Simulation Research of a type of Pressure Vessel under Complex Loading
Part 2 Complex Load of the Numerical Analysis

Junchen Li[1,2]

1.College of Materials Science and Engineering,
Lanzhou University of Technology;
2.State Key Laboratory of Gansu Advanced Non-ferrous Metal Materials,
Lanzhou University of Technology
Lanzhou, 730050, China
E-mail: lijunchen605@163.com

Jie Sheng

College of Materials Science and Engineering,
Lanzhou University of Technology
Lanzhou, 730050, China
E-mail: shengj605@163.com

Yufu Zhang[1,3]

1.College of Materials Science and Engineering,
Lanzhou University of Technology;
3.Gansu Lanpec Technologies Limited
Lanzhou, 730070, China
E-mail: zhangyufu@lanpec.com

Abstract—**Loading of pressure vessel was usually complicated in practical service operating conditions. Simulation model of pressure vessel was built by method of finite element simulation analysis, and structured mesh generation was realized. Numerical calculation was come true, stress/strain distribution of pressure vessel was obtained in applying of the multi-load. On this basis, this condition compared with alone applied many loadings. The calculation results indicate the validity of this model, and results are evaluated according to relevant standards, which provide a way to study mechanical response in the actual working conditions. In addition, sub-model is analyzed for key part of pressure vessel, and transition is come true from large scale simulation to small scale simulation.**

Keywords- pressure vessel; finite element; simulation; numerical analysis

I.   INTRODUCTION

In recent decades, with the development of economy and the change of energy structure, pressure vessel has become a kind of special equipment widely used in production and life[1]. Because the pressure vessel is generally going to dress gas or liquid and at the same time loading a certain pressure, which determine that its service condition is more complex, especially used in nuclear power, chemical industry, boiler and other areas of pressure vessel. It often works in extreme complex conditions[2]. It is difficult for scientists to use analytic method analysis and design in this kind of service condition of pressure container; therefore they must find other ways. Pressure vessel simulation analysis and design is a close product of computer and mechanics and material science. It can not only solve the problem that conventional design can not but also represents the modern pressure vessel design of advanced level[3]. The simulation research on the computer can reappear as service under the condition of pressure vessel mechanical response and assess its safety as well as optimize the design scheme.

This study is using the macro-local mechanical response in the complex load of finite element simulation software ABAQUS calculation pressure vessel. In this process, using sub model related macro and local response, gradually inheriting boundary condition, which realizes the transition from the simulation of large scale to small scale. This paper is the numerical analysis part in the complex load of a pressure vessel, introducing the pressure vessel from the simulation modeling to the grid division as well as the numerical calculation of mechanical response with common load of self-respect, hydraulic, uniform internal pressure. Comparison proves the accuracy of the simulation calculation results under many loads at the same time and evaluates the simulation calculation result according to the relevant standard. On the other hand, on the basis of analysis results in the global model, the sub model technique is adopted to calculate more accurate stress/strain response of the pressure vessel site, which created the premise condition for the safety evaluation of pressure vessel containing defects in the next step in a small area preset defects such as porosity, slag.

II.   A PRESSURE VESSEL GEOMETRY MODELING

Main geometric parameters of pressure vessel are shown in Tab.1, in which geometric parameters are not on this list because of the large amount. Pressure vessel materials are composed by the 2205 dual phase steel with inner thickness of 4mm and the16MnR outer steel with the thickness of 80mm. In order to get more effective numerical calculation results, the weld on pressure vessel can also be seen as materials with different attributes. The concrete material properties are listed in Tab.2. Tab.3 presents design parameters in pressure vessel.

TABLE I.   PRESSURE VESSEL GEOMETRY PARAMETER

Parts	Structure parameters	size（mm）
Barrel	inside	1600
	Barrel length	8000
	Thick of simulated	80+4
Head	Ellipsoid head long radius	800
	Ellipsoid head short radius	400
	Straight side length	50

TABLE II.   PRESSURE VESSEL DESIGN PARAMETERS

Design pressure (P,Pa)	Design temperature (T,℃)	Barrel density (ρ,kg/m³)	Barrel quality (M,t)	Container volume (v,m³)
14e6	70	7930	34.45 (ABAQUS output)	17.36

TABLE III.   PRESSURE VESSEL MATERIAL PARAMETERS

Material	SAF2205	SAF2205 weld	16MnR	16MnR weld
Density (t/mm³)	7.93e-9	7.93e-9	7.85e-9	7.85e-9
Elastic modulus (GPa)	195.8669	213.2719	209	209
Poisson's ratio	0.3	0.3	0.28	0.28
Yield strength (MPa)	625	716	345	450
Tensile strength (MPa)	824	904	535	545

According to the above geometric parameters and the use of fission merger modeling method, the finite element software ABAQUS was established in pressure vessel macroscopic geometric model, as the geometric model is shown in Fig.1. In addition, the different part of the pressure vessel material, according to the material parameters list, defines corresponding material properties.
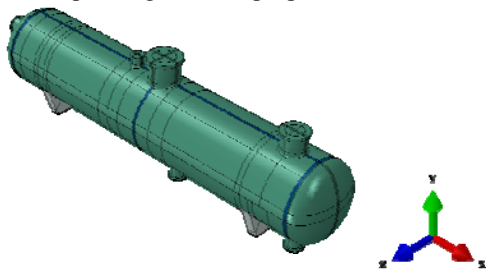

Figure 1.   Geometric model of pressure vessel

III.   THE DIVISION OF FINITE ELEMENT MESHES

Grid partition is a very important content in the finite element calculation. The stand or fall of meshing directly determines whether the calculation results are accurate or not. When parting this pressure vessel by mesh division, the first pressure vessel needs to disseminate seeds in whole. In order to calculate accurately, it requires that all grid use hexahedron units. The reason why grid division way uses structured grid and scanning grid is that the structured grid and scanning for hexahedron mesh unit analysis precision is higher. Before Specific mesh, we need to make use of Medial Axis algorithm to the division of geometric model cutting which is divided into some simple shape, and then use structured grid division mesh technology so that we can get rules of the hexahedral element. Before Specific mesh, we need to make use of Medial Axis algorithm to the division of geometric model cutting which is divided into some simple shape, and then use structured grid division

mesh technology so that we can get rules of the hexahedral element. After meshing, grid quality can be checked specially for solid, geometric area or unit in ABAQUS. Checking and analysis can lead to unit of mistakes or warning information which will be displayed with high brightness will tell the percentage of the total amount of grid. The effectiveness of the grid can be further judged through checking the grid quality[4]. The finite element mesh model of pressure vessel is shown in Fig.2.
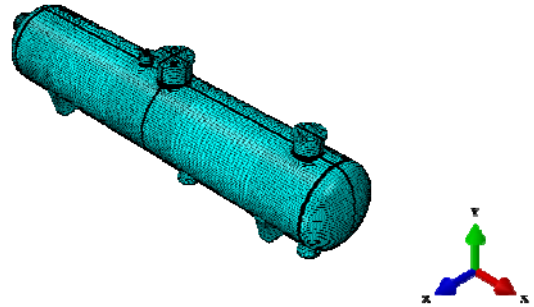


Figure 2.   Grid partition of pressure vessel

IV.   CALCULATION OF PRESSURE VESSEL UNDER MANY LOAD AT THE SAME TIME UNDER THE ACTION OF MECHANICAL RESPONSE

In this paper, the part1 has introduced the mechanical response of pressure vessel under the separate action of the three kinds of load of the self-respect, hydraulic, uniform pressure inside, therefore the validity of the finite element mesh has also been proved; on the other hand, load and the settings of boundary conditions have decided the accuracy of numerical calculation of simulation model. Here, according to the definition of different ways, these three loads of the self-respect, hydraulic, uniform internal pressure are loaded to simulation model of the same pressure vessel at the same time to realize the settings of the load and boundary conditions. Fig.3 is a plan of many loads loading. The yellow arrow in schematic diagram is on behalf of the dead weight load which needs to define the material density and the parameters of acceleration of gravity. Pink arrow represents the pressure which is divided into two parts: one is the uniform internal pressure of 14MPa, and the other is the defined hydraulic according to the function expression P=ρgh in which ρ=1000kg/m³, g=9.8N/kg.

The following is the calculation results of finite element. Fig.3(a) is the Mises stress distribution under many loads of pressure vessel. Fig.3(b) is a displacement profile of many loads of pressure vessel. According to the calculation results, it can be found that the largest position of stress appeared in the welding position of the container body and the biggest nozzle, which is inconsistent with the actual situation, while more detailed analysis need to use the sub-model technology.
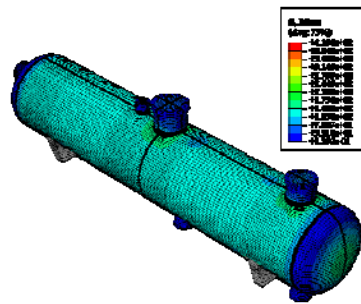
(a) Mises stress distribution
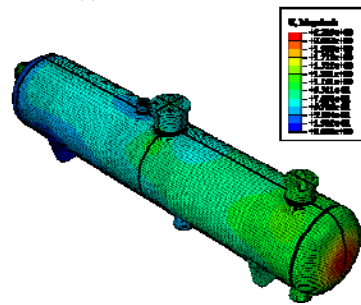
(b) Displacement distribution

Figure 3. Calculated results of multi-load applied at the same time

In order to validate the correctness of the numerical calculation results of simulation model with many loads, as shown in Fig.4, three typical parts of the node were extracted in the simulation model of pressure vessel and their stress data in different directions were saved respectively. In the same way, stress data of these nodes were extracted from the calculation results in a separate applied self-respect, hydraulic, uniform internal pressure load and two sets of data are compared.
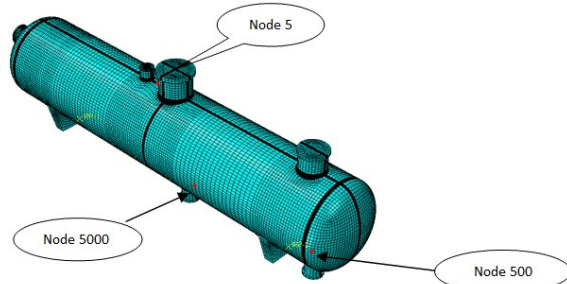

Figure 4. Three typical parts of the node on pressure vessel

Tab.4, Tab.5 and Tab.6 recorded the stress data of three nodes, in which load S11 represents σ_x, S22 represents σ_y, S33 represents σ_z, S12 represents σ_{xy}, S12 represents σ_{xz}, S12 represents σ_{yz}, shows that the sum of each direction stress of pressure vessel with a respective action of the self-respect, hydraulic equals to the stress calculation of many loads, which shows that the loading way of many loads and the calculation result are correct. From the table we can see that internal pressure plays a major role for the stress distribution of pressure vessel.

TABLE IV.   STRESS DATA OF NODE 5

Load	S11 (MPa)	S22 (MPa)	S33 (MPa)	S12 (MPa)	S13 (MPa)	S23 (MPa)
self-respect	-4.77606 E-01	2.39027 E-03	-4.11930 E-01	-1.32270 E-03	-1.99773 E-03	-2.82142 E-03
hydraulic	-2.15751 E-01	1.08691 E-03	-1.33821 E-01	1.04927 E-04	-1.27878 E-03	1.03137 E-03
Uniform internal pressure	7.20925 E+01	-1.29804 E+00	1.38743 E+02	1.20380 E+00	-3.15283 E-01	5.95382 E-02
The sum of the three	7.13991 E+01	-1.29456 E+00	1.38197 E+02	1.20258 E+00	-3.18560 E-01	-6.33910 E-02
Multi-load	7.13991 E+01	-1.29457 E+00	1.38197 E+02	1.20258 E+00	-3.18559 E-01	-6.33910 E-02

TABLE V.   STRESS DATA OF NODE 500

Load	S11 (MPa)	S22 (MPa)	S33 (MPa)	S12 (MPa)	S13 (MPa)	S23 (MPa)
self-respect	1.77142 E-02	2.05749 E-03	1.05335 E-01	1.59074 E-02	-4.03760 E-02	-3.57432 E-02
hydraulic	2.73195 E-03	3.17641 E-02	8.60558 E-02	4.84395 E-03	-4.17462 E-02	-1.21694 E-02
Uniform internal pressure	1.45655 E+00	4.07746 E+01	6.26839 E+01	6.44798 E-01	-3.1832 E+01	-2.5521 E+00
The sum of the three	1.47700 E+00	4.08084 E+01	6.28753 E+01	6.65549 E-01	-3.1914 E+01	-2.6000 E+00
Multi-load	1.47700 E+00	4.08085 E+01	6.28753 E+01	6.65549 E-01	-3.1914 E+01	-2.6000 E+00

TABLE VI.   STRESS DATA OF NODE 5000

Load	S11 (MPa)	S22 (MPa)	S33 (MPa)	S12 (MPa)	S13 (MPa)	S23 (MPa)
self-respect	-1.92796 E-02	1.59931 E-02	-1.09136 E-01	1.58535 E-02	-3.30083 E-02	3.26188 E-02
hydraulic	-2.05521 E-02	3.38608 E-03	-4.21080 E-02	8.78288 E-03	-3.83436 E-03	1.84029 E-02
Uniform internal pressure	-7.09571 E+00	1.18356 E+01	1.59488 E+01	-5.02157 E-01	1.14686 E+01	8.28717 E-02
The sum of the three	-7.1355 E+00	1.18550 E+01	1.57976 E+01	-4.77521 E-01	1.14318 E+01	1.33893 E-01
Multi-load	-7.1355 E+00	1.18549 E+01	1.57975 E+01	-4.77520 E-01	1.14317 E+01	1.33893 E-01

V.   CALCULATION RESULTS EVALUATION

This paper evaluated the results of ABAQUS finite element simulation according to the stress limit in JB4732-

1995(steel pressure vessel-standards of analysis and design)[5]. The calculation results of the stress near nozzle in pressure vessel were extracted in ABAQUS, in which Pm=133.3MPa, Pl=200MPa, the allowable limit of chosen material [σ]=196MPa according to the standard. Calculation is obtained:

1) A general membrane stress

P_m=133.3MPa < [σ]=196MPa

2) A local film stress

Pl=200MPa < 1.5[σ]=294MPa

3) Sum for stress and bending stress

230MPa < 1.5[σ]=294MPa

4) Sum for stress and secondary stress

230MPa < 1.5[σ]=294MPa

The calculation results show that the stress simulation calculation value of pressure vessel meets the standard, although the maximum stress on open hole near nozzle of pressure vessel can lead to the local yield of pressure vessel[6]. While the overall of pressure vessel is still in the elastic stage, therefore, the local yield near nozzle will not lead to the failure of pressure vessel, which is acceptable.

VI. USE SUB-MODEL CALCULATION PRESSURE VESSEL KEY PARTS MECHANICS RESPONSE

Because of various requirements of technology or structure, it needs to open hole or install nozzle on pressure vessel, such as opening, cleaning hole as well as loading and unloading port etc. After the opening, on the one hand, the wall material is weakened, which will cause the increase of stress or the weakness of container strength; On the other hand, the structure continuous place be destroyed, which will produce larger additional bending stress in the open hole and nozzle the office. As a result, stress could reach a big value in local area in the open hole and nozzle. With such a large local stress and the stress caused by the external load on nozzle, additionally various factors of material and manufacturing defects' common effect, open hole and the part of nozzle will become a weak part of pressure vessel[7]. In order to calculate a more accurate stress/strain response in the part of nozzle on pressure vessel, the sub-model technique and the refined grid are adopted to make a further analysis for the part of nozzle on pressure vessel on the basis of analysis results in the global model.

Sub-model of pressure vessel nozzle is set up according to the original size, in which, when inheriting boundary condition, if the boundary conditions, loading, contact and constraint which play a role originally in the global model are located in sub-model area, then it remains unchanged in child model. If it is located outside of sub-model area, it will no longer appear in child model. The sub-model diagram as is shown in Fig.5.
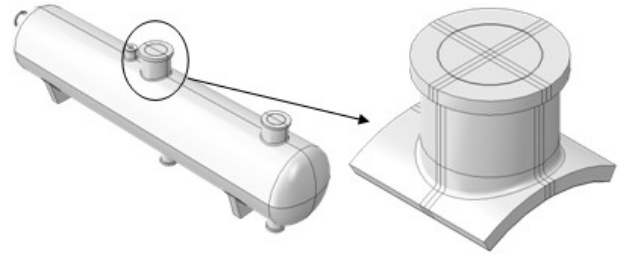

Figure 5. Sub-model of pressure vessel nozzle

The results displacement between the sub-model of pressure vessel and global model inherited by four parting plane connected with the container body, when inheriting boundary conditions, the red lines surrounding areas as shown in Fig.6, still set the uniform internal pressure for the inner wall of nozzle and the load of self- weight is loaded on the whole nozzle.
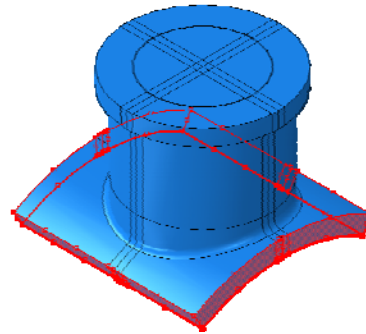

Figure 6. Boundary conditions for sub-model of pressure vessel nozzle

The following is the calculation results of finite element. Fig.7(a) is the Mises stress distribution of the child model. Fig.7(b) is the Mises stress pattern on nozzle of the overall model. Fig.7(c) is the displacement distribution of the child model. Fig.7(d) is the displacement profile on nozzle of the overall model. From the calculation results of Mises stress and displacement of the calculation of child model, it can be seen that the displacement results on sub-model and that on the nozzle of whole model are completely the same, which verifies the inheritance of the conditions on sub-model boundary is right. There is a very small gap in Mises stress because of the interpolation and average of Mises stress on the calculation nodes, which almost has no influence on the quantitative analysis results. Sub-model refines the local grid and calculates a more accurate result, and also realized transition from the simulation of large scale to small scale, which proves that the son model can be the bridges of research of macro simulation and local simulation of setting up pressure vessel. For the next step in a small key area, the realization of the technique of sub-model created the premise condition for presetting defects such as porosity, slag and evaluating the safety of pressure vessel containing defects. Because space is limited, this part of content is introduced by another text.
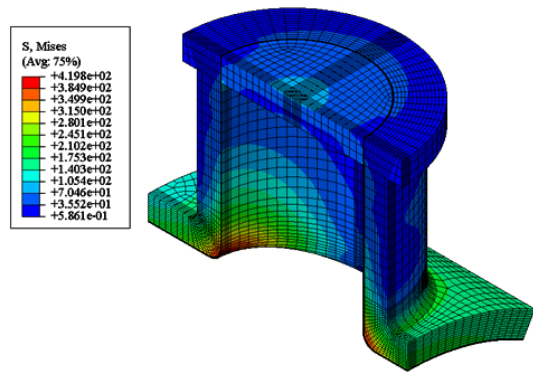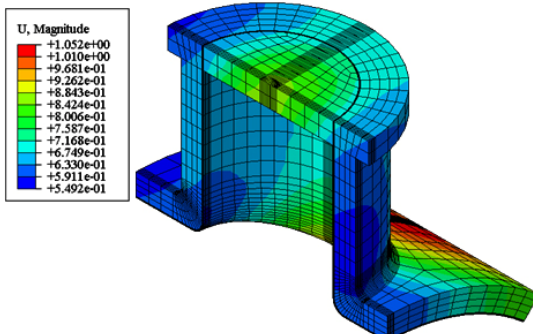
(a)Mises stress of sub-model
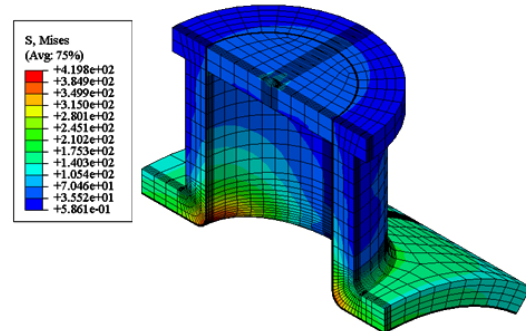
(b) Mises stress of the overall model connecting pipe
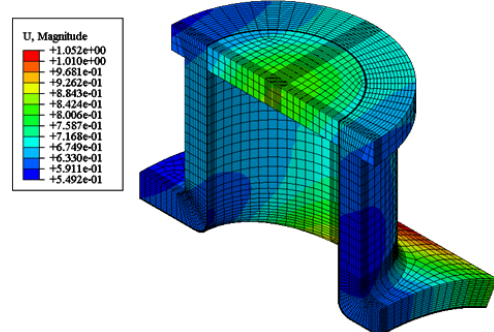
(c) Displacement of sub-model

(d) Displacement of overall model

Figure 7.   Comparing with calculation results between sub-model and overall model

VII.   RESULTS

Based on the analysis method of finite element simulation, the numerical calculation has been realized under the condition of many applied load at the same time for a certain type of pressure vessel, therefore the stress/strain distribution of the simulation model is acquired. In order to validate the reliability of the computation under many loads at the same time, the node stress data has been extracted in several typical parts of the simulation model. At the same time, these nodes stress data have been extracted and calculated from calculation results of self-respect, hydraulic, uniform internal pressure load separately applied, then analyzed with a same result between two data, which illustrates that the loading way of many loads and the calculation result are correct.

According to the relevant standards, the simulation results of ABAQUS finite element can be assessed. The results show that the stress/strain distribution on pressure vessel conforms to the standard, although the maximum stress on the open hole over of pressure vessel can lead to the local yield of pressure vessel. However, the whole pressure vessel is still in the elastic stage, therefore the local yield nearby nozzle will not lead to the failure of pressure vessel.

The sub-model in the key parts between pressure vessel and nozzle has been analyzed, which realize the transition from the simulation of large scale to small scale. The realization of the technique of sub-model created the premise condition for presetting defects such as porosity, slag and evaluating the safety of pressure vessel containing defects.

ACKNOWLEDGMENT

REFERENCES:

[1]   Deng Zhi-wei. Analysis on the operating conditions and material requirements of the pressure Pipe[J]. Pipeline Technique and Equipment, 2009(4): 18-20.

[2]   Liu Jia-ping, Li Xiao-jing. Analysis for defects assessment standard of designing pressure vessel under the extreme conditions[J]. New technology and new process, 2011(2): 18-21.

[3]   Shen Jun. Limit load analysis application in pressure vessel analytical design[J]. Petro-chemical equipment,2011,40(4):35-38.

[4]   Shi Yi-rong, Zhou Yu-rong. ABAQUS finite element analysis example explanation[M]. China machine press, 2006.

[5]   JB4732-1995, Steel pressure vessel—analysis design standard[S].

[6]   Du Si-hong, Zhang Yan, Liu Hai-hong, Yuan Zhen-wei, Wang San-bao. Pressure vessel opening over the finite element analysis[J]. Petroleum and chemical equipment, 2010, 13(4):25-27.

[7]   Lu Zhi-min, Li Qiang-xie, Yun Ye. Finite element analysis of strain at nozzle junction of cylindrical pressure vessel[J]. Journal of Beijing jiaotong university, 2008,32(4):36-39.