Study on Safety Characteristics of Gate Valve Transient Operation in High Temperature Molten Salt Thermal Energy Storage System

Wang Bing, Peng Changhong and Guo Yun*

School of Physical Sciences, University of Science and Technology of China, Hefei, 230026, China

Email: guoyun79@ustc.edu.cn

Abstract. At present, the new energy industry in the world is thriving, but there are obvious intermittent and random characteristics of various power generation methods such as solar energy and wind power. The molten salt energy storage power generation system can effectively reduce the peak fluctuations of the power grid and greatly improve the overall economic advantage. As an important component of the pipe system, the gate valve is widely used in power generation systems. Because molten salt systems may face faster power changes so during certain transient process, the main gate valve may have to withstand rapid changes in mass flow and temperature, which may cause rapid changes in the temperature field of the valve body, resulting in additional thermal stress. In this paper, the CFD method is used to analysis the flow and temperature fields of the main gate valve during the transient process. The key to success is to describe the geometry as accurately as possible and to build a reasonable grid because of larger calculation quality for transient. Then based on the temperature field of valve body the thermal stress and change law can be obtained, which can provide guidance for the design of the gate valve.

Keywords: gate valve; transient process; fluid-structure coupling

1. Introduction

The molten salt has been widely applied to energy storage technology, because of its outstanding thermal stability, high specific heat capacity, good chemical stability, and low saturated vapor pressure. The molten salt energy storage devices usually combined with solar energy and wind energy generation systems, it can storage extra energy and effectively reduce the peak volatility of the power grid, and greatly improve the overall economic advantage. At present, existing studies on the molten salt energy storage technology paid attention to the heat transfer characteristics and safety abilities. Mashui Huq et al. proposed a method to solve the issue of the severe deficiency of clean drinking water in many countries with the enhancement of the thermal capabilities of molten salt to purify the water.[1] Chao Xu et al. investigated the effects of various parameters, such as flow rate and temperature of inlet molten salt, porosity and height of the system, and the thermal losses on the
thermal performance of a packed-bed molten salt thermocline thermal storage system.[2] Zheshao Chang et al. researched the effect of the physical boundary conditions on the thermal performance of molten salt thermocline tank.[3] Sebastiano Turrinia et al. introduced an innovative small-scale prototype plant integrating a solar dish concentrator with a molten salt storage system.[4] Xiaolei Li et al. conducted dynamic simulation of two-tank indirect thermal energy storage system with molten salt.[5]

Different from conventional power plant such system often changes work condition according to external demand. In this system under high pressure and temperature, the gate valve which controls the channel system mass flow rate may have to withstand rapid changes in flow rate and temperature during certain transient process. It is especially important to evaluate the safety of the main gate valve during certain transient process. Computed Fluid Dynamics (CFD) has been widely used in industrial designs, like reactor fuel assembly and power plant channel systems. In this paper, a method which combines CFD method and Fluid-structure coupling method to evaluate the safety of the gate valve was proposed. Using this numerical simulation method, the steady state of discharging process of the molten salt energy storage system is analyzed, then transient analysis is based on steady state results. The three-dimensional flow field and temperature distribution of the main gate valve can be obtained, which is the basic of the gate valve design and thermal-stress analysis. Based on these simulation results, the design of the valve body can be evaluated and some improvement measures can be brought out.

In this paper, the 3D CFD simulations with fluid-structure coupling for both the steady-state and transient process of a gate valve are reported. First, the steady-state is analyzed for selecting the grid and turbulence model. Then based on three different geometry the transient process is simulated. At last, the safety of the gate valve is analyzed in detail.

2. Modelling

![Diagram of gate valve model](image)

Figure 1. The 3D model of the computation domain and its boundaries.

2.1. Turbulent model
Standard $k$-$\varepsilon$ model is used to model the turbulence. The governing equation for the $k$-$\varepsilon$ model is given as:

$$\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_i} (k u_i) = \frac{\partial}{\partial x_j} \left( -\frac{k}{\rho} \frac{\partial u_i}{\partial x_j} - \frac{1}{\rho} \frac{\partial p}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \frac{\partial k}{\partial x_j} \right) - \frac{\partial u_i}{\partial x_j} \frac{\partial k}{\partial x_j} - \frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j}$$

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial x_j} (\varepsilon u_j) = \frac{\partial}{\partial x_j} \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + \frac{\varepsilon}{k} \left( -\frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon f} C_{\varepsilon} \right)$$

For near-wall treatment, a scaled wall function is used to solve the boundary layer. The grid resolution in near wall regions can be characterized by $y^+$, which is the dimensionless wall distance from the wall. As for the scaled wall function approach, the $y^+$ value of the wall adjacent cell is recommended between 11.225 and 500.

The upper chamber shown in Figure 2 is most likely in laminar state, especially in the top region. However, the boundary of laminar turbulent flow region is very hard to be determined and simulated and the transition from turbulent flow to laminar flow is unachievable. So the turbulence model is applied in the whole domain, which may overestimate heat exchange. However, it is relatively conservative for this study and acceptable.

![The upper chamber](image1.png)

Figure 2. Grid system of the gate valve

### 2.2. Boundary condition and solution method

The gate valve under high temperature and high pressure of a molten salt power storage station was adopted in this study. The computation domain is shown in Figure. 1, which has two velocity inlets and one pressure outlet. The gate valve is composed of the valve body and the internal fluid which is shown in Figure 2, and the internal space is divided into an upper chamber and a lower flow path. Meanwhile, the boundaries of the valve body and channels are defined as no-slip wall condition. The thermal boundary of the channel is adiabatic and the surface of the gate valve is defined as natural convection condition with heat transfer coefficient as 10 $W/(m^2 \cdot K)$. The interface between the valve body and the internal fluid is defined as coupled interface to exchange the temperature data. The gate valve with multi-computation domain is a typical complex fluid-solid coupling system.

To make the inlet flow fully developed, a 15 meters long tube is added at the inlet1 to eliminate the influence of the uniform velocity inlet. By using the user-defined functions (UDF), the inlet temperature and velocity is defined as a time dependent function for both of inlet 1 and inlet 2, which is shown in Figure 3. System operating pressure is 6.7 MPa. Since the temperature variation range is not large, the physical properties, such as viscosity, density and thermal conductivity are treated as constant according to the operation pressure.

The finite-volume approach is adopted to discrete the governing equation, and it is expressed by second-order upwind difference. Meanwhile, the typical simple algorithm is used to compute the flow
field. The continuity residual error is set as 1.0e-5. For transient calculation, the time step is set as 0.002s for the first 3 seconds, and 0.01s for the rest. In transient calculation, the results can reach the convergence standard in no more than 40 iterations in every timestep.

![Figure 3. The boundary condition of inlet 1 and inlet 2](image)

### 2.3. Mesh

UG NX 12.0 is used to generate the geometry model. As shown in Figure 2, The most of the flow channel except the upper chamber of the valve adopts the structural hexahedral grid. The valve solid body and the upper chamber is divided by unstructured mesh. The grid of the boundary layer is refined to satisfy the $y^*$ limit. Fig. 4 shows the distribution of wall $y^*$. It can be seen that most $y^*$ reach the requirement. Therefore, the influence of the $y^*$ limit is basically negligible, and the near-wall grid is reasonable.

To validate the mesh independence three different meshes are set and shown in Table 1. When fluid flows into the interior of the gate valve, here is a suddenly expansion in the channel, so the flow field is relatively complicated. Figure 4 shows the velocity distribution in the horizontal centreline of the central cross section of the valve body (vertical to the flow direction). According to the comparison of the results of coarse mesh, base mesh and refined mesh, it can be found that the base mesh is sufficient for the calculation.

![Figure 4. Contour of wall $y^*$](image)

### Table 1. Details of three mesh schemes.

| Item         | Cells of mesh ($10^4$) |
|--------------|-------------------------|
| Coarse mesh  | 140                     |
| Base mesh    | 194                     |
| Refined mesh | 270                     |

![Figure 5. Velocity distributions with different meshes.](image)
3. Results and discussion

3.1. Steady state analysis
In this study, the flow field and the temperature filed in the main gate valve are the input condition for thermal stress calculation, which should be paid more attention. The geometry in the valve body can be considered as a combination of sudden expansion and contraction, and the variation distance is very short, which is similar to a nozzle structure that does not change smoothly.

![Streamlines of the horizontal section](image1)

(a) Standard $k$-$\varepsilon$ model  
(b) Reynolds Stress Model

Figure 6. Streamlines of the horizontal section

Fig. 6 shows the flow field in the mainstream. After the fluid flow into the valve body, vortices are found in the chamber of the valve due to the separation of the boundary layer. It can be interpreted as fluid hitting the wall after entering the sudden expansion, thereby reverse flow forms vortices. This phenomenon can be confirmed in reference [6]. A direct numerical simulation (DNS) is performed for wavy channels, in the upper and lower parts of the flow direction, there are also two vortices. The structural features are similar to those of present study, so the result of the $k$-$\varepsilon$ model is reasonable and acceptable in a way for engineering point.

![Secondary flow field in the gate valve central section](image2)

Figure 7. Secondary flow field in the gate valve central section

The secondary flow in the valve channel is shown in figure 7. Due to the existence of the flow vortices, the secondary flow is very complicated, which have a great influence on the temperature field and heat transfer. There are two large vortexes in the center which is caused by the elbow pipe. In figure 8, as flowing through the bend, it gradually develops into a double vortex which will enhance the heat transfer. The vortexes are still developing along the channel. The flow at the corner is strong
and can reach 0.24 m/s. At the same time, there are several small vortices near the solid boundary of the valve body in figure 7.

![Secondary flow fields before and after the elbow](image1)

(a) Before flowing into the elbow  
(b) After flowing out of the elbow

Figure 8. Secondary flow fields before and after the elbow

![Secondary flow field in gate valve central section](image2)

(a) Standard $k$-$\varepsilon$ model  
(b) Reynolds Stress Model

Figure 9. Secondary flow field in gate valve central section

The Reynolds Stress Model should be the best time-average model at present, which can fully reflect the anisotropy of Reynolds stress, so it can capture the secondary flow more accurately. The RSM model is used to analyse the secondary flow field and validate the reasonability of $k$-$\varepsilon$ model. These results shown in figure 9 do not contain the upper chamber and the. Hence, the vortex in the middle is different from that shown in figure 7. As shown in Figure 6, both standard $k$-$\varepsilon$ model and Reynolds Stress Model can capture the vortexes near the wall in the mainstream direction. As shown in Figure 9, due to the influence of the elbow, there is also a large vortex in the middle, which is consistent with the results of the standard $k$-$\varepsilon$ model. The Reynolds stress model is more accurate for vortexes capture near the wall. But the overall secondary flow is similar, it can be seen that the standard $k$-$\varepsilon$ model can reflect the main characteristics of the flow field. In summary, the calculation using the standard $k$-$\varepsilon$ model can reflect the typical characteristics of the flow in this system. Since the Reynolds stress model requires much more computing resource (more than 10 times), the standard $k$-$\varepsilon$ model is chosen for the following transient calculations.

Figure 10 shows the temperature distribution of the valve surface under steady state. The highest temperature of the valve body is 523.06 K and the lowest temperature of the valve body is 438.91 K.
The maximum temperature difference between the inner and outer walls is 84.15 K. Temperature field analysis is the basis of thermal stress analysis. Figure 11 shows the Total deformation and equivalent stress of the valve body under discharging process. Fixed support is given on both ends of the gate valve. The largest deformation position appears at the top of the gate valve, which reaches 1.92mm. Ignoring the stress concentration at the sharp corners the thermal stress in the valve body reaches about 1400 MPa. Using the method proposed in this paper, both the temperature field and stress field of the valve body can be obtained, so the overall safety of the gate valve can be evaluated.

3.2. Transient analysis
The boundary condition of transient process is shown in Figure 3. In the transient process, the temperature difference between the inner and outer walls reaches 88.79 Celsius degrees at highest, and 70.96 at lowest. Two optimized designs are proposed as comparisons, for optimized design 1 the branch pipe is moved to the upstream of the elbow pipe. For optimized design 2, the gap between the upper and lower chamber in the valve body is changed to 100 mm which is 160mm originally.

Figure 11 (a) shows the max temperature of the valve body. From 2s the max temperature of the valve body increases about 3 Celsius degree due to the influence of hot water injection from the branch pipe. Then because of the increasing of mainstream flow rate, the cooling effect of the mainstream dominates. The max temperature of inner surface gradually drops at an average of 0.52 Celsius degree per second. For optimized design 1, because the hot water injected by the branch pipe...
is fully entangled with the mainstream, the temperature of the valve body increases slowly. For optimized design 2, the changes in the first 15s is basically the same. Because the flow in the upper chamber is weakened, the maximum temperature decreases slowly and gradually stabilize. At 35s, the maximum temperature is 0.99 degrees Celsius lower than that of the prototype. Figure 11 (b) shows the average temperature of the valve body. Due to the stirring of hot and cold fluids from the branch pipe and the mainstream, the average temperature remains unchanged in the first 10s. After 10s, the branch flow rate is gradually reduced to 0. Then the average temperature drops about 2.30 degrees Celsius. For optimized design 1, average temperature is almost the same as prototype. For optimized design 2, the average temperature is 1.2 degrees Celsius lower than prototype, which is due to the influence of the attenuation of upper chamber flow.

![Figure 11](image)

(a) Maximum temperature of the valve body  
(b) Average temperature of the valve body  

(c) Maximum temperature of the inner surface  
(d) Average temperature of the inner surface

Figure 11. Temperature change during transient operation

Figure 11 (c) shows the maximum temperature of the inner surface. Different from the maximum temperature of the valve body, the temperature of the optimized design 2 is higher than that of prototype and optimized design 1 after 15s. This is due to the narrower gap, temperature in the upper chamber changes more slowly. Figure 11 (d) shows the lowest temperature of the inner surface. For optimized design 2, the lowest temperature of the inner surface is significantly lower than the prototype in the first 10s. In the end, for three designs the lowest temperatures tend to be the same due to mainstream cooling.

In general, the average temperature drops 2.30 Celsius degrees for the valve body and 22.1 Celsius degrees for the inner surface. The average rate of change is about 0.52 Celsius degree per second. The
temperature variation will cause additional thermal stress, which will affect the intensity and sealing of the valve body. For optimized design 1, the mixing of hot and cold fluids is enhanced, so the maximum temperature is lowered than that of the prototype. For optimized design 2, the influence of the mainstream is reduced, the average temperature drop is reduced at 1.2 degrees Celsius.

4. Conclusion
In this paper, the temperature field and flow field of the main gate valve during a transient process is analyzed based on 3D CFD simulations. The evolution laws of temperature are provided in detail. And the safety of the valve body was evaluated for three different designs. The conclusions are as follows:

- The k-ε model can fairly accurately simulate the dynamic hydraulic characteristics of a gate valve. And a three-dimensional temperature distribution is obtained for thermal stress analysis.
- Moving the branch pipe to the upstream of the bent pipe can lower the maximum temperature of the valve body and uniform the temperature field. Narrowing the gap between the upper and lower chamber can reduce the average temperature. Both optimized designs can reduce the thermal stress.

Acknowledgement
This work is financially supported by the National Key R&D Program of China (2017YFE0300502)

References
[1] Huq, Mashiul, and Sakib Ahmed. "Prospects of incorporation of nanoparticles in molten salt for water purification." Renewable and Sustainable Energy Reviews 82 (2018): 2814-2819.
[2] Xu, Chao, et al. "Parametric study and standby behavior of a packed-bed molten salt thermochline thermal storage system." Renewable Energy 48 (2012): 1-9.
[3] Chang, Zheshao, et al. "The effect of the physical boundary conditions on the thermal performance of molten salt thermochline tank." Renewable energy 96 (2016): 190-202.
[4] Turrini, Sebastiano, et al. "An innovative small-scale prototype plant integrating a solar dish concentrator with a molten salt storage system." Renewable Energy 123 (2018): 150-161.
[5] Li, Xiaolei, et al. "Dynamic simulation of two-tank indirect thermal energy storage system with molten salt." Renewable Energy 113 (2017): 1311-1319.
[6] Stalio, Enrico, and M. Piller. "Direct numerical simulation of heat transfer in converging–diverging wavy channels." Journal of heat transfer 129,7 (2007): 769-777.
[7] Launder, Brian Edward, and Dudley Brian Spalding. Mathematical models of turbulence. BOOK. Academic press, 1972.
[8] Hu, Biao, et al. "Numerical investigation of conjugate heat transfer of an underwater gate valve assembly." Applied Ocean Research 56 (2016): 1-11.