SAP2000 structural model into ABAQUS for calculation [4]. Zhu Ming and Wang Chunlei provided fast and

W. Guo et al. (eds.), *Proceedings of the 3rd International Conference on Green Building, Civil Engineering and Smart City (GBCESC 2024)*, Advances in Engineering Research 264,
https://doi.org/10.2991/978-94-6463-728-1_14

effective parameterization design functions through Rhino software and the Grasshopper plugin and applied them to the modeling of double-layer mesh shell structures [5]. Hu Yue has developed CAD-based terrain map modeling and quality inspection software [6]. Li Kun applied BIM technology to establish a subway station model [7]. Xu Litong et al. studied a 3D engineering route modeling method based on the Autodesk platform [8]. However, the above modeling methods have low efficiency in utilizing powerful CAD systems for modeling, and the modeling environment is relatively weak. They lack sufficient utilization of the powerful modeling environment and visualization operation functions of CAD software, and the surface mesh processing is not detailed and intelligent enough, making it very cumbersome to model complex structural systems. If the visualization tools provided by CAD systems can be utilized to complete some limited modeling work, it can greatly save time spent on modeling.

Therefore, this article proposes a general method for finite element modeling of complex structural systems based on CAD systems. Its core is an intelligent software called DirectMesh, which can quickly establish models for various commercial finite element analysis programs in the Rhinocero CAD environment. DirectMesh is a self-developed software that fully utilizes the powerful visualization modeling function of CAD systems to establish three-dimensional mesh models based on geometric objects such as points, curves, and surfaces. Ultimately, the software generates various commercial software models through three-dimensional mesh models. The software achieves rapid generation of three-dimensional solid models for points, curves, and surfaces, automatically repairing surface meshes, processing compatibility with multiple surface meshes, high-quality personalized surface mesh re partitioning, and flexible and personalized surface mesh information extraction, greatly improving modeling efficiency and accuracy. In addition, the generated model can be converted between different finite element software, solving the cross platform problem of finite element models.

2 A CAD system embedded finite element modeling method

This finite element modeling method mainly includes two units: independent mesh generation unit and comprehensive mesh generation unit. First, an independent surface mesh representing an object is generated by an independent mesh generation unit. In finite element analysis, the most common situation is a coupled system of multiple engineering objects. For this purpose, a mesh that contains multiple objects and is compatible with each other is required. The system, therefore, can solve the above problems by integrated mesh generation units. Finally, the finite element model generator is used to generate mutually compatible mesh systems to form a finite element model, forming a complete process from mesh generation to finite element modeling (as shown in Fig. 1).

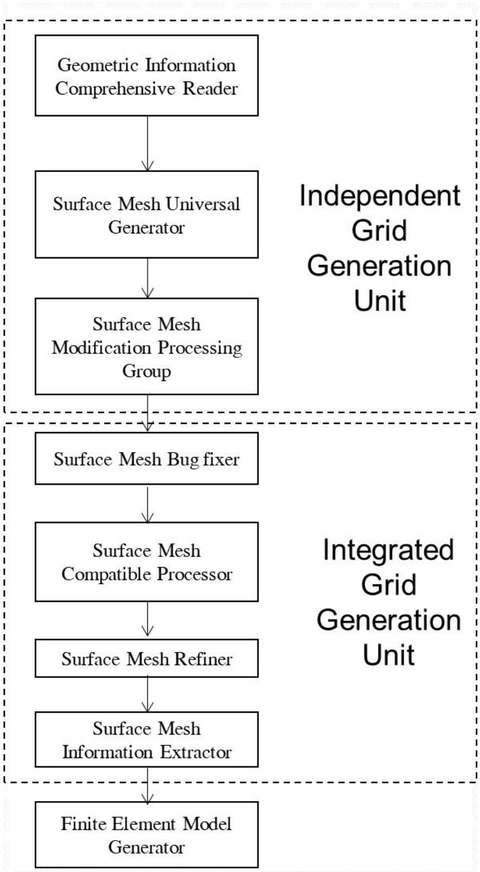


Fig. 1. Process of the proposed finite element modeling method

2.1 Independent mesh generation unit

This unit mainly consists of the first four parts (geometric information comprehensive reader, surface mesh universal generator, surface mesh modification processor and surface mesh error repairer). It is specifically divided into the following four steps to complete:

In the first step, the CAD drawing information is entered through the geometric information integrated reader, including basic axis grids, beams, plates, columns and other plan views, which determines the position information and cross-sectional information of the model.

In the second step, the surface mesh universal generator is used to generate the corresponding mesh through various geometric objects, and the resulting mesh is established and drawn in the corresponding CAD system. The universal surface mesh generation algorithm has the following two modes: (1) Generate meshes for each object individually (2) Use multiple objects to intelligently generate a surface mesh

In the third step, use the surface mesh error repairer to repair the mesh errors formed in the third step to form an error-free independent surface mesh. The repair function mainly includes: grid hole filling module, grid crack repair module, conflict surface re-division module, and protrusion removal module.

2.2 Comprehensive grid generation unit

This unit is mainly composed of a surface mesh compatible processor, a surface mesh fine divider and a surface mesh segmentation extractor. It is specifically divided into the following three steps to complete:

In the first step, the system formed by combining multiple meshes representing different objects formed in the last step of the independent mesh generation unit is processed through the surface mesh compatibility processor to form a complete set of compatible and closed surface meshes.

In the second step, a surface refiner is used to perform high-quality fine meshing on a set of compatible and closed surface meshes formed in the previous step.

The third step is to extract features of the surface mesh formed in the previous step, extract specific parts and name them to facilitate the application of loads and boundary conditions in subsequent finite element analysis. It specifically includes the following five modules: single surface extraction module, multi-surface Extraction module, intersection extraction module, boundary layer extraction module, and entity internal part extraction module.

2.3 Finite element model generator

Finally, put the closed surface meshes formed in the previous step into the finite element model generator to generate model files for various finite element software, such as ANSYS, ABAQUS, FLAC3D, MIDAS, etc.

3 Introduction of DirectMesh

DirectMesh is a comprehensive solution for generating surface meshes and finite element models. It can quickly generate ideal surface meshes in Rhinoceros, and generate models for mainstream commercial finite element software based on the surface meshes. It supports Finite element software includes: ANSYS, ABAQUS, FLAC3D, 3DEC, MIDAS, NASTRAN, etc. DirectMesh includes the following ten main functions:

(1) Comprehensive information reader—DRead

(2) Universal surface mesh generator—DMesh

(3)Surface mesh intelligent repairer—DRepair

(4) Compatible surface mesh generator—DMerge

(5) High-quality surface mesher—DRemesh

(6) Finite element model generator—DBuild

(7) Surface mesh feature extractor—DSplit

(8) Surface mesh extrusion generator—DExtrude

(9) Surface mesh extender—DExtend

(10) Geometric object shader—DColor

4 Engineering application

This section takes the finite element analysis of Wuhan Station (Fig. 2) as an example to demonstrate the practicality and efficiency of the proposed method in finite element modeling of complex structural systems. Wuhan Station has a total construction area of 332, 000 square meters and a total of 20 platforms. Platforms 1 to 15 are high-speed stations, and platforms 16 to 20 are general-speed stations. Wuhan Station consists of underground structure, station structure, ancillary structures, etc. The maximum span of the main arch structure is 116m, and the span of the secondary arch is 36m. The entire roof structure is supported by dendritic V-shaped supports on the arch structure.

Fig. 2. Rendering of the Wuhan Station

4.1 Build the finite element model using DirectMesh

The following takes the waiting floor of Wuhan Station as an example to introduce the modeling process in detail. As shown in Fig. 3, first, based on the positioning map information of the waiting layer in CAD, independent grid generation units are used to import into Rhino, and the function of quickly generating 3D entities and grids is sequentially used to generate models and divide surface grids, creating three layers of columns, beams, and slabs. Then use the comprehensive mesh generation unit to assemble the model in each layer, intelligently repair the surface mesh where the combinations of layers contact, and perform high-quality surface mesh re-grinding for lower-quality meshes. Finally, through the finite element model generator, the finite element model of the waiting layer can be obtained.

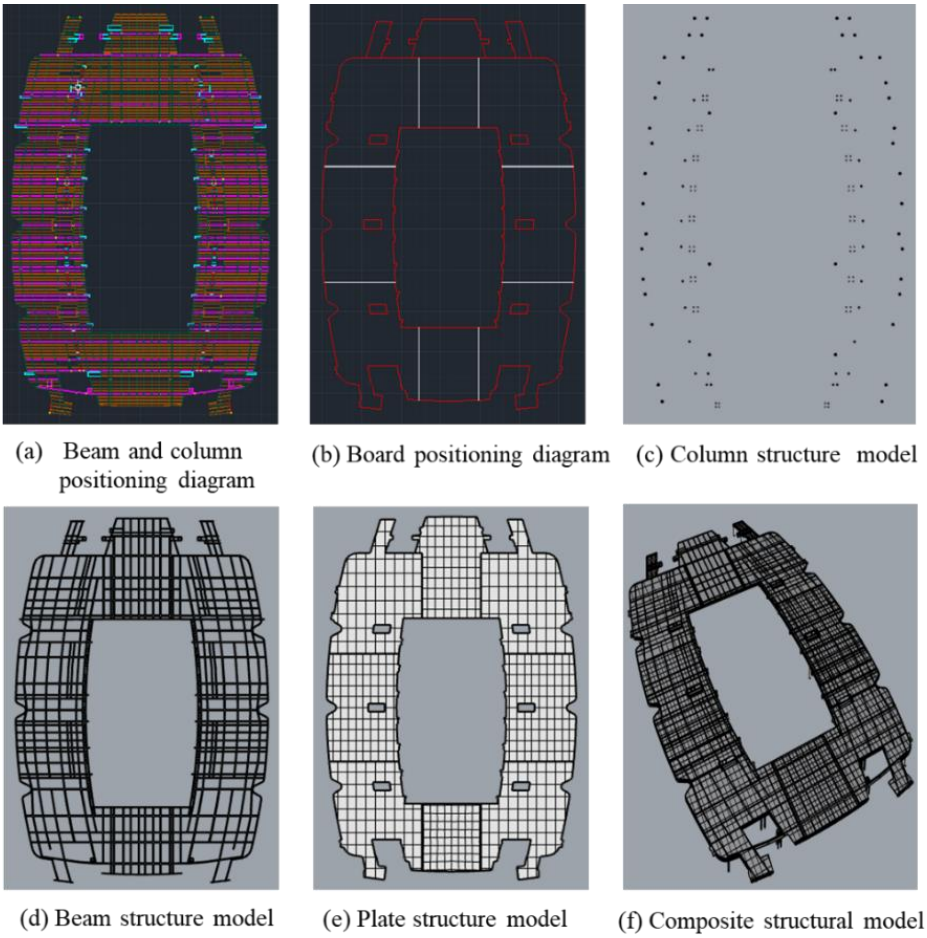


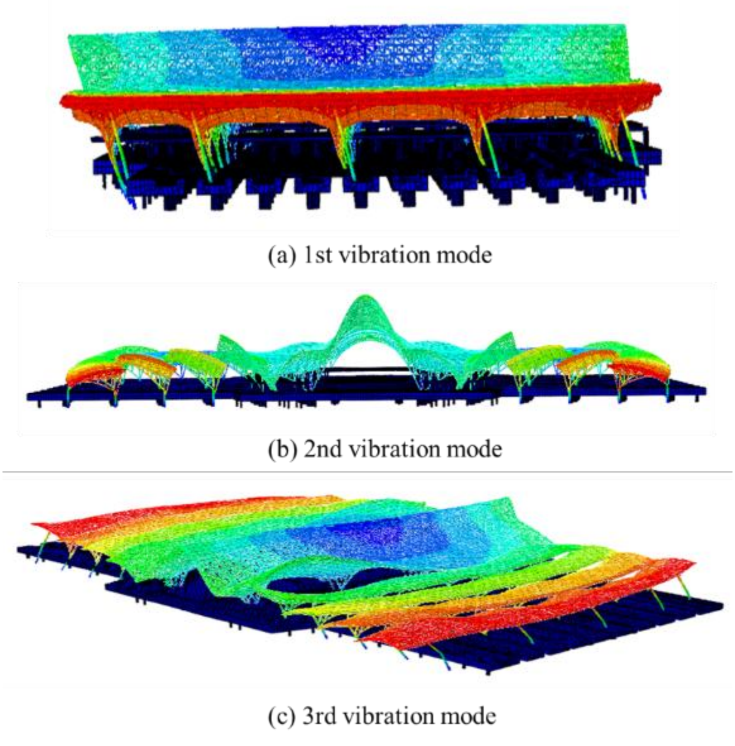
Fig. 3. Modeling process of the Wuhan Station

4.2 Import the model into ABAQUS and complete the analysis

According to the above modeling method, models are established for the soil, platform layer, waiting layer, mezzanine and canopy in sequence, and then combined into an overall finite element model of Wuhan Station. Finally, convert the model file format, import the ABAQUS finite element software, assign material properties to each component in the model in sequence, and define the element types of the components. The beam, column, and roof are simulated using Beam elements, the plate is simulated using Shell elements, and the soil is simulated using Solid elements. For the setting of soil boundary conditions, infinite elements are set around and at the bottom of the soil model, and free boundary conditions are used at the top. The surrounding and bottom elements need to be divided by sweeping. In order to successfully establish the infinite element, the sweep direction must be specified from inside to outside.

4.3 Simulation results and verifications

Use ABAQUS finite element software to analyze the natural vibration characteristics of the imported model, and obtain the results of the first 10 modes of vibration analysis. In addition, Fig. 4 shows the 1-3 order vibration mode diagrams of the overall model. The first vibration mode of the overall model is the Y-axis translation of the roof, the second vibration mode is the vertical vibration of the roof, and the third vibration mode is the Y-axis and vertical vibration of the roof. That is to say, the first three vibration modes are mainly the Y-axis and vertical vibration, which is basically consistent with the measured natural vibration characteristics, proving the accuracy of this modeling method.



(a) 1st vibration mode

(b) 2nd vibration mode

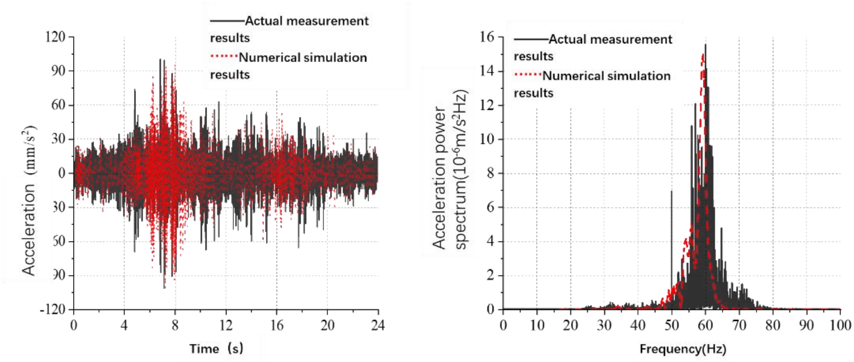
(c) 3rd vibration mode

Fig. 4. Diagrams of the first three vibration modes

Finally, to further validate the accuracy of the modeling method, train loads of the CRH380AL model (see Table 1) were applied at the platform level track of the model. Compare and analyze the numerical simulation results and actual measurement results of vibration response at the same measurement point from the perspectives of time domain and frequency domain. The comparison results are shown in Fig. 12. It can be seen that the time history response and frequency domain distribution of the numerical simulation results are basically consistent with the measured results, which verifies the correctness of the model.

Table 1. Model parameters

Vehicle model	Axle load (ton)	Wheelbase (mm)	Track gauge (mm)	Top speed (km/h)
CRH380AL	15	2500	1435	310



(a) Time domain analysis (b) Frequency domain analysis

Fig. 5. Comparison between the numerical simulations and the experimental measurements

5 Conclusions

This research proposes a general method to complete finite element modeling of complex structural systems based on CAD systems, and implements this method through self-developed software, i.e., DirectMesh, to establish a large-scale comprehensive transportation hub structure-Wuhan Station finite element model. In addition, the following conclusions were drawn:

(1) Using the convenient visualization tools provided by the CAD system can greatly improve the efficiency of finite element modeling;

(2) The method proposed in this article has a good error correction function and can automatically correct errors in the modeling process, ensuring the correctness of the finite element model;

(3) The models generated by this method can be converted between different finite element software, solving the problem of cross-platform incompatibility of finite element models;

(4) By applying this method to the actual design of a large-scale comprehensive transportation hub, a finite element model of Wuhan Station was established. The simulation analysis results were basically consistent with the actual measurement results, verifying the correctness and consistency of the model established based on this method.

Fundings

This work was supported by the National Natural Science Foundation of China (No. 52178301).

Acknowledgments

The authors wish to acknowledge the financial supports provided by the National Natural Science Foundation of China (No. 52178301).

References

1. Zhao Pengfei, Qian Jihong. Characteristics and Prospects of Large-scale Railway Station Building Structure in my country in the New Period[J]. Building Structure, 2011, 41(11):64-68.
2. Zhao Yu, Zhang Xun, Han Feng, Gu Ying, Gu Song. Research on the Rapid Modeling Techniques for Large Railway Stations[J]. Railway Standard Design, 2019, 63(03):125-129+137.
3. Zhu Huiqing, Zhang Chengtao, Zhang Yexing. Development of pre-processing conversion between SAP2000 and ABAQUS based on MATLAB program[J]. Building Structure, 2021, 51(S1):804-807.
4. Zhang Yueqiang, Jiao Chunchun, Ding Jiemin. Critical issues and software development research on model conversion from SAP2000 to ABAQUS for complex high-rise structures[J]. Building Structure, 2013, 43(06):54-57.
5. Zhu Ming, Wang Chunlei. Applying of Rhinoceros software and Grasshopper plug-in for quick structural modeling of double-layer reticulated shells[J]. Building Structure, 2012, 42(S2):424-427.
6. Hu Yue. Development of CAD based automatic quality inspection software for topographic maps [J]. Railway Survey, 2019, 45 (06): 22-24+72. DOI: 10. 19630/j. cnki. tdkc. 201906100001
7. Li Kun. The Application Study of BlM Technology in the Structure Design of Subway Station[J]. Journal of Railway Engineering Society, 2015, 32(02):103-108.
8. Xu Litong, Cui Yanji, Qi Xiangcheng. Research on 3D Engineering Route Construction and Accuracy Based on the Autodesk Platform [J]. Tunnel Construction (Chinese and English), 2021, 41 (06): 964-971