Stress Analysis of Pressure Vessels Based on ANSYS

[1]Jiangui Sun, [2]Jiaxu Xin and [3]Yu Wang
[1,2,3]College of Mechanical and Electronic Engineering, Shandong University of Science and Technology, Qingdao, China

Abstract-- With the continuous development of society, pressure vessel applications are becoming more and more widespread. Traditional analytical calculation methods have become increasingly unable to meet the needs of today's social development. Moreover, today's pressure vessels are moving toward large-scale and complicated, and the use of traditional methods for some special structures cannot solve the problem at all. At this time, the finite element analysis software ANSYS provides a basic idea for solving these problems. This paper is based on ANSYS to calculate the finite element strength of a company's equipment, and then verify that the design requirements are met under certain conditions. Through the finite element analysis software, the deficiencies of the traditional design method are made up, the stress value of the equipment is calculated accurately, and then compared with the standard, the rationality of the equipment design scheme is determined, and the design scheme is directly used for production and the production efficiency is improved.

Keywords-- *ANSYS,large size ,complexity,calculation analysis*

I. INTRODUCTION

At present, pressure vessels are not only large in number, but also growing rapidly. With the continuous development of society, pressure vessels are developing in the direction of large-scale and complicated. If the existing special equipment is calculated and analyzed according to the previous traditional methods, and then the basic scheme that meets the design requirements is designed, the design scheme may be too conservative, resulting in waste of equipment materials and increasing manufacturing costs. Therefore, the traditional analysis method can not meet the requirements of the current design method, and it is even more impossible to meet the precise analysis of some problems. The emergence and application of the finite element design method has brought a historic breakthrough to the pressure vessel industry. ANSYS is a finite element application software used in many fields such as geometry and meshing, multiphysics, fluid dynamics, structural mechanics, nonlinear structures, simulation processes, data management, and display dynamics. This paper uses ANSYS to analyze and calculate the equipment through an instance of a company.

Pressure vessels are extremely versatile and are a special type of pressure-bearing special equipment that is indispensable in industrial production. In the refining, chemical, fertilizer, pesticide, organic synthesis and other industries, pressure vessels are the main production equipment. For example, in an ethylene plant with an annual output of 300,000 tons, pressure vessels account for about 35% of the total equipment.

II. THE ANALYSIS AND CALCULATION OF EQUIPMENT

Using the finite element method, ANSYS software is used to analyze the stress distribution characteristics of a given pressure swing adsorber easily by creating a model, meshing, applying loads and constraints, and then calculating its fatigue life to ensure its safety. Optimize product structure and reduce production costs under the conditions.

A. Device parameters

The equipment analyzed in this paper mainly includes the cylinder, the supporting beam and the support. As a pressure vessel, the structure of the equipment has strict requirements, and the discontinuous structure is avoided as much as possible, and the welding of the cylinder body, the supporting beam and the support is smooth. The material of the equipment cylinder 304, the support and the support frame are all Q235B.

The structural diagram of the analysis equipment is shown in Figure2-1. The basic design data is shown in Table 2-1, and the material performance parameters are shown in Table2- 2.

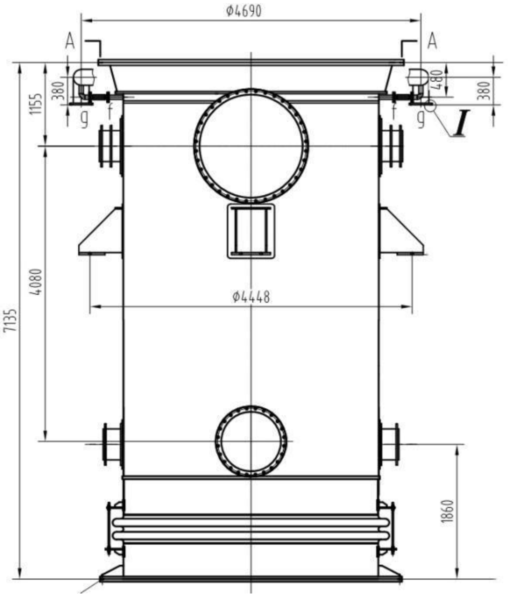

Figure 1: Schematic diagram of equipment structure

Table 1: Equipment basic design data

	Cylinder
material	304
design pressure	40KPa
Working pressure	25KPa
set tempreture	650℃
weight	52683kg

Table 2: Equipment material performance parameters

Component	material	Allowable stress at design temperature S_m(MPa)
Cylinder	304	85
Support	Q235B	75
Support steel beam	Q235B	75

B. Finite element model

The process of establishing a finite element model is called finite element modeling, which is the key link of the whole finite element analysis process. The rationality of the model will directly affect the accuracy of the calculation results and the length of calculation time. In order to simplify the finite element calculation, and at the same time ensure the accuracy of the analysis, and then neglect the structure that has little effect on the analysis results, only the cylinder, the support and the support beam are drawn. During the strength assessment process, stress linearization is required to determine the primary stress (Pm, Pl, Pb) and secondary stress (Q) for evaluation of the results. In the stress linearization, the path selection principle is: the stress linearization path is set by the node with the largest stress intensity in each part and the shortest direction along the wall thickness direction, and the path is set along the wall thickness direction of the relatively high stress area. The specific three-dimensional model of this device is shown in Figure2- 2.

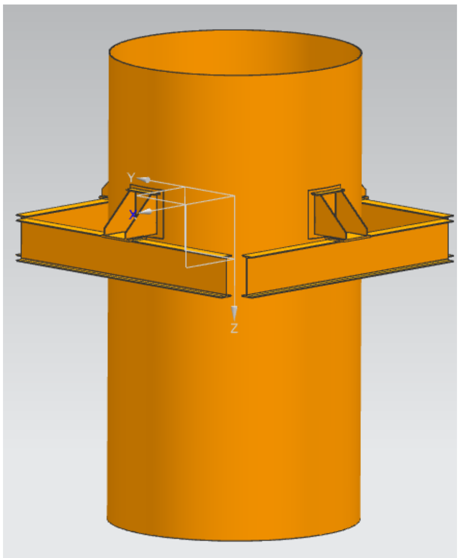

Figure 2: Three-dimensional model

C. Grid division

In order to make the above established model into a finite element, depreciation requires the division of the grid, and is divided into grids for finite element solution. The meshing of the model is an important step in the finite element numerical simulation analysis, which directly affects the accuracy of the subsequent numerical calculation analysis results. Meshing involves the shape of the element and its topology type, cell type, mesh generator selection, mesh density, cell number, and geometry voxels. In terms of geometric expression, the beam and the rod are the same, which is different from the physical and numerical solution. Similarly, the element solving equations for plane stress and plane strain conditions are also different.

The division of the grid can be divided into a free grid, a mapped grid, and a sweep grid. The properties of the defined grid are mainly to define the shape and size of the unit. The unit size is basically defined on the line segment, and can be divided by the number of line segments or the length of the line segment, which can be declared immediately after the online segment is created, or declared one by one after the completion of the entire entity model. The meshing process can also be carried out step by step, that is, the solid model object is completed at a certain stage and the mesh is spoken. If the obtained result is satisfactory, other objects are continuously built and meshed.

In the meshing, the hexahedral meshing method is adopted, and the total number of nodes is 613,614, and the number of meshes is 202,467. The meshing situation is shown in Figure 2-3.

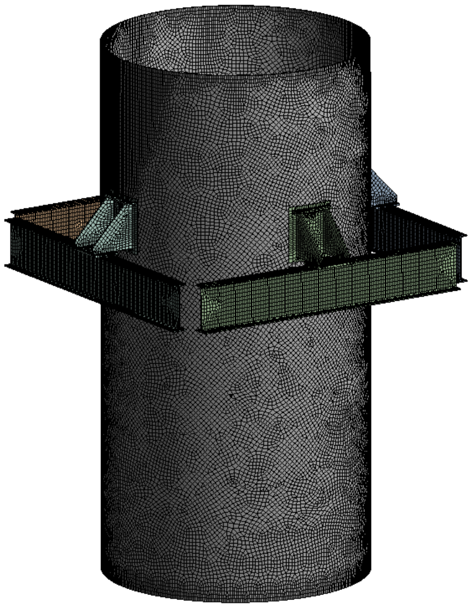

Figure 3: Meshing situation

D. Load and displacement conditions

The forces and or constraints imposed are called boundary conditions. Applying load is an important step in finite element analysis. Different types of analysis are almost always based on loads (such as force, temperature, speed, etc.) to study the changes of the system under load. For example, structural static analysis is used to solve the displacement, stress and force changes caused by external loads. Structural dynamics analysis is used to solve the influence of time-varying loads on structures or components, mainly considering the force load over time and its Damping and inertia effects. Structural Nonlinearity Analysis Structural nonlinearities cause the response of a structure or component to vary disproportionately with external loads.

Considering the actual working conditions, it is necessary to add the weight load at the upper end of the equipment. In order to be practical, the density of the cylinder was modified to 47,761 kg/m^3, so that the weight of the cylinder became 52,683 kg, and this gravity equivalent was applied to the cylinder, and the gravitational acceleration was taken as g=9.806 m/s^2.

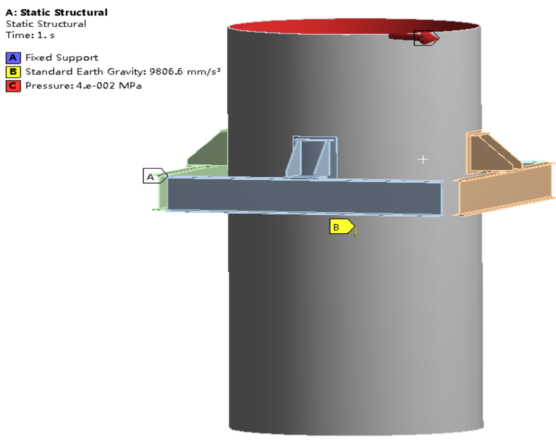

Figure 4: Load and displacement constraint application diagram

The pressure on the inner cylinder surface, p = 40 KPa. A fixed constraint is imposed on the end of the supporting steel beam of the device. The specific load and displacement conditions are applied as shown in Figure 2-4.

III. EQUIPMENT STRENGTH ASSESSMENT

A. The difference between conventional design and stress analysis design

Conventional design treats the "maximum load" that the container is subjected to at a single applied static load, without involving fatigue, and without considering thermal stress. While the container is in operation, there are not only mechanical loads, but also thermal stress, and there will be different operating conditions. In this way, it cannot be solved simply by increasing the material design factor or increasing the thickness. Sometimes the thickness increase may also be counterproductive. In the analysis design, thermal analysis can be taken into account, and for multi-operational operation, fatigue analysis can be used to predict its life.

Conventional design basically controls the average stress of the structure and limits it to a certain range. This does not take into account local high stress areas, which are often sources of damage. To identify and evaluate these high stress zones, an analytical design is required. In addition, the conventional design is applicable to a specific structural form, and is not applicable to other structural forms or other loads. In the analysis design, as long as the boundary conditions are properly set and the model is properly processed, the stress distribution of each part of the structure can be obtained.

The design criteria for conventional design are based on the first strength theory, based on the elastic failure criterion, which is considered to be a failure when the maximum tensile stress exceeds the allowable stress. The design criteria for analytical design are based on the third strength theory, based on the elastoplastic failure criterion, when the maximum shear stress exceeds the allowable stress.

B. Evaluation of finite element analysis results of equipment

The stress analysis results and strength of the above structure were evaluated as follows. In the strength evaluation process, the selection principle of the stress linearization path is to set the stress linearization path through the nodes with the highest stress intensity in each part and the shortest direction in the wall thickness direction, and set the path along the wall thickness direction for the relatively high stress area. .

The results of the stress analysis are shown in Figure3-1. It can be seen from the figure that the maximum stress point of this part is located at the joint between the support and the cylinder, and the maximum stress value is 602.86 MPa. The stress analysis results were evaluated according to the requirements of JB4732-1995 (2005 confirmed version), and the strength evaluation is shown in Table 3-3.

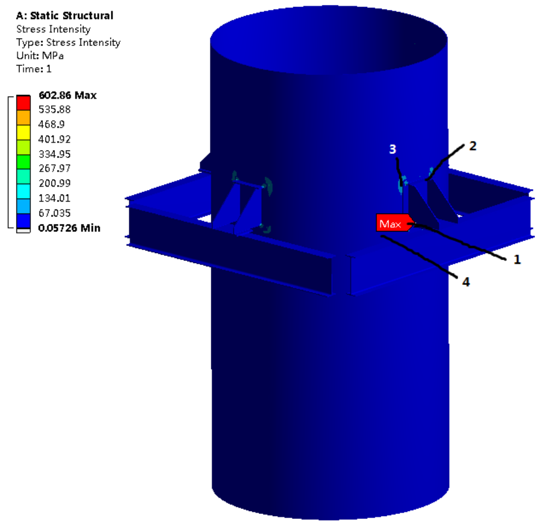

Figure 5: Stress distribution cloud

Table 3: Stress intensity assessment

Stress intensity and combined stress intensity	path	material	Stress intensity calculation(MPa)	Stress intensity allowable limit(MPa)	Evaluation Results
Primary partial film stressS_{II}	1	Q235B	14.087	$1.5S_m^t$=61.5	qualified
Primary + secondary stress intensityS_{IV}			17.569	$3S_m^t$=123	
Primary partial film stressS_{II}	2	Q235B	13.193	$1.5S_m^t$=61.5	qualified
Primary + secondary stress intensityS_{IV}			18.628	$3S_m^t$=123	
Primary partial film stressS_{II}	3	304	18.069	$1.5S_m^t$=63	qualified
Primary partial film stressS_{II}	4	Q235B	20.033	$1.5S_m^t$=61.5	qualified
Primary + secondary stress intensityS_{IV}			39.17	$3S_m^t$=123	

CONCLUSION

This paper first studies the development history and current situation of the finite element analysis method, and briefly describes the analysis design of the current pressure vessel, and then describes the advantages of the finite element analysis method compared to the traditional analysis design method. Through the actual case of a company to introduce the basic process of ANSYS finite element analysis, the establishment of finite element model, the division of the model, the application of constraints and the evaluation of stress analysis results are introduced. This is the whole process of the finite element method applied to the analysis and design of the pressure vessel. It is also a good compensation for the traditional analytical design method, ensuring the accuracy and high efficiency of the analysis design, reducing the manufacturing cost and improving the production efficiency.

ACKNOWLEDGEMENTS

I would like to thank my teachers and classmates for their help. When I encounter problems that I can't solve, they can always give me an accurate explanation to help me continue to complete the thesis. I also want to thank my family, who always support me silently behind me and give me spiritual help.

References

[1] Huang S C, Huang Y M, Shieh S M. Vibration and stability of arotating shaft containing a transerse crack[J].Sound andVibration,1993,162(3):387-401.

[2] Jiahu Huang,Deding Lu.Stainless Steel for Low Temperature Pressure Vessels (II)[J].Pressure Vessel,2014,31(06):1-14.

[3] Sunyi Chen.Several engineering factors worthy of attention in finite element analysis modeling of pressure vessels[J].Pressure Vessel,2015,32(05):50-57

[4] Huan Li,Ye qing WE.Finite Element Analysis of Wall Thickness of Pressure Vessel[J].Journal of Yunnan Agricultural University,2014,29(04):549-552.

[5] Haiyong Bai,Yongli Fang.Application of ANSYS Ultimate Load Analysis Method in Pressure Vessel Design[J].Pressure Vessel,2014,31(06):47-50.

[6] Guodong Ren,Chun yuan Kong,.Multivariable Optimization Design of Pressure Vessels Based on ANSYS[J].Mechanical Engineering & Automation, 2010(01):68-70.