Research Overview on Numerical Simulation of The Dismantling Process of Large-span Steel Structural Roof

Yang Xiaochi
School of Engineering and Technology, Xi’an Fanyi University, Xi’an, 710105, China
zhangchunhua@xafy.edu.cn

Abstract: The dismantling of large-span steel roof is a complex evolutionary process involving redistribution of internal force. In order to ensure conformity of the structural internal forces to the design, safety and reliability during the demolition, it is necessary to make accurate simulation analysis of the dismantling process to determine a reasonable dismantling plan. With a review of the simulation methods adopted in the dismantling process of large-span steel structure and analysis of the relevant research results, this paper elaborates on advantages, disadvantages and adaptability of each method, which can serve as a basis for the study of roof dismantling.

1. Introduction
In the construction process of a large-span steel structure roof, temporary supports are generally pre-installed during the installation stage. After the structural installation is completed, these temporary supports are removed according to a predetermined method and sequence. Temporary supports can be a reliable platform for high altitude installation of the main structure, reduce its internal force and deformation during the installation process, and improve the safety and installation accuracy of the main structure construction (Figure1). The temporary supports adopted during construction process generally include two types: independent support structure and combined support structure (Figure2).

The tire frame demolition involves the complex transition of mechanical states due to interaction between the main structure and the temporary supports. Temporary loads gradually transfer from the temporary support into permanent load in the main structure. Therefore, it may be a loading process
for the main structure, but discharging process for the temporary supports. The challenges for numerical simulation of the dismantling process may include simulation of the jack, the simulation of the separation and contact between the temporary supports and the main structure, and simulation of rebounds of the temporary supports.

Therefore, the simulation of the dismantling process is rather complicated. One of the key priorities is to choose a proper numerical calculation model to simulate the mechanical characteristics of the jack in the demolition process. There are several methods to simulate characteristic of the jack, which include seat displacement method, equivalent rod-end displacement method, Jack Element Model and Jack-gap Element Model, constraint equation method, and constant-force jack unloading method.

2. Simulation calculation models of dismantling process

Currently, a number of methods are available for simulation of the dismantling process, including seat displacement method, Jack Element Model, equivalent rod-end displacement method, Jack-gap Element Model and constraint equation method.

2.1. Seat Displacement Method

The temporary supports are replaced with corresponding support seats (vertical constraints, see figure 3) during the modelling. In the calculation, a vertical downward displacement load is applied to each support to simulate falling and separation of the support substructure from the main structure. This method is featured by simplicity of the model and no necessity to build temporary support structure in the model. In the calculation, keep track of the counterforce direction of seat corresponding to the removing temporary support. In real applications, the temporary support cannot bear the tensile force. When the calculated reaction force of the support is tensile, the support should be withdrawn. Change the support conditions of the main structure by hand and re-calculate iteratively. When all the seats corresponding to the temporary supports are removed, analysis of the removal process is complete.
2.2. Equivalent Rod-end Displacement Method

This method well considers the compression and rebounds of the supporting tower due to axial pressure change in the falling process. It adopts elastic rod with the right degree. At the same time, the elastic rod adopts an element that can only be compressed but not pulled for simulation. The falling of the support frames is simulated through the displacement of the support seats at the elastic rod-end, as shown in Figure 4.

2.3. Jack Element Model

Jack Element Model is to use both main structure and temporary support substructure to build the model and simulate the real jack with the unit subject only to compression instead of tensile strength. When modelling, set the axial stiffness of the jack element to infinity; when calculating, apply temperature load to control the axial deformation of the jack element.

In this method, the initial length of the jack unit is set to \(l_0\), the material linear expansion coefficient is \(\alpha = l/l_0\). When the temperature change is \(\Delta T\), the elongation of the jack unit is \(\Delta l = \alpha l_0 \Delta T\). That is, the change in the length of the jack unit is equal to temperature change. This method can simulate the compression or rebound caused by the axial pressure change of the support in the process of disassembling. At the same time, due to the characteristics of element being only compressed instead of being pulled, it can accurately simulate the phenomenon that the jack may be temporarily separated from the main structure in the process of disassembling. During the dismantling process when the axial pressure of all the jack elements reaches zero, it signifies completion of the dismantling process. However, when this method adopts temperature load to control the axial deformation of the jack element, it is temporarily constrained by the main structure and temporary supports, it cannot deform completely to match the set deformation value. Therefore, certain errors may arise.

2.4. Jack-gap Element Model

Jack-gap Element Model is proposed by Guo et al. [3] et al to set a two-node gap element between the jack vertex and the node associated with the main structure, connecting the jack element and the gap element in series. It simulates the separation and contact between the temporary supports and the main structure by determination of mechanical states and change of parameters. In this method, as the vertical freedom of at the upper node of the jack is not coupled with the main structure, its length can be reduced accurately through temperature load. Therefore, it is a more accurate method.

2.5. Constraint Equation Method

Constraint equation method is proposed by Guo et al. [5] to make up for the disadvantages of the above four methods. This method models both the main structure and the temporary support system in its calculation. However, it does not adopt a real element to simulate the jack between the two systems,
but avails of constraint equation to simulate the vertical displacement between the upper and lower ends of the jack. In order to simulate the temporary separation that may take place between the jack and the main structure during the actual dismantling process, a short compression-only element is set on the support vertex in the model. The vertical displacements of the corresponding main structure nodes are coupled with the top node of the element through constraint equation.

Constraint equation is a linear equation to link degree of freedom, in the following form:

\[ \text{const} = \sum_{i=1}^{N} (C_{o}(i) \times U(i)) \]

In this equation, where \( U(i) \) is the displacement of node \( i \) in the direction of a certain degree of freedom, \( CO(i) \) is the coefficient corresponding to the displacement \( U(i) \), and \( \text{const} \) is the total displacement corresponding to \( N \) degrees of freedom.

To simulate the falling of the jack during the dismantling process, this equation can express as:

\[ \Delta z_{i-j} = U(\bar{i}) \times C_{o}(\bar{i}) + U(\bar{j}) \times C_{o}(\bar{j}) \]

In the equation, \( U \) is the displacement of the node in the vertical direction of freedom; \( i \) is the upper node of jack corresponding to the actual structure in the model; \( j \) is the node at the lower end of the corresponding jack; where \( \Delta z_{i-j} \) is \( i \), node \( j \) displaces vertically.

For modelling, where \( C_{o}(\bar{i}) = 1 \), \( C_{o}(\bar{j}) = -1 \), \( \text{const} \) is the total falling of the jack under such load. By constantly changing \( \text{const} \), it can simulate the falling process of the jack. When the internal force of all the nodes of the jack is zero, the unloading process is completed.

2.6. Constant-force Jack Unloading Method

Unlike the conventional displacement-control method, Constant-force jack unloading method sets a constant value for the jack at the unloading fulcrum of the structure during demolition. When the force applied to the jack exceeds the set value, the jack is in a pressure maintaining state. The jack-loading rod retracts automatically, but the pressure is constant to the set value. When the force acting on the jack is less than the set value, the stiffness of the jack is infinite. There is no telescopic deformation.

3. The advantages and disadvantages of various algorithm methods and their applications

As these methods are based on different principles, they have respective strength, disadvantages and scope of applications. Seat displacement method is the oldest method, which simulates the falling of the jack by imposing displacement on the seat. However, as the seat is coupled with the vertical degree of node corresponding to the main structure, it is not able to simulate the separation between the main structure and temporary supports and their rebounding.

Equivalent rod-end displacement method can simulate the separation between the temporary supports and main structure as well as the deformation of temporary supports. However, it cannot simulate its combined temporary supports nor the relative displacement between the temporary supports and main structure.

In [9], the author connects the two elements with different characteristics in parallel based on a universal finite element software platform and taking advantage of the unique properties of the tension-only element and the classical beam element in its module. He develops a combined jack element by using degrees of freedom of the corresponding nodes at both ends of the element and the stiffness of the modifying element. He introduces the stiffness coefficients \( \alpha \) and \( \beta \) to endow the combined jack element with the properties of an actual jack.

In [5], the author adopts constraint equation method to simulate and analyse the removal process of the three-point support steel beam. By comparing the features of different simulation methods of demolition process, he has verified the effectiveness and accuracy of constraint equation method. This method can accurately simulate the separation of the temporary support from the main structure during the dismantling process. It can also accurately control the retraction of the jack element, and analyse the internal force and deformation change of the main structure and the temporary supports.

In [1], based on the operation mechanics of the constant-force jack, the author adopts the spring
element COMBIN37 in the universal finite element software ANSYS to simulate the constant-force jack. The results show that the constant-force jack unloading method features convenient modelling and accurate calculation results. It can effectively control internal force in the structure. The deflection of the joint obtained with this method changes linearly. During the displacement and unloading process, some nodes may rebound. However, during the unloading process of the constant-force jack, the displacement will not rebound.

4. Algorithm Implementation

Thanks to the progress of computer software and hardware, particularly the application of large universal software, most of the studies on simulation of the removal process rely on the finite element software. The most popular finite software mainly includes Ansys, MIDAS/Gen, SAP2000. Most of the scholars adopt the above universal software for simulation calculation. Some researchers may program their own software for calculation. But their methods are highly targeted, which cannot be popularized.

ANSYS is a large-scale general finite element analysis (FEA) software developed by Ansys based in USA. It is the fastest growing computer-aided engineering (CAE) software in the world, which can interface with most computer-aided design (CAD, computer Aided design) software for sharing and exchange of data.

MIDAS/Gen is developed by the CAD/CAE of Pohang Group in South Korea in 1989, which entered China in 2002. The structural analysis program RFEM is the basis of a modular software system, which forms a series of software to be applied in civil engineering. That series software includes MIDAS/Gen for construction structural analysis & engineering, MIDAS/CIVIL for bridge structural analysis and optimization, MMIDAS/GTS for rock and tunnel finite element analysis and software for pre-treatment and post treatment.

SAP2000 is an extension from SAP (Structure Analysis Program) created by Edwards Wilson, with a number of versions available in the market. SAP2000 (a replacement for SAP90) is the latest and most mature product among these new generation programs. In addition to all the functions of Plus version, the latest SAP2000 Nonlinear is well suited for time-history analysis during dynamic evaluation of nonlinear structural systems. It is well designed to fit the properties of damping members, damping devices, Gap and Hook materials. It is mainly used for analysis of complex structures with local nonlinear (such as local yielding of base isolation or superstructure elements).

In [4], it elaborates on the whole unloading process of the roof structure of Huoqiu Stadium. It adopts the finite element software MIDAS/Gen to simulate and analyse the various working conditions of the structure unloading process. According to the results, there is consistency among FEA value, actual force under monitoring and the vertical displacement deflection value. Nevertheless, there is also certain deviation. Upon analysis, it is mainly caused by the discrepancies of restraints and load between the model and real situation, as it fails to consider temperature.

In [9], based on the Jack Element Model built by ansys software, it can simulate the operation mechanism of the jack and temporary supports used in the removal process and their interaction with the main structure. The experiment proves the correctness and convenience of Jack Element Model, which can be used to simulate the dismantling process in real applications. It can be used to analyse the internal force of the main structure and the change of deformation.

In [10], it provides analysis on the safety issues arising in the removal of temporary supports for "National Cyber Security Talent and Innovation Base Exhibition Centre" project by using finite element software SPA2000. During the process of the finite element analysis, mathematical calculation is done to compare each unloading process with the actual situation on the site. It proposes for safety evaluation based on the stress and displacement test results.

In [13], based on large-scale finite element software ANSYS, the scholar uses the rod element Link10 in its module featured only by compression without being pulled and connects it with Beam 4 in parallel at the same position. The degrees of freedom at the line displacement of two cell nodes are coupled together to form the new combination element, which is called a jack element.
In [3], based on Jack-gap Element Model, the analysis program is developed by FORTRAN language to accurately and truthfully simulate the demolition process of large-span steel structural roof.

5. Conclusion
This paper reviews the principles, advantages and disadvantages of a number of simulation methods, and explores their real engineering applications, which can serve as a basis for the studies on the dismantling of roofs.

Acknowledgements
Xi’an Fanyi University” School of Engineering and Technology Demonstration Centre” (Z1901).

Reference:
[1] Guo, Yang et al. Constant-force Jack Unloading Method for the Dismantling of Large-span Steel Structural Roof[J], Journal of Zhejiang University (Engineering Science Edition), 2014, 48(10): 1809-1815
[2] Pan. Mechanical Analysis and Application Research on Construction of Large-span Steel Structure[D], PhD Dissertation, South China University of Technology, 2011
[3] Guo, Yang et al. Constant-force Jack Unloading Method for the Dismantling of Large-span Steel Structural Roof[J], Journal of Xi’an University of Architecture and Technology (Natural Science Edition), 2008, 40 (1) : 1-7
[4] Sun. Mechanical Analysis and Monitoring of the Construction Process of Large-span Steel Structure[D], Master Dissertation, Hefei University of Technology, 2017
[5] Guo, Qiu, et al. Constraint Equation Method for Simulating Analysis of Large-span Roof Steel Structure Disassembly Process, Applied Mathematics and Mechanics, 2013, 34(9): 986-994
[6] Luo, Numerical Simulation and CAD Realization of the Whole Process of Large Steel Structure Construction [D], Doctoral Dissertation of Tongji University, 2003
[7] Wang, Shu, Liu, Li. Numerical Analysis on Steel Structure Construction of Art Museums, Steel Structure, 2018, 33(2): 93-97
[8] Li. Construction Technology of Temporary Supports for Steel Structural Roof Truss of a Certain Stadium, Construction Technology, 2018, 47(20): 62-64
[9] Liu, Guo, Guo. Application of Jack Element Model in Numerical Simulation of Large-span Steel Roof Demolition Process, Construction Technology, 2010, 39 (Supplement)
[10] Liao, Guan. Finite Element Analysis on Removal of temporary Supports for Curved Steel Structural Roof, World of Building Materials, 2018, 39(5): 55-57
[11] Liu, Li et al. Overview of Steel Structure Construction Methods for Modern Space, Space Structure, 2016, 22(3): 70-76
[12] Yin, Jiang, Han. Hoisting Stress Analysis of Prefabricated Cement Concrete Pavement Blocks, Construction Technology, 2015, 44 (Supplement)
[13] Liu. Mechanical Analysis and Application Research on Construction of Large-span Complex Steel Structure [D]. Beijing: Tsinghua University, 2008
[14] Pan DZ, wei DM. Construction mechanics analysis & erection monitoring of the roof steel girder for SZCEC [A]. Chen SL, Proceedings of sixth international conference in steel structures[C], Hong Kong, 2009
[15] Ortega N F, Robles S I. Form finding of revolution shells by analyzing mechanical behaviour by means of the finite element method [J]. Journal of Strain Analysis for Engineering Design, 2005, 40(8):775-784
[16] Guan, Chen. Numerical Calculation Methods. Beijing: Tsinghua University Press, 1990
[17] ANSYS Inc. ANSYS Release 10.0 Documentation.