

Explore the application of bim binding with abaqus in bridges

Fanzhou Gui*

School of Civil Engineering, Central South University, Changsha City, Hunan Province, China

*Corresponding author: ext-fanggui@monash.edu

Abstract. Intelligent construction is of great significance to civil engineering, in which BIM modeling technology can greatly improve the efficiency of construction design. However, this technology is still lacking in the field of bridges. Therefore, this paper will explore how to use BIM technology in the field of bridges to improve the efficiency of bridge construction and reduce the construction difficulty. This paper takes Nanping Minjiang Bridge as a model, establishes the bridge model with BIM software and analyzes the model with finite element analysis software abaqus, analyzes the force of the bridge main girder with spatial finite element method and analyzes the construction process, which will provide some reference for the future construction. The experimental results show that by exporting the Revit model to SAT format, the model can be directly imported into Abaqus for the corresponding analysis, thus avoiding the secondary modeling, saving the design time and improving the modeling efficiency. In the preconstruction phase of the bridge, the BIM model can deepen the understanding of the designers and constructors of the project, clarify the internal distribution of components, and accurately calculate the most dangerous areas of stress. This process not only ensures construction safety but also provides important support for engineering inspection after the bridge is completed.

Keywords: Cable-stayed bridge, model, BIM, abaqus, finite element analysis.

1. Introduction

With the development of science and technology and economic growth, there is a growing demand for complex buildings in society, so it is necessary to carry out the necessary structural analysis of building structures.

And ABAQUS software can provide powerful computational ability for the analysis of the model to solve the linear, nonlinear problems of complex structures [1]. At the same time, BIM, as the main core of construction building information modeling, can combine its REVIT software modeling on 3D models with ABAQUS.

In recent years, a group of scholars have carried out research on the combination of BIM technology and finite element analysis, such as Pingping Rao [2], who used BIM technology combined with finite element analysis in the foundation pit project; Jiaran Wang [3], who carried out the secondary development of REVIT through MVVM and combined with HyperWorks and ABAQUS to carry out forward design of the slope protection project; and Yanfan Yang [4], who wrote the Yang Fan [4], by writing the RTA-Trans conversion program, effectively obtained the shear wall and structural reinforcement information in the Revit model and converted it to generate the ABAQUS model; Yang Zhu [5], by using Dynamo in combination with ABAQUS and REVIT, simulated the excavation process of the tunnel to validate the stability of the support structure and the surrounding rock; Tang Fanlong [6], by proposing to use YJK as the conversion interface to import the model in REVIT into the ABAQUS model; Tang Fanlong [6], by using YJK as the conversion interface, imported the model in REVIT into the ABAQUS model. In summary, such studies undoubtedly prove the feasibility of combining BIM technology with ABAQUS.

Currently, there is a part of BIM technology combined with finite element analysis software used in bridge research, Bao Yuxi [7] used BIM technology to establish the Martyrs River Bridge model and through the interface conversion into ANSYS analysis to visualize the results; Wu Wei [8] used commercial software Midas Civil on the bridge steel anchor beams for the establishment of finite element model. But in general, the combination of BIM technology and finite element analysis in the

field of bridges is in the development and exploration stage, the practical experience is insufficient, want to realize intelligent construction in the field of bridges still needs a lot of research.

For this reason, this paper is based on a cable-stayed bridge as the background of the project, the analysis of its engineering projects, according to the construction drawings of the cable-stayed bridge and related data to establish a three-dimensional BIM model, and the realization of REVIT software and finite element software ABAQUS data conversion, the bridge force analysis and collision check, dedicated to improving the stability and safety of the bridge and put forward the feasibility of the relevant recommendations.

2. Method

2.1. Engineering background

This paper takes the Nanping Minjiang River Cable-stayed Bridge as the research background, the bridge is a double-tower and double-surface prestressed concrete cable-stayed continuous girder cantilever collaborative system bridge with a span combination of 45m+160m+272m+130m and a total length of 607m for the main bridge, with a deck slope of 1.15% unidirectional longitudinal slope, and a starting and ending mileage of K0+107.779 ~ K0+714.779, with a center mileage of K0+448.779. +The elevation of the bridge is as follows. The whole bridge has 84 pairs of tension cables, 76 standard sections, 3 merging sections and 4 cast-in-place sections.

2.2. Method

This paper mainly uses two kinds of software for analysis, respectively revit software and abaqus software. Revit software has a powerful modeling function and can do visualization, can greatly improve the designers and builders to understand the construction plan, and improve the efficiency of the design of the collaborative; Abaqus software is mainly applied to the finite element analysis method, and its powerful calculation ability can accurately analyze the deformation of the objects and the force situation

3. Result

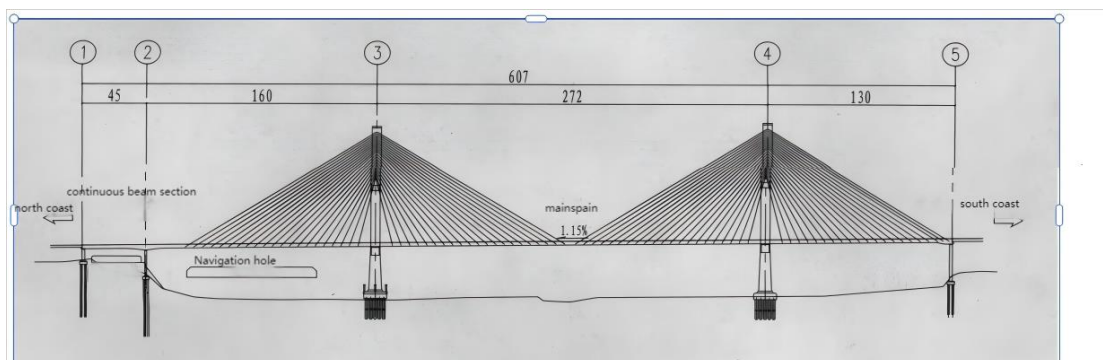


Fig. 1 Elevation drawing of the Minjiang River Bridge

The bridge construction starts from the substructure, the steel cofferdam construction, the bridge pile foundation construction and the bearing platform construction first.

According to the corresponding data, the substructure of the bridge is modeled in Revit, the foundation of the main pier (pier 3 and pier 4) each adopts 22 piles of $\Phi 2.0\text{m}$ drilled rock pile foundation, and the end of the piles enters into the micro-weathering dolomitic quartz schist not less than 5m. the bearing platform of the main pier is 5m thick, the plane of the bearing platform is streamlined, and the top of the bearing platform is set with a separated tower seat. Pier 3 # pier bearing platform set around the flexible energy-absorbing anti-ship collision device (Fig.1).

Pier 1,2,5 adopts pile structure, using 8 Φ 1.2m drilling embedded rock columns, bearing platform plane size 5.2 \times 5.2m, surrounded by an inverted circle, tie beam connection, bearing platform thickness 2.5m.

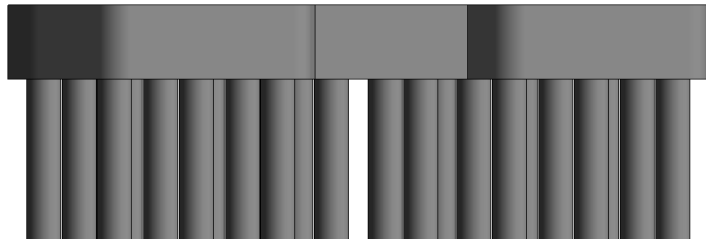


Fig. 2 Pier diagram

The upper crossbeam of the tower is a prestressed reinforced concrete structure, with a crossbeam height of 4m and a width of 5.5m, and the crossbeam is a rectangular hollow box section, with wall thicknesses of 60cm (Fig.2). The lower crossbeam is a prestressed reinforced concrete structure, with a crossbeam height of 5.5m and a width of 6.5m, and the crossbeam is a rectangular hollow box section, with wall thicknesses of 60cm. The main tower is a rectangular body, with a height of 1m and a plan dimension of 6.5m \times 11m. the concrete design grade is C30.

The tower base of the main tower is rectangular, with a height of 1.5m and a plan dimension of 6.5m x 11m. The concrete design grade is C30.

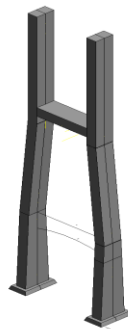


Fig. 3 Main tower diagram

After completing the construction of the bridge tower and substructure, the construction of the main girder was carried out (Fig.3). The main girder of the bridge is divided into a bridge collaboration section, closing section, anchorage section, standard cast-in-suspension section, and gradient section at the main girder of the main tower. Firstly, the construction of collaborative section is carried out, the collaborative span cast-in-situ section of variable cross-section single-box four-compartment structure, the shape of which is consistent with the shape of cast-in-situ girders, the width of the end beams on the side of the approach bridge is 2m, and the end is equipped with a bull-leg structure, which is used as a support point for the cast-in-situ simply supported box girder structure on the south side of the main bridge, the width of the middle cross beams is 2.5m.

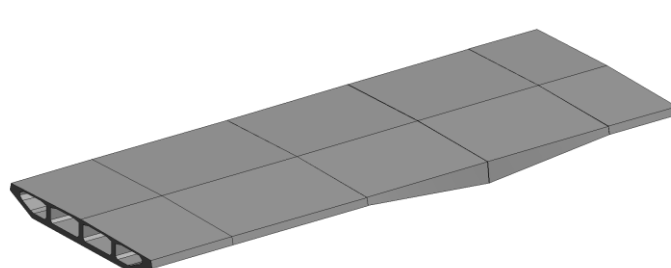


Fig. 3 Picture of the collaboration cross

After completing the collaborative section construction, the anchor section construction will be carried out, the length of the cast-in-place section of the north side anchor span is 18.5m, and the width of the end crossbeam is 2m (Fig.3). The spacing of the last three tail cables of the north side anchor span is 4m, and the design of cross-section heightening is carried out in the range of 10.25m at the end of the anchor span, and the height of box girder cross-section is 4.2m, and the box room is set up with steel-sand concrete for compression weight, and the bull leg is set up at the end of the tail end to be the support point for the approach of 38m prefabricated box girder at the north side of the main bridge. Adopting full-tower bracing method

The cross-section of the suspended main girder of the No.3 main tower still adopts the box girder cross-section at the two tail cables and starts to change gradually at the third cable, completing the transition from box girder cross-section to bilateral box girder cross-section through the length of two segments. This is the gradual change section of the bridge, with 2 sections of 11.75 meters each.

4. Rebar layout and model import

4.1. Arrangement of bar

After the completion of the above construction, the bridge will first carry out the side span of the close construction, a total of 2 places, respectively, connecting the bridge's collaborative section with the standard section of the south bank, the bridge's anchoring section with the standard section of the north bank. The standard section length of the bridge is 6 meters a section, and the north and south side at the same time began to casting, pier 3 # pier a total of 20 pairs of casting sections, section lengths are 6m, Pier 4 south side of the 20 casting sections, north side of the 16 casting sections, the main girder tail 4 sections cast in situ using brackets.

4.2. Reinforcement arrangement and model import

Three-dimensional visual modeling in BIM can see the distribution of rebar in the physical project, the installation order of rebar is the first transverse spacer beams, then beam ribs, and then the top plate. In Revit software, first of all, the model should be set up in the steel protective layer of 35 mm, and then the relevant rebar and layout in the model, the grade of reinforcing steel for HPB335 (Fig.4).

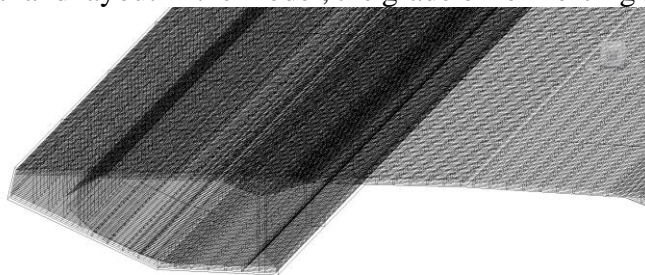


Fig. 4 Section of standard section of main beam

After building the rebar model, export the file in a sat format, import the exported file as a part in Abaqus parts, and assign the modulus of elasticity and Poisson's ratio to the cross-section in the properties (Table 1).

Table 1 Concrete material parameters

Item	C50	C35	C30
Elastic modulus (Mpa)	34500	31500	30000
Shear modulus (Mpa)	13800	12600	12000
Poisson ratio	0.2	0.2	0.2
Design compressive strength	22.4	16.1	13.8
Design tensile strength	1.83	1.52	1.39

4.3. Grid division and optimization

The established model is meshed, and since most of the box girder models are irregular, a tetrahedral mesh is used to divide the grid (Fig.5).

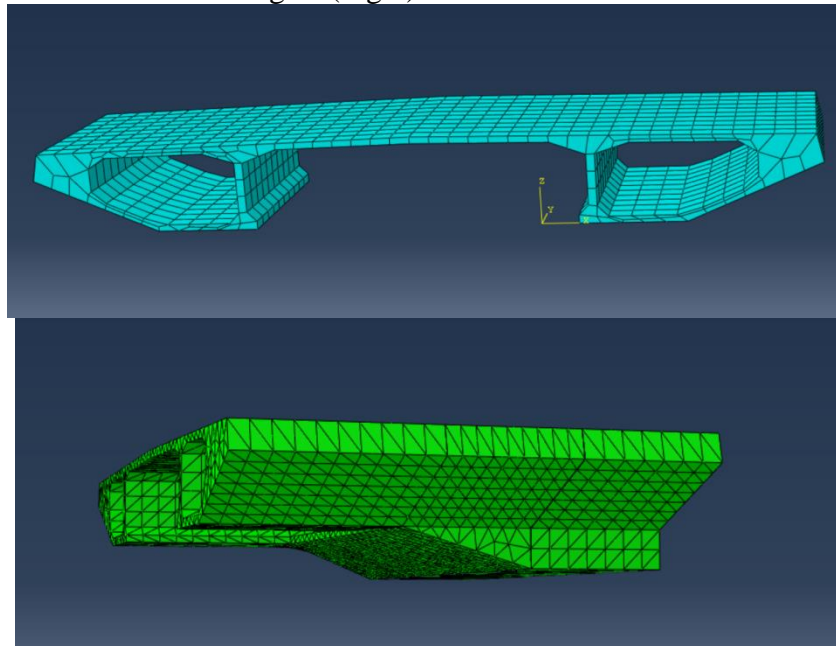


Fig. 5 Model

5. Discussion

After importing the Revit model into Abaqus, this paper observes the lack of rebar information within it, a phenomenon that reveals the shortcomings of the study. Effectively importing rebar data from Building Information Modeling (BIM) into the model analysis is the key to achieving accurate analysis. If the accurate import of rebar nodes can be realized through C# programming, it will help to solve the above problem. However, due to the steep learning curve of C# programming, it is difficult for civil engineering designers to master this skill in a short period of time. Therefore, driving success in the field of smart construction requires interdisciplinary and cross-disciplinary teamwork in order to integrate various aspects of expertise and technology.

6. Conclusion

In this paper, using the software abaqus, we tried to use the spatial finite element method to analyze the force of the bridge, and the construction process of the bridge was analyzed by Revit software. The experimental results show that by exporting the model in Revit in sat format, it can be directly imported into Abaqus to analyze it accordingly, which avoids secondary modeling, saves design time and improves modeling efficiency. In the pre-construction stage of the bridge, the BIM model can deepen the understanding of the designers and constructors of the project, clearly recognize the distribution of components inside the model, and accurately calculate the most dangerous place of the force, to place the sensors here, that is, to ensure the safety of the construction, but also for the construction of the bridge after the completion of the engineering testing can be helpful. However, the modeling in the Revit model is the same as CAD, it can not be completely accurate, and there is a certain discrepancy with the actual construction of the project, so how to combine the actual and the model may be the direction of development in the future, combined with the drone LIDAR and other equipment to achieve the interoperability of the two, to establish an accurate model that can guide the construction of the model.

Reference

- [1] Rao Pingping, Liu Zhuoli, Ban Hongxia, et al. Research on the application of BIM technology combined with finite element analysis in foundation pit engineering [J]. *Information Technology of Civil Engineering and Construction*, 2017, 9(03): 52-57. DOI: 10.16670/j.cnki.cn11-5823/tu.2017.03.09.
- [2] Wang Jiaran. Research on forward design of slope protection engineering based on BIM technology [D]. *Wuhan University of Technology*, 2023. DOI: 10.27727/d.cnki.gwhxc.2023.000296.
- [3] Yang Fan, Long Yuhang, Wei Guohua. Research on Revit-ABAQUS reinforced concrete frame shear wall structure model conversion [J]. *Value Engineering*, 2022, 41(24): 113-115.
- [4] Yang Zhu. Parametric modeling and application of tunnels based on BIM technology [J]. *Engineering Construction*, 2022, 54(09):25-30.DOI:10.13402/j.gcjs.2022.09.111.
- [5] Bao Yuxi. Research on bridge operation and maintenance technology based on BIM and finite element[D]. *Southeast University*,2022.DOI:10.27014/d.cnki.gdnau.2022.000535.
- [6] Wu Wei. Static calculation and analysis of steel anchor beam of a cable-stayed bridge based on ABAQUS[J]. *Guangdong Building Materials*, 2019, 35(06): 47-49.
- [7] Xu Hongsheng, Guo Peiyun, Yan Donghuang, et al. Research on application of integrated analysis of BIM and ABAQUS model[J]. *Highway and Automobile Transportation*, 2022, (01): 128-132.
- [8] Qin Honglei. Construction technology of single-tower double-cable-plane extra-large cable-stayed bridge based on BIM+[J]. *Construction Technology (Chinese and English)*, 2024, 53(05): 62-65.