Simulation Analysis of Water Vapor Condensation during Tire Capsule Vulcanization

Shuzhen Lu, Songying Chen and Yang Cao*
Key Laboratory of High Efficiency and Clean Mechanical Manufacture, School of Mechanical Engineering, Shandong University, Jinan, China

*Corresponding author

Abstract. Even the vapor vulcanization technology has developed a lot. The uneven temperature distribution caused by the condensation water inside the capsule still exists. In this paper, a numerical simulation study was conducted on vapor condensation of tyre capsule. Using a simplified two-dimensional model, after the 240s calculation, we analysed the result. The result shows that there was flow inside, and it was mainly induced by the condensation water flowing down the wall after it had been produced on the wall. The height of the condensed water layer is about 4.0mm. The volume of condensed water in three dimensions is about 240ml, which results in uneven temperature distribution. The temperature of the wall area with condensed water is about 3°C to 13°C lower than other wall areas, and the maximum temperature difference occurs at the deepest part of the condensate layer.

Keywords: Capsule vulcanization; VOF model; Water vapor; Numerical simulation.

1. Introduction
Since the 1950s, the inner capsule of tire setting vulcanizer has gradually replaced the outer water tire methods. Nowadays, the capsule has been made an important part of the tire setting vulcanizer. The coordination between the capsule and the outer mould ensures the quality and production efficiency of the vulcanizer. The capsule vulcanization technology still has defects like incomplete expansion, asymmetric structure, low vulcanization pressure vapor Condensation water deposition and other problems which seriously affects the vulcanization quality [1-3]. The curing internal pressure medium to tire setting vulcanizer mainly includes superheated water, water vapor and nitrogen. It has been learned after the communication with a tire company that the vapor condensation mainly occurs in the vapor pressure maintaining stage, and the condensate produced is deposited at the bottom, resulting in uneven distribution of pressure and temperature, which affects quality and service life of the tire. To analyse this problem, we use ANSYS fluent to simulate the process of vapor condensation deposition.

2. Calculation Model and Method

2.1. Numerical Calculation Method
For the problem of gas-liquid phase transition, there are three kinds of flow models in Eulerian method: VOF model, Mixture model and Eulerian model. VOF model is used to deal with the problem of multiphase flow without interpenetration. Volume rate function is used to express the position of free surface of fluid and the volume occupied by fluid. It is simple and effective. Compared with Mixture model and Eulerian model, it can better describe discrete phase problem. The VOF model is selected to discuss the problem of vapor condensation deposition.
VOF model [4] is a surface tracking method that based on fixed Eulerian grid. Based on the premise that two or more fluids (or phases) do not mix with each other. This model can be used when one or more incompatible fluid interfaces are needed. In the VOF model, different fluid components share a set of momentum equations. By introducing the variable of phase volume fraction, the phase interface of each calculation unit can be traced. Within each control volume, the sum of all phase volume integral amounts to 1. All variables and their attributes are controlling the sharing of phases within the volume and represent the volume average. In this way, the variables and their properties in any given control volume purely represent a phase or a mixture phase, and are determined by the phase volume fraction. In other words, in a cell, the volume fraction of the q-phase fluid, there may be three situations: (1) \( \alpha_q = 0 \): there is no q-phase fluid in the unit. (2) \( \alpha_q = 1 \): the unit is fully filled with q-phase fluid. (3) \( 0 < \alpha_q < 1 \): the unit contains the interface between the q-phase fluid and one or other multiphase fluid. Based on the local values, appropriate attributes and variables are allocated to each control unit within a certain range.

2.1.1. Volume fraction Equation (Continuity Equation). The interface between tracking phases is completed by solving the continuity equation of volume ratio of one or more phases. For phase \( Q \)

\[
\frac{\partial \alpha_q}{\partial t} + \mathbf{v} \cdot \nabla \alpha_q = \frac{S\alpha_q}{\rho_q}
\]  

(1)

Where \( S\alpha_q \) is the mass source term. By default, the right source term of equation 1 is 0, but when you specify a constant or user-defined mass source for each phase, the right end is not 0. The calculation of the volume fraction of the main phase is based on the following constraints:

\[
\sum_{q=1}^{n} \alpha_q = 1
\]  

(2)

2.1.2. Attribute Calculation. The attributes appearing in the transport equation are determined by the phases existing in each control volume. Suppose that in the two-phase flow system, the phase is represented by subscripts 1 and 2. If the volume fraction of the second phase is tracked, the density in each cell is as follows [5] :

\[
\rho = \alpha_2 \rho_2 + (1 - \alpha_2) \rho_1
\]  

(3)

In general, for n-phase system, the average density of volume ratio is as follows:

\[
\rho = \sum \alpha_q \rho_q
\]  

(4)

All other properties are evaluated in this way.

2.1.3. Momentum Equation. By solving a single momentum equation in the whole region, the velocity field obtained is shared by all phases. The momentum equation depends on the volume ratio of all phases through the properties \( \rho \) and \( \mu \), as follows:

\[
\frac{\partial}{\partial t} (\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \left[ \mu \left( \nabla \mathbf{v} + (\nabla \mathbf{v})^T \right) \right] + \rho \mathbf{g} + \mathbf{F}
\]  

(5)

2.1.4. Energy Equation. The energy equation is also shared in each phase, as shown below:

\[
\frac{\partial}{\partial t} (\rho E) + \nabla \cdot \left[ \mathbf{v} \left( \rho E + p \right) \right] = \nabla \cdot \left( k_{\text{eff}} \nabla T \right) + S_h
\]  

(6)
Where $k_{\text{eff}}$ is Effective thermal conductivity, $S_h$ is Source term, including radiation and other volume heat sources; $E$ is Total energy.

The VOF model deals with energy $E$ and temperature $T$ as the mass average variable:

$$E = \frac{\sum_{q=1}^{n} \alpha_q \rho_q E_q}{\sum_{q=1}^{n} \alpha_q \rho_q}$$

(7)

$E_q$ of each phase is based on the specific heat and shared temperature of the phase. Properties $\rho$ and $k_{\text{eff}}$ are shared by all parties. $S_h$ includes the contribution of radiation, and also other volume heat sources.

2.2. Mesh and Boundary Conditions

In order to simulate the vapor condensation accurately, high grid resolution is usually needed. Due to the large size of the vulcanization capsule model, three-dimensional simulation will lead to a large number of the mesh, which can reach millions and make the calculation very slow. Considering that the vulcanization capsule is a rotary structure, it can be simplified into a two-dimensional model according to the symmetry, so as to greatly improve the calculation efficiency. Through communication and sharing with the tire company, it is found that the complete numerical simulation of the whole vulcanization process is not realistic, so it is simplified based on the following two aspects:

(1) Through communication, we know that the stage is water vapor filling stage, and the second stage is nitrogen filling and pressure maintaining stage. In the first stage, temperature is the key factor to determine the quality of vulcanization, and it is also the key factor to affect the condensation of water vapor, and the water generated by condensation is the cause of the current temperature difference between the upper and lower capsules. In the second stage, normal temperature nitrogen is charged. Its main function is to ensure that the pressure in the chamber does not decrease with the condensation of water vapor. The existence of nitrogen will not affect the condensation rate of water vapor. The condensation temperature of a gas is only related to the component pressure of the gas. Therefore, the simulation of nitrogen filling stage can be omitted when studying the influence of water vapor condensation on the temperature difference of sulfuration.

(2) On the other hand, considering that the condensation problem is actually a process of 'Filling a certain amount of saturated water vapor, the system radiates heat outwards (tire vulcanization heat absorption), resulting in the phase change of water vapor and generating condensation water’, the intermediate process is ignored, only the initial state and the final state are considered, and this simplified way is equivalent to the original process to a certain extent, so the current simulation can it is simplified as: at the initial time of calculation, it is considered that the capsule is full of saturated water vapor, and then a negative heat flow density is applied on the capsule wall close to the tire to simulate the heat absorption effect during the tire curing process. Under the condition of the negative heat flow density applied, the water vapor gradually condenses and deposits at the bottom of the capsule.

The grid is divided by Gambit. Because that the main area where condensation occurs is near the wall, the grid is identified on the wall. The average side length of the triangular grid on the wall is 0.5mm. The side length of the square grid in the central area is 1mm. The result of grid division is shown in Figure 1, and the effect of wall encryption is shown in Figure 2.
The current mesh has about 50000 grid cells, and the time step is set to 0.0004s. The initial state is: the chamber is full of static saturated water vapor (static pressure is 1.4MPa, temperature is 195℃). The boundary conditions are as follows: all the boundaries are wall surfaces, in which the heat flow density of tyre contour is - 3500W/m², and the rest are adiabatic.

3. Simulation Results and Analysis

3.1. Velocity Distribution
At 240s, the highest velocity in the flow field is about 0.5m/s. Due to the influence of gravity and condensate, the flow field in the system does not have symmetry, showing various complex flow patterns, as shown in Figure 3 and Figure 4. The flow is mainly induced by the condensation water flowing down the wall after it is produced on the wall.

3.2. Phase Transformation and Density Distribution
Density is an important physical quantity in this simulation. As the tire absorbs heat, a negative heat flow density generated on the capsule wall, that is, the system radiates heat outwards. Figure 5 shows the main area where water vapor condensation occurs. It can be seen that the shape of the condensation area is basically the same as that of the inner contour of the tire, that is, condensation basically occurs near the capsule wall, but after more condensation water is deposited, condensation also occurs near the gas-liquid two-phase interface, as shown in Figure 6. After the condensation water is produced, it is deposited on the tire crown under the action of gravity, thus forming a clear gas-liquid two-phase interface, as shown in Figure 7.
3.3. Condensate Water Volume

Condensation water is a very important factor in the current simulation, because it directly determines the wall thermal resistance of water vapor when heating the tire, and the temperature difference between the top and bottom of the tire also mainly comes from this. When the gas-liquid interface is formed, the temperature of the tire covered by condensation water will be lower. Figure 8 shows the condensate cloud and grid at the same time. The triangle grid side length of the bottom wall is 0.5mm, and it can be seen from the Figure 8 that the condensed water accounts for about 8 layers (i.e. the number of red grid layers from the lowest position of the tire crown in the figure9). According to the result, the height of the condensed water layer is about 4.0mm. The volume of condensed water in three dimensions is about 240ml, which is basically consistent with the measured condensed water in the actual curing process. With the same method, the condensation water depth at different times can be measured, and the change process with time is shown in Figure 9.
3.4. Temperature Distribution

The uneven temperature distribution is the main reason why the tire is difficult to be vulcanized uniformly, so the simulation of temperature distribution is an important part of the current simulation. Figure 10 is the temperature distribution cloud chart, and Figure 11 is the partial enlarged picture near the condensate area. It can be seen from the figure that the temperature of the part with condensation water deposition in the crown is indeed lower than other areas.

Figure 10. Temperature cloud.  
Figure 11. Enlarged condensation area.

In order to quantitatively compare and analyse the temperature difference caused by condensate, 36 sampling points are selected near the condensate area, as showed in the black spot in Figure 12. These points cover all condensate areas and some non-condensate areas. Figure 13 is the temperature distribution curve of these points. It can be seen that in the non-condensing area, the wall temperature is about 195°C, while in the area covered by condensate, the average wall temperature is about 187.9°C, and the temperature difference is about 7.1°C, the lowest temperature appears in the deepest area of condensate, about 182.7°C, and the temperature difference is about 12.3°C. This is in good agreement with the measured temperature difference during vulcanization. Using the same method, the temperature difference values at different times can be measured, and the change curve with time is shown in Figure 14.

Figure 12. Sampling point of wall temperature distribution.

Figure 13. Temperature distribution.  
Figure 14. Wall temperature difference with time.
4. Conclusion
In this paper, the numerical simulation is carried out under the two-dimensional simplified condition. The results show that the height of the condensed water layer deposited on the tire crown is about 4.0 mm and the volume of three-dimensional condensation water is about 240 ml. According to the phase transition cloud and location of condensation, condensation basically occurs near the capsule wall, but condensation also occurs near the gas-liquid interface after more condensation water is deposited. Because that the specific heat capacity of the liquid water is much larger than water vapor, the heat gain from the inside of the capsule passes through the area to the outside, so that the temperature of the wall area with condensed water is about 3°C to 13°C lower than other wall areas, and the maximum temperature difference occurs at the deepest part of the condensate layer, these numerical results are basically consistent with the measured values. In future work on the improvement of tire capsule curing technology, this paper provides experience of numerical simulation and verification of basic data, which is benefit for the future research work on uneven temperature distribution that caused by condensation water.

Acknowledgments
Project supported by the Young Scholars Program of Shandong University (Gran No. 2018WLJH59), the Fundamental Research Funds for the Central Universities (Grant No. 2017TB0031).

References
[1] Ruan Jun. Tire curing bladder life improvement measures[J]. Chemical Enterprise Management, 2016(29):273.
[2] Qitao Sun, Xiuhong Yao. Analysis on the cause of abnormal damage of semi-steel radial tire curing bladder during use and its solution[J]. Rubber Science and Technology, 2019,17(10):586-588.
[3] Wen Zhang. Effect of curing bladder structure on its service life and quality of semi-steel radial tire[J]. Tire Industry, 2018, 38(05):302-307.
[4] Mohammad Akhlaghi, Vahid Mohammad, Nowrouz Mohammad Nouri, Morteza Taherkhani, Mehdi Karimi. Multi-Fluid VOF Model Assessment to Simulate the Horizontal Air-Water Intermittent Flow[J]. Chemical Engineering Research and Design, 2019(152):48-59.
[5] Zhengguo Zhao, Xiaolin Pu, Gui Wang, Zhijun Li, Liqun Chen, Cheng Cao. Numerical study of the gas-fluid displacement invasion behavior in a fractured carbonate reservoir[J]. Journal of Natural Gas Science and Engineering, 2015, 27:686-691.