Opinion Article

MATLAB combined with APDL simulation in reference to Y-shaped steel turnout pipe

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；APDL Finite Element Analysis combined with MATLAB 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 Creating Parametric Model

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.

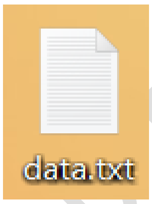


Figure 1 Input data TXT file

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.

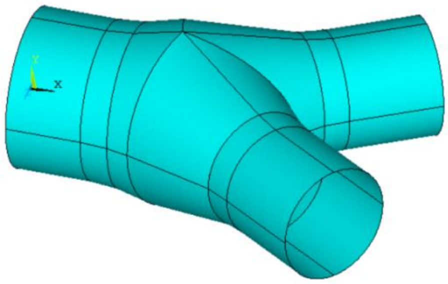


Figure 2 Steel bifurcation model

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° :

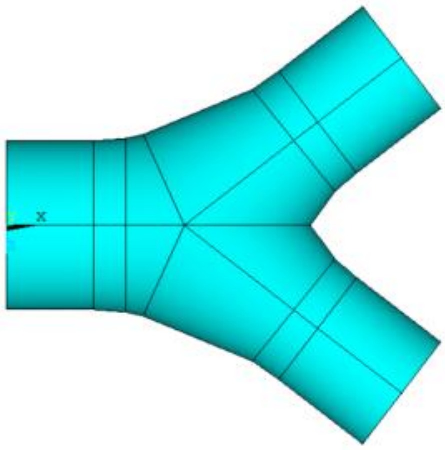


Figure 3(a) 80° bifurcation angle steel pipe

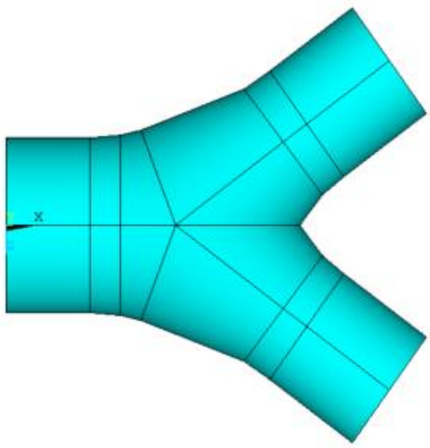


Figure 3(b) 75° bifurcation angle steel pipe

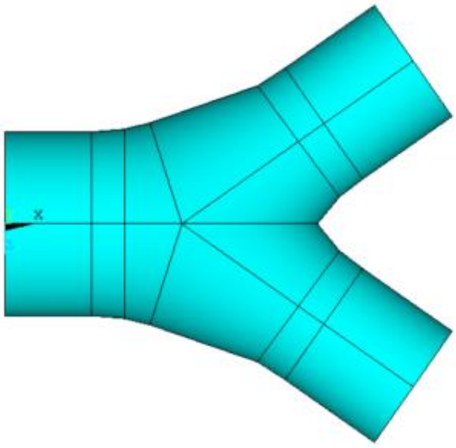


Figure3(c) 70° bifurcation angle steel pipe

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.

Once the parameters have been determined, the model can be built. When modeling, you can create it using the programs identified in the second section above. First, the parameters are written into the input file in the form of a matrix, and then APDL calls the input file according to the written command process to complete the modeling. In this case, the finite element simulation is performed with SHELL181 elements, and the initial mesh is 5.

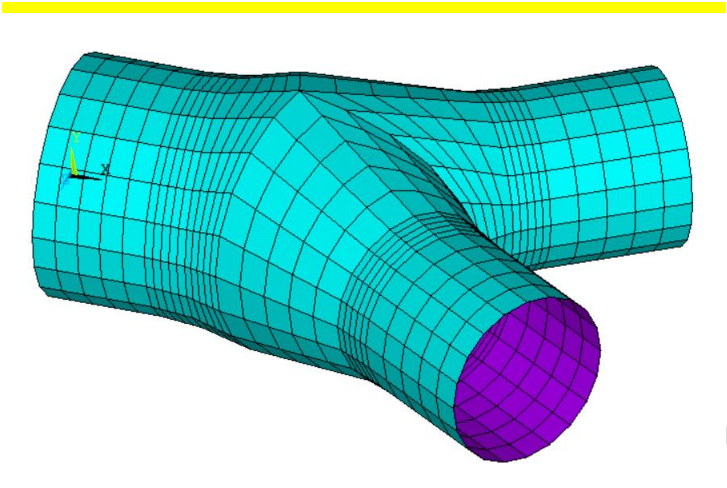


Figure 4 Setting the meshing SHELL181 element

Based on the size, the initial grid number for each region is determined to be 5. The final model setup result is shown below, with a maximum stress of 303.10Mpa.

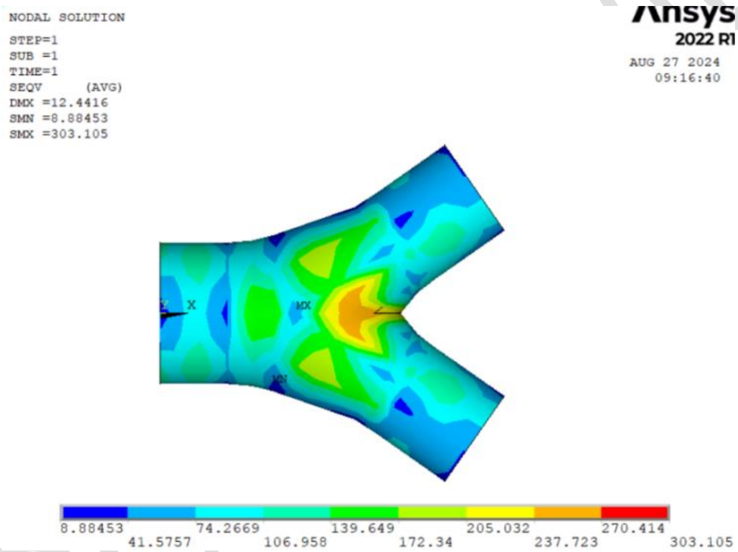


Figure 5 Stress cloud diagram of steel turnout pipe

Once the maximum stress is generated, you need to create a OUTDATA.TXT file as an output file to ensure that MATLAB can invoke loops. Once the setup is complete, the model is processed to explore the effect of different mesh numbers on the maximum stress: the mesh number is initially set to 5 to 50, and then 5 meshes are divided according to each region in the middle, resulting in 10 mesh number data. After calling the inputs in turn, the relationship between the maximum stress and the mesh size is obtained. For details, please refer to Table 1 for the relationship between maximum stress and mesh size :

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

Represent the table no 1 as an image in MATLAB to get the line chart from figure :

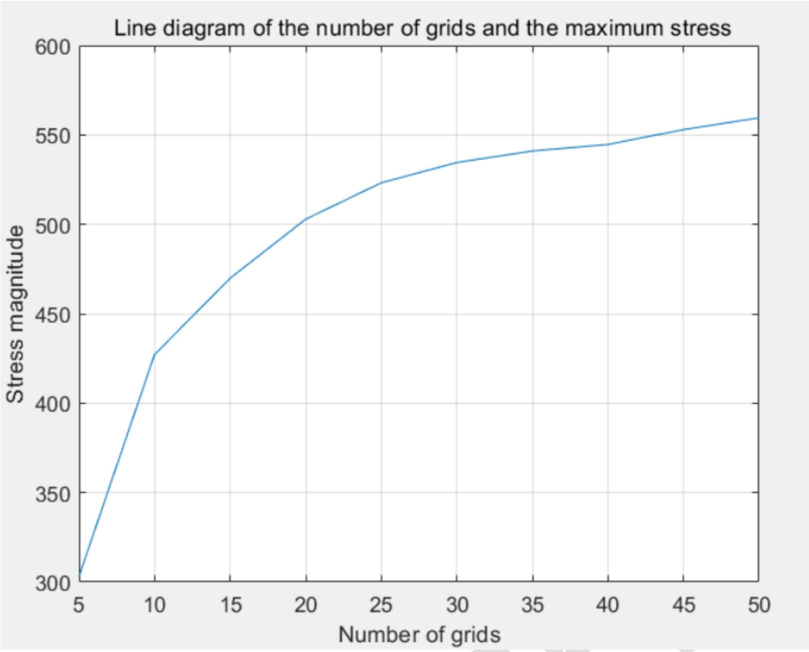
5. 

Figure 6 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 Disclaimer

This paper was completed by the authors from beginning to end, and the software used in it is ANSYS2022Ra and Matlab2022Ra. Line charts are also drawn using Matlab's own drawing software.

Disclaimer (Artificial intelligence)

Option 1:
Author(s) hereby declare that NO generative AI technologies such as Large Language Models (ChatGPT, COPILOT, etc.) and text-to-image generators have been used during the writing or editing of this manuscript.

7 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.