Applied Mathematics, Modeling and Computer Simulation
C.-H. Chen et al. (Eds.)

doi:10.3233/ATDE230988

Design of Static Strength Analysis Software for Conveyor Head Tripod Based on ANSYS and MATLAB

Dong Zheng[1]
China Coal Technology & Engineering Group Shanghai Co., Ltd., Shanghai, China

Abstract. This paper has realized the implementation the specific command program of the MATLAB GUI to build the belt conveyor head tripod strength analysis system. Through the classical theoretical formula and ANSYS APDL command flow, the tripod strength theoretical calculation and finite element calculation are realized respectively, and the triangle is input in the MATLAB GUI interface. The corresponding structural size parameters and loads of the frame will automatically complete the strength check of the classical theoretical formula, and at the same time, the corresponding data will be transferred with ANSYS, and the calculation and result output will be automatically completed. Finally, the corresponding interface software is formed and provided to the designer, which greatly improves Simulation efficiency.

Key words. ANSYS; MATLAB; theoretical calculation; interface.

1. Introduction

The head tripod of the belt conveyor is the key component of the belt conveyor, which is mainly used to install the diversion drum and drive drum and bear the tension of the belt. With the continuous development of economy, more and more conveyors are used in production [1-3]. There are many methods to calculate the force on the conveyor head tripod, but classical theoretical calculation and finite element simulation are undoubtedly the most time-saving and labor-saving methods. Through the interface theoretical formula and simulation analysis, the strength analysis of the conveyor head tripod can be quickly realized, and the calculation and analysis efficiency of the tripod can be greatly improved [4,5].

MATLAB GUI is a graphical user interface software. It can display the program language in a visual way, produce a friendly graphical interface, and greatly improve the calculation efficiency. The APDL language provided by ANSYS can automatically complete the establishment of the size and load through the adjustment of structural design parameters, and finally get the corresponding simulation results. Based on MATLAB and ANSYS software [6], this paper designs a set of strength calculation

[1] Corresponding Author, Dong Zheng, China Coal Technology & Engineering Group shanghai Co., Ltd., Shanghai., China; E-mail: 191419408@qq.com.

system for belt conveyor head tripod, which realizes the data transfer and nested operation between different software, and quickly realizes the strength analysis and calculation of belt conveyor head tripod by means of interface parameterization, which improves the simulation efficiency and shortens the design cycle.

2. Triangle Theoretical Analysis Interface

The tripod is mainly welded from steel plates, and the relevant structure is shown in figure 1. During normal operation, it is mainly subject to the tension of the conveyor belt, ignoring the weight, friction and inertia force of the drum itself. Therefore, the stress of the tripod is as follows:

$$F_{\text{horizontal}} = F_1 \cos\alpha + F_2 \cos\beta$$
$$F_{vertical} = F_1 \sin\alpha - F_2 \sin\beta$$

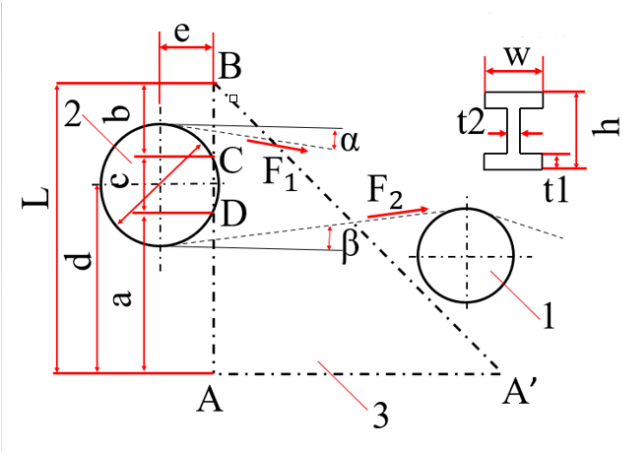

Figure 1. Structural parameters of tripod.

1. Drum 2. Drum 3. Tripod. F_1. Belt tension. F_2. Belt tension. α, β. Angle between the belt and the horizontal direction a. Vertical height of the connection point D between the drum and the tripod b. Vertical height of the connection point C between the drum and the tripod from the top B c. Vertical height of the connection point C and the connection point D d.Center height of the drum 2 e. Horizontal distance from the drum center to the tripod h.Web height w.Flange width t1.Flange thickness t2.Web thickness。

The calculation formulas of key points are written and calculated in MATLAB, and presented in the GUI interface. The input of structural parameters is shown in figure 2.

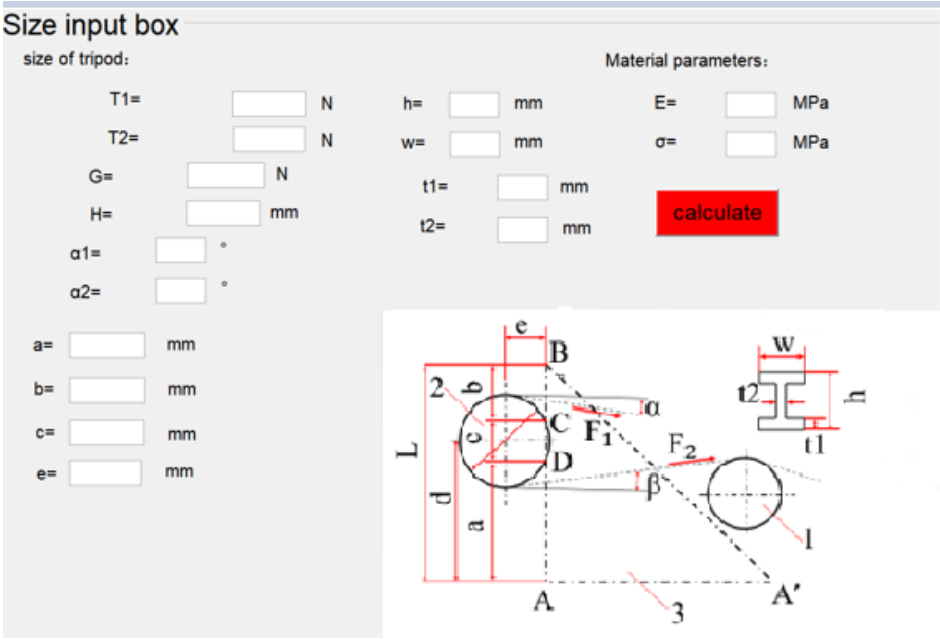

Figure 2. MATLAB theoretical analysis interface.

3. Finite Element Analysis Interface

The MATLAB GUI is based on C language for calculation. Therefore, first of all, these parameters on the interface must be correctly transferred to ANSYS, and then the ANSYS software completes modeling, meshing and calculation according to the input parameters[7-9]. The idea of this paper is to first read and write the geometric parameters and corresponding load parameters entered in the GUI theoretical interface into a*. txt file. Here, it should be noted that each parameter should be given a corresponding name, and then call it by writing the corresponding APDL command stream, and then complete parametric modeling, mesh division, boundary condition application and finite element calculation.

(1) MATLAB transfers data:

rtd=str2num(get(handles.edit5,'String'));%Read in one of the parameters on the GUI interface;

scl=str2num(get(handles.edit6,'String'));%Read in one of the parameters on the GUI interface;

pra=[rtd scl];%Two parameters are combined into a matrix;

duru='\shuru.txt';%Start a txt file named shuru;

shuru1=strcat(filep1,duru);%Create a new txt file named shuru under the working directory filep1;

fid=fopen(shuru1,'wt');%Open the new txt file;

fprintf(fid,'%9.4f\n',pra);%Read the parameters of the GUI interface into the new txt file;

fclose(fid);%Close txt file;

The above program is a part of the program for parameter input in MATLAB, in which the data on the GUI interface is kept in the shuru.txt file, and then called through APDL.

(2) ANSYS read in data

copyfile('jiazai1.txt',filep1);%Copy the written APDL command stream jiazai1.txt (which contains commands such as modeling, meshing, and applying boundary loads) to the corresponding working directory;

*DIM,AA,,1,2 % APDL command stream defines AA array;

*VREAD,AA(1,1),shuru,txt,,JIK,2,1%Read in the data parameters on the storage GUI interface and assign them to the array AA;

(3) MATLAB starts ANSYS and calculates

system('"D:\ansys\ANSYSInc\v192\ansys\bin\winx64\ansys192.exe" -b -i "jiazai1.txt" -o "mshuchu.out"');%Start ANSYS. Execute APDL file;

Using the ANSYS software through the system command. Note that the installation path of ANSYS needs to be set in advance when MATLAB calls the ANSYS software.

(4) ANSYS analysis and calculation results are transferred to MATLAB and displayed

/show,jpeg and PLNSOL, U,SUM, 0,1.0 %Generate displacement nephogram in ANSYS;

/show,close amo=imread('jpeg.jpg');%Read displacement nephogram;

imshow(amo);%Displacement nephogram is displayed in the interface;

(5) Operation analysis interface flow

According to the analysis process of the strength of the conveyor head tripod, the corresponding finite element analysis interface is developed and edited. The analysis process is shown in figure 3, and the finite element analysis interface is shown in figure 4.

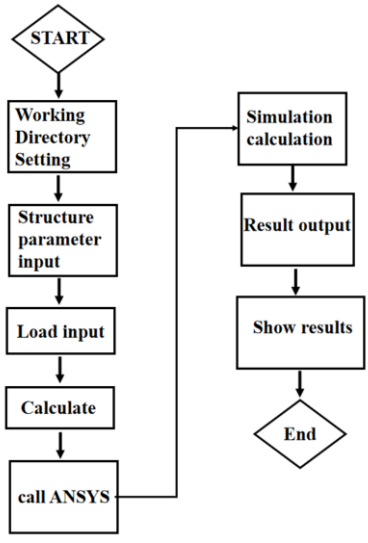

Figure 3. Flow Chart of Analysis Interface.

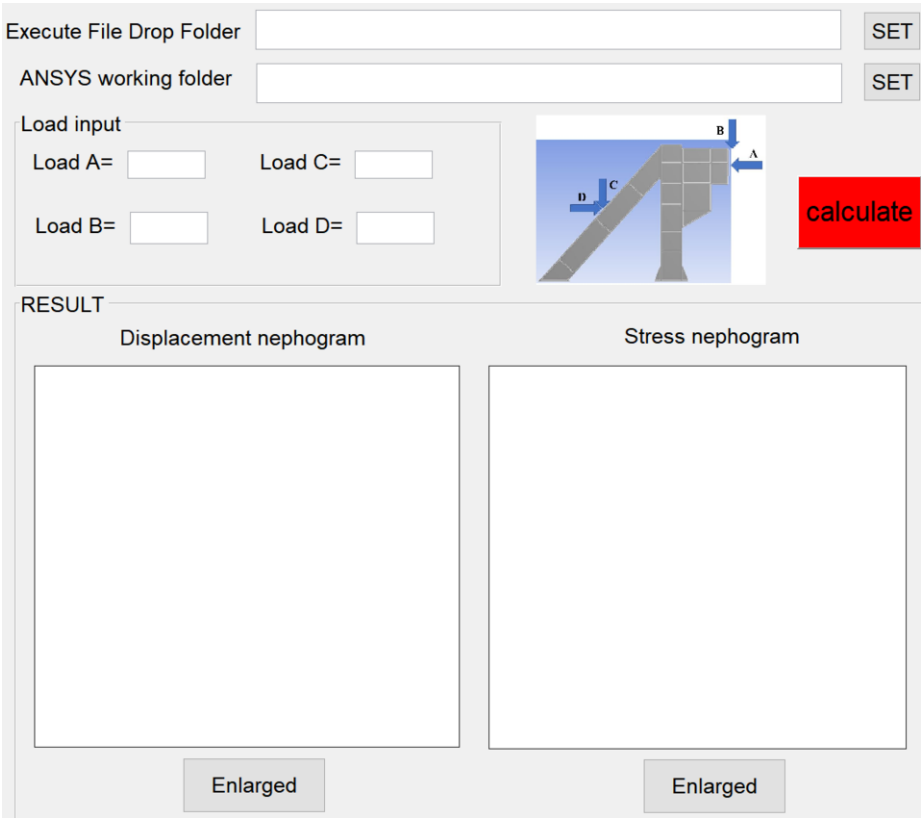

Figure 4. MATLAB finite element analysis interface.

4. Case Analysis and Verification

According to the above methods, the theoretical calculation and finite element analysis interface of the strength of the conveyor head tripod is designed. The theoretical calculation formula and finite element analysis are presented in a visual interface. The strength of the conveyor head tripod under different design parameters is checked. The combined time of the two analysis is not more than 5 minutes. The corresponding calculation result display window is shown in figure 5.

On the same computer and under the same setting conditions, the adoption of this interface setting analysis can greatly improve the analysis efficiency and reduce the workload of designers. At the same time, the interface realizes the integrated analysis of MATLAB and ANSYS [10]. The designers can quickly complete the strength check of the conveyor head tripod by inputting the corresponding parameters, greatly reducing the threshold of analysis and calculation.

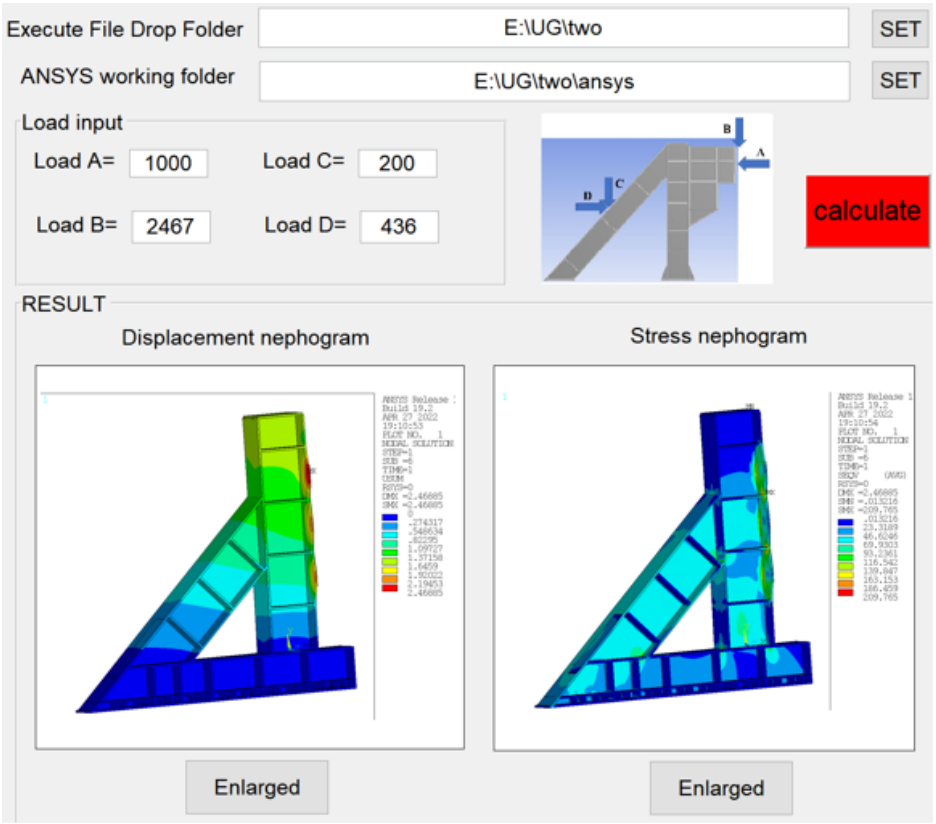

Figure 5. Calculation result display interface.

5. Conclusion

The APDL command flow of ANSYS can easily complete parameterized modeling, analysis and calculation. The powerful numerical calculation ability and rich function call instructions provide powerful tools of MATLAB for developing specialized software. This paper combines the advantages of the two software to develop a professional analysis software, which reduces a lot of repetitive work. The designer can complete the corresponding analysis only by inputting the structural parameters, reduces the threshold of simulation analysis, greatly improves the design efficiency and design level of the conveyor head tripod, and provides a new method for other subsequent design analysis.

References

[1] Ma H, Mao Q, Zhang X. Research progress of intelligent monitoring technology for mining power belt conveyors [J]. Vibration. Testing and Diagnosis, 2016, 36(02): 213-219.

[2] Wu P, Zeng H, Han M. Implementation method of secondary development technology based on ANSYS [J]. Journal of Liaoning Institute of Technology, 2004 (5): 25-29.

[3] Xia R. APDL Parametric Finite-Free Analysis Technique and Its Application [J]. Development and Innovation of Mechanical and Electrical Products, 2008(2): 103-104.
[4] Mao Q, Mao J, Ma H, Zhang X, Li Z. Research on Intelligent Monitoring System of Mining Belt Conveyor [J]. Industrial and Mining Automation, 2020, 46(06): 48-52.
[5] Wang H. Parametric design and finite element analysis of belt conveyor rollers and tripods [D]. Northeastern University, 2007.
[6] Shi X, Zhou J. Proficient in GUI programming [M]. Beijing: Peking University Press, 2003.
[7] SM Khot, Nitesh P Yelve. Modeling and response analysis of dynamic systems by using ANSYS and MATLAB[J].Journal of Vibration and Control,2010,17(6),953-958.
[8] S. A. Siddiqui, A. Ahmad, A. A. Siddiqui and P. Chaturvedi, Stability Analysis of a Cantilever Structure using ANSYS and MATLAB, 2nd International Conference on Intelligent Engineering and Management (ICIEM), London, United Kingdom, 2021, pp. 7-12.
[9] Duc Cong Nguyen, Marek Salamak, Andrzej Katunin, Michael Gerges, Mohamed Abdel-Maguid. Finite element model updating of steel-concrete composite bridge: A study case of the Ruri bridge in Vietnam[J]. Archives of Civil Engineering,2023,60(03),pp,425-443
[10] Yuan F, Miao J, Zheng L, Wang W. Optimal design of Multi-Objective Parameters for Interference Fit of Motor Fan Pedestal based on ANSYS [J]. Journal of Physics: Conference Series, 2022.