Study on the Blowing Scheme of a Propellant Storage Vessel

Xu Zhigao [a], Liu Xinguo and Ma Jing

Xi′an Research Institute of High Technology, Hongqing Town, Xi′an 710025, China

Abstract. The blowing flow field of a propellant storage vessel are simulated using Fluent software. Through steady flow field calculation, some important flow field parameters, such as pressure, temperature, velocity and Streamline, are obtained under three different working conditions. Based on these parameters, the effects of three different working conditions are analysed. According to the analysis, the improved blowing scheme is given, which can effectively improve the efficiency of the blowing process.

1 Preface

After storing propellant, a propellant storage vessel still will have a certain amount of residual propellant. The propellant has strong toxic and corrosive, which may cause certain damage. After the propellant releasing, it is very necessary to clean the propellant storage vessel. Cleaning is usually done with cleaning and drying, and the process requires a long blowing with a certain pressure of high temperature nitrogen.

In this paper, flow field simulation software is used to simulate the process of vessel blowing, and the steady flow field simulation is carried out. The vessel has upper and lower export and an import. By changing the export way of opening and closing, three conditions are formed, namely two open at the same time as well as the upper and lower export alternating open. Three kinds of simulation computation mode are operated, so as to obtain the important flow field parameters inside the vessel, such as pressure, temperature, velocity and streamline distribution. Based on the distribution and size of these parameters, the effect of the blowing and the causes of phenomenon are analyzed. Comparing the effect of blowing in different working conditions, and make recommendations on how to improve the disposal plan and improve the efficiency.

2 Vessel model

2.1 Physical model of the vessel

Solidworks software is used to construct the geometric model of a vessel. In order to facilitate the construction of the geometric model to facilitate the calculation of the previous understanding and analysis, the original geometric model was processed using the ANSYS DesignModeler module before the calculation, as shown in Figure 1.

Figure 1. Physical model of the vessel.

2.2 Computational grid model of the Vessel

After the construction of the geometry model of the vessel structure is completed, the model is imported into the Meshing software for grid division in the Workbench. Because there are many detail structures in the vessel, the mesh of the detail structure needs to be refined. This paper USES tetrahedral mesh to divide and divide, and finally the number of grid is about 8.9 million [1,2].

2.3 The model of the screen mesh

There is some screen mesh inside the vessel, which is woven from stainless steel filaments. There are many gaps in the middle of the mesh. The ratio of the gap area to the total area, referred to as the mesh opening rate. The specification of the mesh is divided by the opening rate.

In this paper, there are two specifications for the screen mesh used in the vessel. The opening rate of one screen mesh is 0.049, and the opening rate of another is 0.058. Refer to the practical resistance manual, using the calculation method given in this paper, the resistance coefficient of the two screens is 378.6739 and 263.8391 respectively [3].

3 Simulation of vessel blowing process

*Corresponding author: [a] ayue113@163.com

The calculation setting of the blowing process simulation is as follows. (1) Steady-state unsteady simulation method is used; (2) The ideal gas model is used; (3) The pressure base coupling solver is used; (4) The second order windward format discrete control equation is used; (5) SSTK-ω the turbulent model to simulate the turbulent flow in the flow is used; (6) The porous media step model is used [4,5].

The porous medium step model in Fluent software is generally considered to be infinitely thin. It is assumed that the pressure drop of the porous medium step model is proportional to the dynamic pressure head of the fluid. After the loss coefficient is introduced, the empirical formula is obtained, namely, the relationship between the pressure drop of the porous medium step model Δp and the normal velocity component through the screen mesh v is as follows:

$$\Delta p = \frac{1}{2} k_L \rho v^2 \qquad (1)$$

Where: ρ represents the density of the fluid, k_L denotes the loss coefficient, and is the dimensionless number.

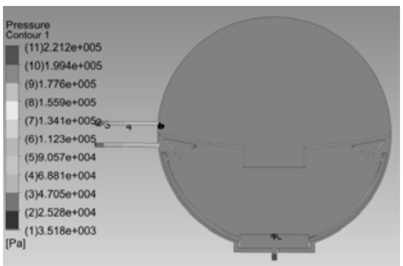
The boundary conditions are as follows: using nitrogen as the blowing medium, from a vessel entry to 0.25 MPa pressure, 55℃ constant temperature constant blowing. The fluid is set to the ideal gas. The outlet pressure is set to 0 and the temperature is set to the normal temperature of 26℃.

4 Analysis and optimization of simulation calculation results

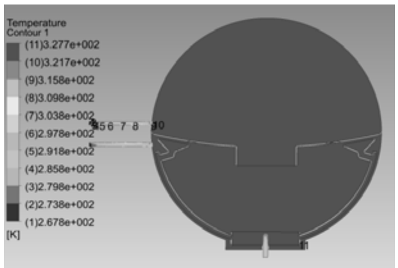
The above methods are used to calculate three working conditions. The results are as follows.

4.1 Working condition 1

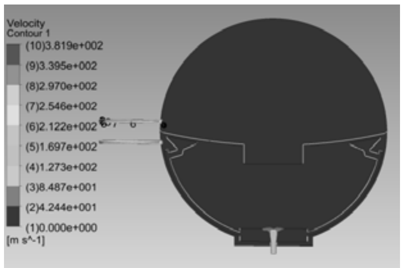
Under this condition, the upper and lower exits are opened simultaneously. Figure 2 is the key data of flow field, such as velocity, temperature and pressure of each part of the vessel.



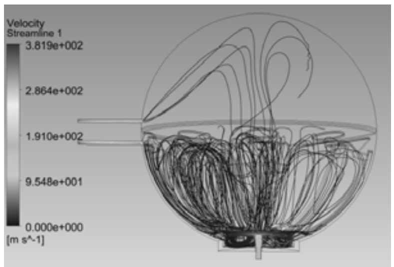
Pressure distribution



Temperature distribution



Velocity distribution



Streamline distribution

Figure 2. Calculation result of working condition 1.

4.1.1 The pressure distribution

The pressure distribution in the vessel is relatively uniform, and the pressure is between 0.48 MPa and 0.1951MPa, which are not very different, and there is no area of pressure mutation.

4.1.2 The temperature distribution

The temperature distribution in the vessel is relatively uniform, and the change is extremely small between 327K~327.7K.

4.1.3 The velocity distribution

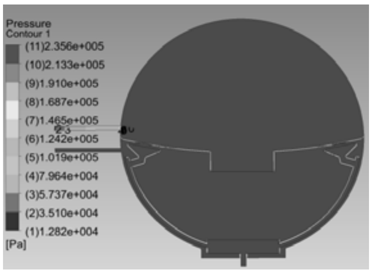
The vessel is divided into three regions: the import, the inner space of the vessel, the export, and the velocity between each region is very different. The inner space of the vessel is the smallest, with the maximum velocity of about 12m/s, and the minimum velocity is almost 0. The inlet velocity of the vessel is very high, reaching 167.9m/s. The maximum velocity of exports is 355.1m/s, even slightly higher than the local sound velocity. In addition, at the bottom corner, the cup body is close to the inner wall, and the middle bottom and upper part of the area are particularly low, which can be considered as the dead zone.

4.1.4 Streamline distribution

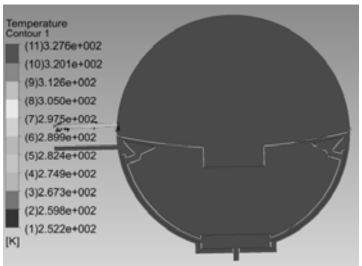
The flow field of the lower chamber of the vessel is more complex, which indicates that the gas can be blown to a large area in the lower chamber, while the flow field in the upper chamber is simple and only flows in a small area.

4.2 Working condition 2

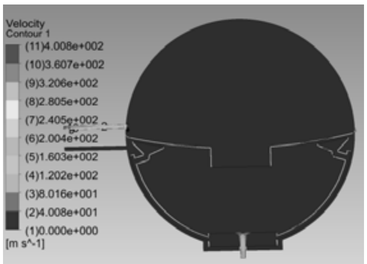
Under this condition, the upper outlet is opened, and the lower outlet is closed. Figure 3 shows the parameters of each flow field in the vessel under the working condition 2.



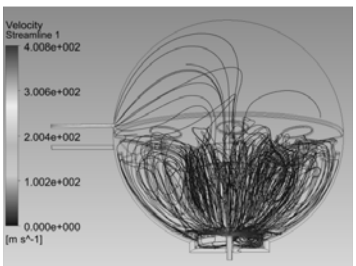
Pressure distribution



Temperature distribution



Velocity distribution



Streamline distribution

Figure 3. Calculation result of working condition 2.

4.2.1 The pressure distribution

The pressure distribution in the vessel is relatively uniform, and the pressure is between 0.2223MPa and 0.2227MPa, which is not very different, and there is no area of pressure mutation.

4.2.2 The temperature distribution

The temperature distribution in the vessel is relatively uniform, and the change is extremely small between 327k and 327.7 K.

4.2.3 The velocity distribution

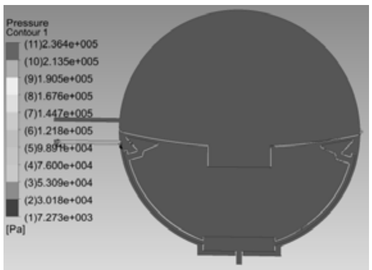
The vessel is divided into three regions: the import, the inner space of the vessel, the export, and the velocity between each region is very different. The velocity of the inner space of the vessel is the smallest, the maximum velocity is about 10m/s, and the minimum velocity is almost 0. The velocity of vessel inlet is very high, reaching 125.3m/s. The maximum velocity of export is 369.4m/s. In addition, at the bottom corner, the cup body is close to the inner wall, and the middle bottom and upper part of the area are particularly low, which can be considered as the dead zone.

4.2.4 Streamline distribution

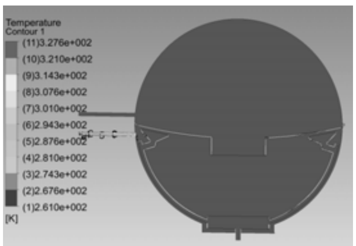
The flow field of the lower chamber of the vessel is more complex, which indicates that the gas can be blown to a large area in the lower chamber, while the flow field in the upper chamber is simple and only flows in a small area.

4.3 Working condition 3

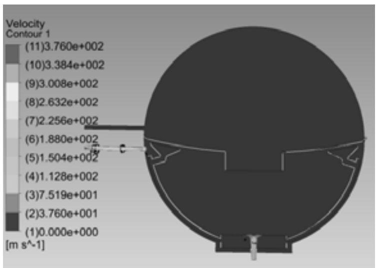
Under this condition, the upper outlet is closed, and the lower outlet is opened. Figure 4 shows the parameters of each flow field in the vessel under the working condition 3.



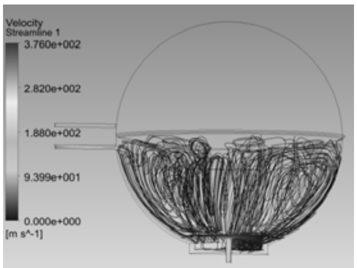
Pressure distribution



Temperature distribution



Velocity distribution



Streamline distribution

Figure 4. Calculation result of working condition 3.

4.3.1 The pressure distribution

The pressure distribution in the vessel was relatively uniform, and the pressure was between 0.2238MPa and 0.2242MPa and there was little difference in the pressure.

4.3.2 The temperature distribution

The temperature distribution in the vessel is relatively uniform, and the change is extremely small between 327k and 327.7 K.

4.3.3 The velocity distribution

The velocity of the vessels varies greatly between regions. The inner space of the vessel is the smallest, with the maximum velocity of 3m/s, and the minimum velocity is almost 0. The velocity of vessel inlet is very high, reaching 112.8m/s. The maximum velocity of exit is 375.9m/s. In addition, at the bottom corner, the cup body is close to the inner wall, and the whole upper chamber and other regions have a very low velocity, which can be considered as the dead zone.

4.3.4 Streamline distribution

Streamline flow field concentrated in inferior vena room, filled with almost the whole of inferior vena chamber, and upper chamber no flow, the flow chart shows that the gas in the inferior vena chamber can be blown to the area is larger. The chamber can think not form flow.

4.4 Comparison and analysis of calculation results

4.4.1 Temperature

Under the three working conditions, the internal temperature of the vessel is uniformly distributed, and the temperature between each working condition is basically the same.

4.4.2 Pressure

Under the three working conditions, the internal pressure of the vessel is uniformly distributed. However, under different working conditions, the pressure difference on the two sides of the screen mesh is different for the same specific region, such as the middle bottom and the cup body.

The pressure in the vessel is different between different blowing conditions.

(1) Under the condition of working condition 1, the pressure range in the vessel is between 0.48MPa and 0.1951MPa.

(2) Under the condition of working condition 2, the pressure range of the vessel is between 0.2223MPa and 0.2227MPa.

(3) Under the condition of working condition 3, the pressure range of the vessel is between 0.2238MPa and 0.2242MPa.

This can clearly compare the pressure of the vessel under different working conditions.

4.4.3 Velocity, Streamline and blowing dead zone

(1) Velocity

In the three working conditions, the velocity of the vessel is small, and it has many low-velocity areas. However, the inlet and outlet velocity of each working condition are large, and the internal pressure and import and export velocity of the vessel under each working condition are shown in table 1.

Table 1 Internal pressure and the velocity of import and export.

	Vessel chamber pressure （MPa）	Inlet velocity （m/s）	Export velocity （m/s）
Working condition 1	0.1948-0.1951	167.9	355.1
Working condition 2	0.2223-0.2227	125.3	369.4
Working condition 3	0.2238-0.2242	112.8	375.9

$$\frac{p^*}{p}=\left(1+\frac{k-1}{2}Ma^2\omega^2\right)^{\frac{k}{k-1}} \qquad Ma=\frac{v}{c} \qquad (2)$$

According to the formula (2), it can be seen that for the entrance, the larger the pressure inside the vessel, the smaller the velocity of inlet. For exports, the larger the pressure inside the vessel, the greater the velocity of export. Compared with the simulation results under the three working conditions, the velocities of inlet and outlet are in accordance with the formula (2).

(2) Streamline

Under the three working conditions, the flow field of the lower chamber of the vessel is very complicated, and the entire lower chamber is filled with flowing gas.

For the upper chamber of the vessel, the flow field distribution in the working condition 1 and the working condition 2 is similar. The flow field is relatively simple. By observing the flow chart under the working condition 3, the whole upper chamber has no streamline.

(3) Dead zone

According to the distribution of velocity and streamline in the vessel, it is found that under no matter which working condition, the formation of the gas flow is extremely weak in the entrance on the corner, near the wall on one side of the cup body, the middle bottom, near the inner wall of the vessel and the chamber of a certain area. The velocity is extremely low, gas or even cannot reach. Namely, there is a certain area of blowing dead zone in all three working conditions.

5 Conclusion

It is not difficult to find that the opposite side of the export must be blowing dead zones. It is easy to understand that the flow of gas is bound to be biased towards the exit side. Therefore, in the beginning of the blowing to open the exit, it is inevitable to produce the dead zone.

It is suggested to combine the three types of blowing to complement each other, thus reducing the dead zone. In addition, both exports should be shut down at the beginning, waiting for the vessel reaches a certain pressure. When the vessel is full of considerable pressure of gas, open the export right now. Under the action of pressure, the gas inside the vessel will flow toward the exits. There are gas flows everywhere inside the vessel, the blowing dead zone will be greatly reduced or eliminated.

References

1. John D.Anderson. Computational Fluid Dynamics: the Basic with Applications[M]. London: Macmillan, (2012)
2. Liang P Y. Numerical method for calculation of surface tension flows in arbitrary grids[J]. AIAA J 1991 29:161-167
3. Maureen T Kudlac, John M Jurns. Screen channel liquid acquisition devices for liquid oxygen. AIAA-2006-5054
4. Pan Hailin, Li Yong, Zhao Chunzhang, WeiYanming. Numerical Analysis of the Flow Resistance for Large Volume Surface Tension Tank, Aerospace Control and Application, 2008(03)
5. LI Yong, PAN Hai-lin, WEI Yan-ming. The Evolvement of the Study and Application on the Second Generation Surface Tension Tank, Journal of Astronautics,2007(02)