Opinion Article

MATLAB combined with APDL simulation -- taking Y-shaped steel turnout pipe as an example

Abstract: This paper takes a symmetric Y-shaped steel bifurcation pipe as an example, uses ANSYS finite element analysis software to carry out parametric modeling, grid division and load application, combined with MATLAB software to carry out operation processing, and finally analyzes the results and draws a conclusion.

Keywords: ANSYS; Steel fork pipe; Finite element analysis MATLAB combined with APDL simulation

1 Introduction

In modern engineering analysis and design, the ability to combine different computational tools and platforms has become a powerful strategy. The co-simulation of MATLAB and APDL is an important example. By combining the powerful numerical computation and data processing capabilities of MATLAB with the application capabilities of APDL in finite element analysis, more efficient and accurate simulations and optimizations can be achieved. By combining MATLAB with APDL, we can make full use programming ability of MATLAB to generate and process data, and carry out detailed finite element simulation through APDL [6-8]. This kind of co-simulation not only improves the flexibility of model building and analysis, but also expands the application range of simulation.

2 Finite element treatment of symmetric Y-shaped steel bifurcation pipe

In this paper, the model is established by using line rotation to form a plane. First, the line is generated from the point, and then the line rotation generates a plane. When establishing symmetric Y-shaped steel turnout pipe, parametric modeling is carried out first. The purpose of this step is to express the data of some parts with parameters, and use the relevant addition, subtraction, multiplication and division operations of these parameters to obtain the relationship formula to represent the position. On the one hand, it is convenient to modify the parameters later, and on the other hand, it can make the joint call smoothly. Because the symmetric Y-shaped steel bifurcation pipe is symmetrical along the center of the symmetry axis, it is only necessary to create part of the model and then mirror the related axis to get the completed model. Let's start creating the model.

2.1 Preparing for model parameterization Creation

This model is the structure of a hydropower station pumping steel bifurcation, which is symmetrical Y-shaped steel bifurcation. First of all, the data that needs to be used is expressed with parameters, and a TXT file is created for APDL call processing, and it is also prepared for the subsequent MATLAB joint analysis. First of all, the above model size is written into a TXT file in the form of a matrix, named as the input file, indicating that the required input data is in this TXT file. DATA.TXT.

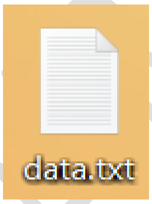

Figure 1 Input data TXT file

2.2 Model Creation and Post-processing

For the construction of bifurcated tube, the mirror method is used to create a model and get the mirror image on the symmetry axis. First determine the pipe axis, and then establish the pipe section of the main section. It is necessary to determine the axis of the pipe when creating the model of the turnout pipe, and to determine the position and rotation of the shell surface by the axis of the pipe. The relevant points are connected into a straight line and then rotated to get a surface. As for the irregular shape of the pyramidal canal, the main pyramidal canal can be seen as formed by cutting the torus, so the main vertebral canal surface can be obtained by creating the torus and cutting it. So the main segment is created,After creating the main surface part, create the branch spinal canal surface part. In addition to the irregular shape of the spinal canal model, the other transitional canals and branches of the surface are relatively regular, which can be created by the above method, so it is not necessary to go into details. The creation of the branch canal surface needs to use the command provided by ANSYS to create the point of the surface. After the creation of the branch canal surface, the relevant points can be selected to create the surface. In this way, a partial steel switch model is obtained, and then a complete steel switch model as shown in the figure below is obtained by mirroring.

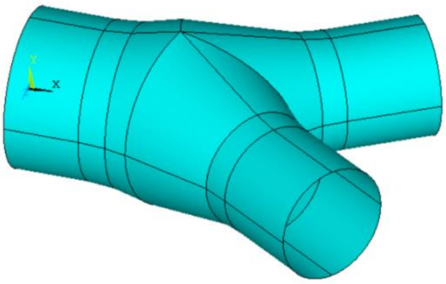

Figure 2 Steel bifurcation model

When changing the model, it is only necessary to change the relevant parameters to obtain the corresponding model, such as changing the Angle of the fork Angle. The following is the model of the steel fork pipe under different division angles, from left to right, respectively, when the division Angle is 80°, 75°, 70° :

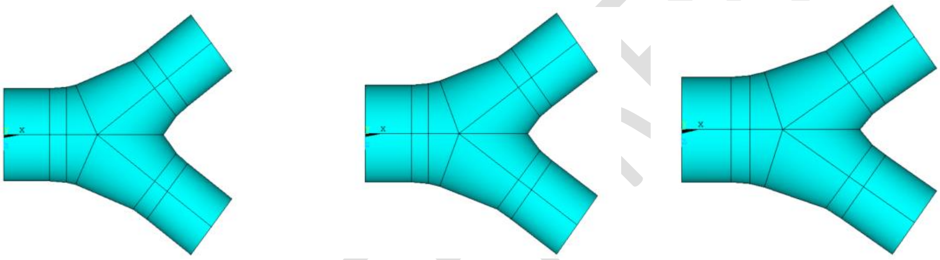

Figure 3 Models of steel bifurcations with different parting angles

3 MATLAB and APDL Settings

MATLAB calls ANSYS for finite element analysis as follows:

(1) MATLAB generates data and writes it into a TXT file "input.TXT" in the form of scientific notation;

(2) Write the APDL program of ANSYS, use the APDL program in the MATLAB environment, ANSYS runs in batch mode for analysis and solution, and outputs the results to be analyzed, and writes them into a TXT file "output.TXT";

(3) MATLAB calls "output.TXT" for data analysis.

4. Joint analysis of MATLAB and APDL

The steel bifurcation pipe of a hydropower station is symmetrical Y-shaped, the material is Q690 steel, the elastic modulus is 2.06×105, the Poisson ratio is 0.3, the maximum radius of the common tangent sphere is 2.7 meters, the inner radius of the main entrance is 2.3 meters, the inner radius of the branch pipe is 1.6 meters, and the difference Angle is 70°. The radius of the main transition tube and the main conical tube is 2.4 meters, and the radius of the branch transition tube and the main conical tube is 1.7 meters.

After the parameters are determined, the model is established. When modeling, you can use the program creation identified in the second section above. The parameters are first written as input files in the form of matrices, and then APDL calls the input files to complete a series of procedures such as modeling, grid division, and load application according to the written command flow. According to the size, the initial grid number of each area is determined to be 5. The final model setting results are shown below, and the maximum stress is 303.10Mpa.

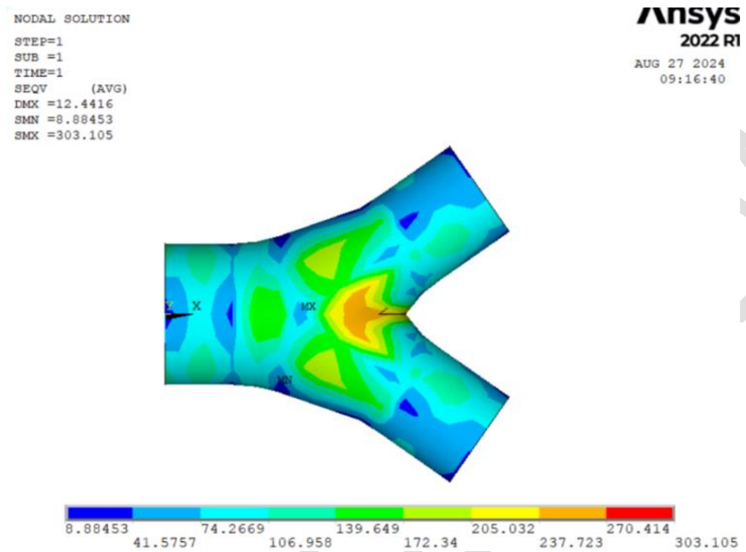

Figure 4 Stress cloud diagram of steel turnout pipe

After the maximum stress is generated, an OUTDATA.TXT file needs to be created as an output file to ensure that MATLAB can call the loop. After the setting is completed, the model is processed to explore the influence of different mesh numbers on the maximum stress:The number of grids is initially set to increase from 5 to 50, and in the middle, 5 more grids are divided according to each area, and 10 grid number data is obtained. After calling the input in turn, the relationship between the maximum stress and the number of grids is obtained.The obtained data are summarized into the following table:

Number of grids	5	10	15	20	25	30	35	40	45	50
Stress magnitude	303.11	427.17	469.87	502.89	523.23	534.56	541.03	544.66	552.98	559.58

Table 1 Relationship between mesh size and maximum stress

Draw the above table as an image in MATLAB to get the following line chart:

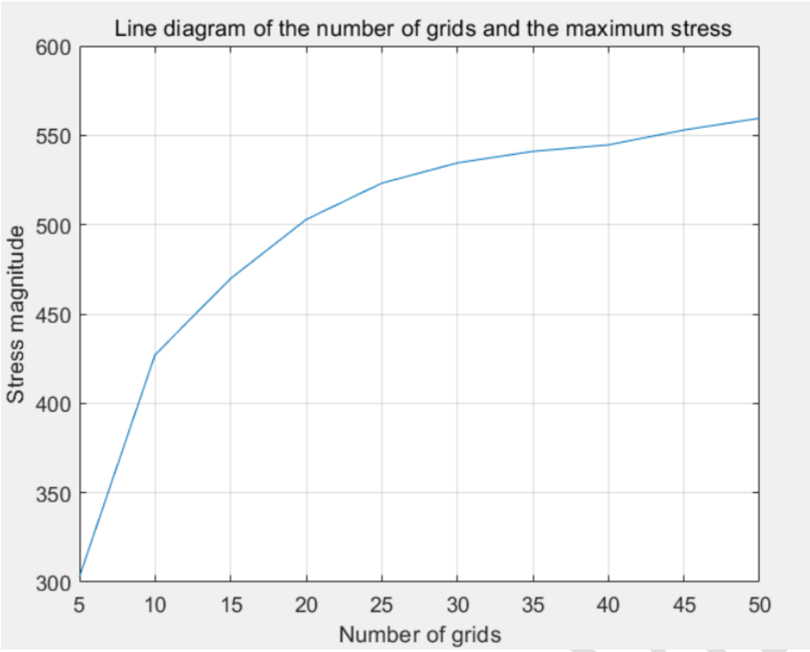

Figure 5 Line diagram of the number of grids and the maximum stress

According to the above table and line diagram, it can be seen that in ANSYS, the number of grids affects the model stress situation, and the fewer the number of grids, the larger the mesh size, and thus the smaller the stress. Conversely, the greater the number of grids, the smaller the mesh size and the greater the stress. In this model, when the number of grids reaches a certain value, that is, 30, the maximum stress increases gently.

5 Conclusion

In this paper, the parametric modeling method of symmetric Y-shaped steel bifurcated pipe is explored and the results of different models are created when the parameters are changed. At the same time, MATLAB software is used to analyze the influence of different mesh sizes on the maximum stress. It is concluded that the smaller the mesh size, the greater the maximum stress, and beyond a certain critical value, the maximum stress increase tends to be stable.

6 References

[1] Keding Liu,and Zhichao Yang."Finite Element Analysis for Steel Bifurcation Pipe of Zhanghewan Pumped Storage Power Station".Ed.2014,

[2] Chang Zhi Ji, et al."Numerical Simulation of Hydraulic Shape Optimization for Bifurcated Pipe of Hydropower Station."Applied Mechanics and Materials 1800.170-173(2012):3507-3511.

[3] Pan Shaohong,Wu Yunen,and Ding Jiajia."Research on the Design of Crescent Ribbed Steel Bifurcation Pipe of High Head Hydropower Station- Take Jinzhan River -I Hydropower Station as an Example."IOP Conference Series: Earth and Environmental Science 719.3(2021):

[4] Hu Lei, et al."Hydraulic load-bearing mechanism in rib-strengthened steel bifurcation structures of hydroelectric power plants: Numerical simulation."Mechanics Based Design of Structures and Machines 51.8(2023):4330-4346.

[5] Jingxi Duan,and Ning Chen."Finite Element Analysis of UHV Transmission Line Stringing System Based on APDL."Journal of Physics: Conference Series 2755.1(2024):

[6] Hilário JC, Braz-Cesar M, Andrade C, Borges AS. Matlab® algorithm to simulate the dynamic behavior of an NiTi alloy through Ansys® APDLTM models. KnE Engineering. 2020 Jun 2:88-102.

[7] Gauchía A, Boada BL, Boada MJ, Díaz V. Integration of MATLAB and ANSYS for advanced analysis of vehicle structures. MATLAB applications for the practical engineer. 2014 Sep 8;2017.

[8] Zaman I, Rozlan SA, Azmir NA, Ismon M, Madlan MA, Yahya MN, Zainulabidin MH, Sani MS, Noh MF. Modelling of Rigid Walled Enclosure Couple to a Flexible Wall using Matlab and Ansys APDL. InJournal of Physics: Conference Series 2017 Oct 1 (Vol. 914, No. 1, p. 012038). IOP Publishing.