Research on Parametric Finite Element Analysis Method of Escalator Truss

Jie Jiang, Shuijun Wang, Lidan Zhao, Haiping Wu* and Lingxiang Kong
Hangzhou Special Equipment Inspection and Research Institute, Hangzhou, China

*Corresponding author e-mail: 258310606@qq.com

Abstract. On the premise of ensuring the structural strength and safety performance of escalator truss, in order to improve the efficiency of design and development of serial escalator truss, a parametric finite element analysis platform for escalator truss is developed by using the secondary development of finite element method and Visual C++ and data interface technology. In this paper, the structural design and strength characteristics of escalator truss with different parameters are studied, and the strength characteristics of escalator truss with maximum boundary load are analysed. The results show that by using the parameterized finite element analysis platform of escalator, users can get the structural strength results of the serialized truss through automatic modelling calculation only by inputting parameters. While satisfying the structural strength of the truss, the analysis process is greatly simplified and the analysis efficiency is improved.

1. Introduction

As a fixed power equipment with circular moving ladder to transport passengers upward or downward, escalator has been widely used in Department stores, supermarkets, office buildings, hotels, airports, urban transportation and many other fields [1]. The manufacturing process of escalator is complex, which requires a great deal of mechanical analysis. Especially the safety performance analysis of escalator truss directly affects the safe use of escalator. At present, enterprises can only grasp and improve product performance by analyzing and testing products of various varieties one by one. In this way, on the one hand, it is impossible to understand and solve the design problems in the initial stage of product development, and the problem solving takes a long time and costs a lot. On the other hand, it is also very difficult to acquire the understanding and mastery of the comprehensive performance of the product.

In recent years, CAE (Computer Aided Engineering) technique has been widely applied in product development process. CAE has become a useful analysis and optimization technique, since it provides a technical support for reliability and efficiency of the product, avoiding a lot of experimental verifications. However, it is professional and technical. It needs many skills to master the CAE software on the one hand, and the physical process of the product working is also difficult to induce on the other hand. It is one of the reasons which hinder the extensive use of CAE technique. Secondly development technique may be the good way to let the CAE technique use easier [2].

At present, the main production mode of escalator manufacturer is serialized products. Truss, as the key component of escalator, also changes regularly. Therefore, in the process of finite element
analysis of serialized truss, redundant and tedious work and subtraction are the difficulties that enterprises have to solve, and also the starting point of R&D.

This paper presents a concept of parametric finite element analysis of escalator truss by using Visual C++ and finite element secondary development tools. This method will greatly improve the efficiency of R&D and reduce the cost of product development.

2. Truss parameterization
The function of escalator truss is to install and support the components of escalator, bear various loads and connect the two different floors of the building. Figure 1 shows the truss structure. The truss is welded by angle steel, section steel or rectangular steel pipe (Figure 2). The escalator has the following parameters: lifting height; nominal width, usually 600mm, 800mm and 1000mm; rated speed, generally less than 0.75m/s; inclination angle, generally 27.3 degrees, 30 degrees and 35 degrees; maximum conveying capacity.

![Figure 1. Truss Structural Diagram.](image1)

![Figure 2. Truss Angle Steel, Section Steel, Rectangular and Circular Steel.](image2)
From the point of view of serialization development, parameterization of escalator truss is operable. The escalator can be divided into three types according to the angle. The same series of escalators with elevation height, nominal width and material change can be made according to the typical truss. Other parameters can be implemented according to these three typical models. The escalator with inclined angle of 30 degrees, lifting height of 4 m, nominal width of 800 mm and square steel as an example, ①To develop escalators with 6m height, designers only need to build escalators with 4m height to ensure that the connecting parts remain unchanged. They only need to increase the number of truss variable brackets to increase the elevation height and develop the same series of escalators. ②To develop escalators with a width of 1000mm and 6m height, designers should increase the number of variable supports and change the length of the connecting beams on both sides. ③To develop elevators with 1000mm width, 6m height and angle steel, the number of supports, the length of connecting beams and the material of elements must be changed. So the idea of enterprise serialization development of escalator is a reference for parametric finite element analysis.

3. Finite Element Parametric Realization Process

3.1. Secondary Development of ANSYS

APDL (ANSYS Parametric Design Language) is an interpretative text language, which is the simplest and most effective secondary development tool for ANSYS. APDL can organize ANSYS command stream in macro form and develop parameterized user program to realize the whole process of step-by-step finite element analysis. In this way, ANSYS can customize analysis navigation toolbar, a series of parameterized dialog boxes, three-dimensional model, material definition, meshing, load and boundary conditions, analysis control and solution, and parameterized post-processing.

Visual C++ (VC) is an efficient programming language tool. To start ANSYS program in V C++ 6.0, Win Exec, and Shell Execute and Create Process functions can be used. The Wine exec () function is used to run another program in an application. Its prototype is as follows:

```
UINT Winexecexece (LPCSTR lpCmdLine, //Program Path
UINT cumshaw//Window Display);
```

Function creates a process to execute other programs. It can specify the process's security attributes, inheritance information, class priority, etc. Using VC, it can develop integrated ANSYS analysis navigation operation interface. The data interface between ANSYS and VC is batch processing. APDL macros store command stream statements, which can be directly invoked by VC. The parameterized ANSYS command is directly invoked by VC through command [3, 4].

3.2. Parametric Realization of Finite Element Method

According to the above analysis characteristics, the finite element method of truss can be parameterized. The parameterized analysis process is shown in Figure.3: ①Input geometric parameters and analysis parameters through the interface parametric dialog box; ②VC generates APDL executable files; ③VC calls APDL for finite element analysis; ④The generated files are stored in the database for users to view.
Figure 3. Flowchart of parametric CAE analysis.

For parameterized analysis of truss, APDL template files for truss to be analyzed are selected, variable data are input, new APDL command stream files are generated, and ANSYS is called by VC for finite element analysis. The analysis results are stored in database for interface call [5-10].

4. Example application

Based on the background of manufacturing informationization and intelligent manufacturing in Zhejiang Province and the production mode of a local escalator enterprise, this paper develops a parametric truss analysis platform (Figure.4). Taking the truss analysis as an example, the parametric settings are used in the series analysis of the same type of truss. The designers only need to input the corresponding data to obtain the desired results. In the interface of Fig.4, first create the project number, project name, new file name, input inclination angle and select typical template, the material parameters of section steel, angle steel and round steel are changed in the form of elastic modulus and Poisson's ratio. The lifting height can be input directly or can change the number of supports per unit. In addition to the connecting parts, the length of the upper and lower ends can also be changed. The parameter input completes the generation of CAD model, and the interface clicks to perform the calculation. The CAD model is imported into ANSYS in IGES format, and then into ANSYS interface for finite element analysis. Simple toolbars and step-by-step toolbars are customized in ANSYS (Figure. 5 and Figure. 6) to adjust the finite element model, which is conducive to the success of finite element analysis and calculation. The results of analysis are stored in the database. The platform can view the stress and deformation maps (Figures.7 and 8), and can also view the comparative information of serialization.
Figure 4. Parametric CAE Analysis Interface.

Figure 5. Direct Analysis Toolbar.

Figure 6. Multi-step Analysis Toolbar.

Figure 7. Equivalent Stress Nephogram of the Third Strength Theory.
5. Conclusion
In this paper, an efficient analysis platform for escalator truss parameters is developed by using programming software and finite element APDL tools. Using this platform, users only need to input the required parameters, and the platform can perform simple operation to obtain the data of finite element analysis results of truss structure, which makes the finite element analysis operation not a professional CAE analyst, and the general design and development of structural products can also be easily started. The advantages of the method presented in this paper are to ensure the results of finite element analysis, improve the accuracy of design calculation and enhance the reliability of products, simplify the analysis process and avoid a lot of repetitive work, thus speeding up the analysis and shortening the development cycle of products.

References
[1] Xinfang Shi, Qingdong Jiang, Chunlei Li, Meiwan Rao, Escalator, Machinery Industry Press, Beijing, 2014.
[2] Fang Shi, The Second Development and Application of ANSYS, China WaterPower Press, Beijing, 2012.
[3] Canonsburg, ANSYS, INC. ANSYS 17.0 Help, PA: ANSYS, INC, 2017.
[4] Wenwei Zhu, Visual C++ 2013 From Introduction to Proficiency, Tsinghua University Press, Beijing, 2017.
[5] Biao Liu, Wenming Cheng, Liyuan Garden, Parametric Finite Element Analysis of Portal Crane Structure Based on VB and APDL, Lifting and Transportation Machinery. 12 (2010) 12 - 13.
[6] Cuncen Li, Shunrong Li, Rong Xu, Development of Mobile Office System for Special Equipment Supervision and Inspection Based on Internet of Things, China Safety Production Science and Technology. 9 (2013) 165 - 170.
[7] Hongbing Zhang, Changdong Wan, Guangqing Shang, Finite Element Lightweight Analysis of Escalator Truss Structures, Mechanical Engineering. 29 (2012) 1139 - 1142.
[8] Guangyuan Zhan, Finite Element Analysis of Escalator Truss, Elevator Industry. 7 (2015) 61 - 64.

[9] Yuan Gao, Guohua Cao, Jiancong Qin, Finite Element Analysis of Marine Escalator Truss and Its Application, Mechanical Strength. 3 (2017) 78 - 80.

[10] Yang Chen, Development of Escalator Truss Structure Automatic Modeling and Analysis System Based on ANSYS, Nanjing University of Aeronautics and Astronautics. 3 (2016) 147 - 148.