Applied Mechanics and Materials
ISSN: 1662-7482, Vols. 48-49, pp 1310-1314
doi:10.4028/www.scientific.net/AMM.48-49.1310

Online: 2011-02-02

The Simulation and Experiment Study on the Gas Flow in the No-loaded and Cold Vacuum High-pressure Gas Quenching Furnace

Zhijian Wang [1,a] and Xiaofeng Shang [1,b]

[1]School of Mechanical and Electrical Engineering, Shenyang Aerospace University, Shenyang Liaoning, China, 110136

[a]wangzhijian1974@sina.com, [b]xfshang@vip.sohu.com

Keywords: Vacuum high-pressure gas quenching, Hot wire anemometer, CFD, Flow field, Pressure field.

Abstract. In order to learn gas flow state in the vacuum high pressure gas quenching furnace, this paper simulates and tests the gas flow under the no-loaded and cold state. Hot wire anemometer is used to measure the speeds of some feature points, on the one hand to provide boundary conditions for the numerical simulation, and on the other hand to compare with the numerical simulation results. FLUENT software is used to simulate the gas flow of nozzle-type vacuum high-pressure gas quenching furnace. The results show that at the center of the furnace appears high-pressure low-speed zone in which is resulted by the gas collision there, and the vortex also appears in the area around the furnace. The results mean that the cooling rate of works will be slow there. Different exit velocities of five nozzles cause the uneven flow distribution, which will affect the cooling uniformity of works. The comparison between the simulation results and the measured results shows that the error is within 10%. It means that numerical simulation method to predict gas flow is feasible and the results are reliable in high pressure gas quenching furnace.

Introduction

Vacuum high-pressure gas quenching is to heat works under vacuum state, and then cool and harden them by high-pressure and high-velocity flow gas. The cooling rate and cooling uniformity of the works are the main indexes to evaluate the performance of vacuum high-pressure gas quenching equipment. Under the specific parameters and batch of the furnace, the temperature change of works is mainly affected by the gas flow field in furnace chamber. The research on the gas flow can help us know the heat transfer by convection between the gas and works, which can guide the rational design and selection of the furnace parameters, and quench the works to the perfect effect [1-4]. Computational Fluid Dynamics (CFD) is an important method to study the complex gas flow during high pressure gas quenching.

Vacuum high pressure gas quenching furnace is an opaque sealed pressure vessel, so it is difficult to measure the flow field by the optical measuring instrument which is the most advanced measuring method at present. When the hot-wire anemometer is used to the gas flow velocity, there are some difficulties we face during the measurement. The first one is the high-temperature and high-pressure state makes it difficult to ensure the accuracy of the measuring instrument. The second is the complexity of flow field makes it difficult to select the measuring points. Less measuring points can not reflect the real flow state, but more ones will disturb the flow field. Furthermore, it is difficult to reflect the detailed flow from the limited measuring points. In order to deeply understand the flow and heat-transfer mechanism to improve equipment performance, we are eager to acquire more precise and more detailed information, and numerical simulation is just the right manner.

This paper simulates and tests the gas flow under the no-loaded and cold state. No-loaded and cold state is mainly based on the following considerations. The cold state is chosen is because the flow measuring devices can not guarantee the accuracy at higher temperature and it is inconvenient to measure under hot state. In addition, the gas flow is the forced convection driven by the fan and we can ignore the impact of the buoyancy force generated by the density change on the gas flow. No-loaded furnace is chosen is because the smaller turbulence makes the measuring accuracy higher relative to loaded one, and the gas flow is free jet at the nozzle exit, on which no-loaded furnace and loaded one have less affect. In this paper, some characteristic points are selected to measure the gas flow velocity under the cold-state and no-loaded state, and the results are compared with the simulating ones.

Experimental study on the gas flow of no-loaded vacuum high-pressure gas-quenching furnace

Equipment. In this paper, hot wire anemometer is used to measure the velocity of gas flow in the furnace. The type of hot wire anemometer is KA22, the measuring range is between 0 and 50m/s and the accuracy is ±2%.

Physical Model. Figure 1 shows the main structure and gas circular flow route of the nozzle-type high- pressure gas quenching system. The high-pressure gas quenching system consists of fan, heat exchanger, wind pipes, nozzles, heat shields and the furnace mantle and other components. After the process of the works heated and kept temperature under and vacuum state, the hot zone is filled with the high-speed and high-pressure cool gas which is forced from the fan through the 12 wind pipes, 60 nozzles to the works. After the heat exchange with the works, the hot gas runs out through the orifices of rear and front heat shields, is cooled through the heat exchanger and then to the air inlet of the fans. So the whole circular flow process is finished. The maximum working pressure of the furnace is 0.6MPa.

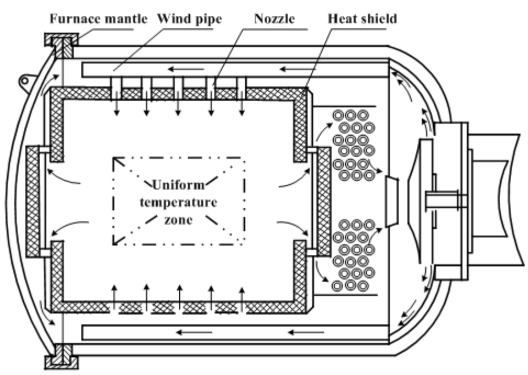


Fig.1 The main structure and gas circular route of nozzle-type high-pressure gas quenching furnace

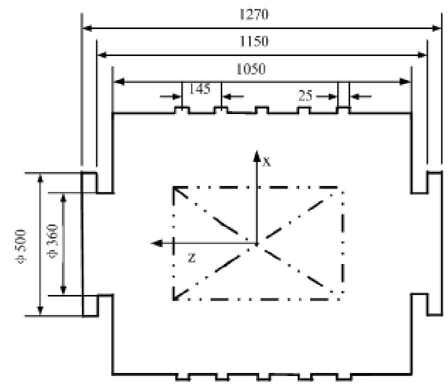


Fig.2 The structure and coordinate of hot zone

The center of hot zone is taken as the origin of coordinate system. The structure, size and the location of the axes of hot zone are shown in Figure 2. During the computer simulation, because the hot zone is symmetrical on the X axis, Y axis, a quarter of the structure is taken as the computational model. However, because the gas effluent speeds of 5 nozzles are different on every wind pipe along the Z direction, the full length of hot zone along the Z direction must be taken. In order to contrast with simulating results, the experimental data points are also taken within the region.

Measuring Results. The experimental conditions are that nitrogen is the quenching gas and quenching pressure is 0.45MPa. In order to compare the real results and simulating ones and provide boundary conditions for the computer simulation, we pick up some points to measure the velocities and the results are shown the following.

Table 1 show the velocities of different measuring points which lie on X=0, Y=0.4m and Z=-0.29, -0.145, 0, 0.145 and 0.29m. The measuring points are just the axis of five nozzles. We can find that different nozzle has different exit velocity and farther away from wind pipe, the higher velocity of the nozzle is. This is because the effluent of constant cross-section wind pipe makes static pressure of nozzles unequal.

Table 1 The measuring results of velocities (on X =0, Y=0.4m)

Z(m)	-0.29	-0.145	0	0.145	0.29
V(m/s)	23.2	26.8	32.3	33.2	34.3

Table 2 shows the velocities of the measured points which are on X = 0, Z = 0.29m and along the direction of Y axis, Y is respectively 0, 0.15, 0.25, 0.35, 0.45m. We can see that farther away from the nozzle, the flow rate is lower and gas from the nozzle collides at the center and low velocity region appears there.

Table 2 The measuring results of velocities (on X=0, Z=0.29m)

Y(m)	0	0.15	0.25	0.35	0.45
V(m/s)	3.1	8.5	15	21.3	45.8

The table 3 shows the velocities of measured points which are at X = 0, Z = 0.525m, away from the inlet of wind pipe, along the Y axis to take Y = 0, 0.05, 0.1, 0.15 m values.

Table 3 The measuring results of velocities (on X=0, Z=0.525m)

Y(m)	0	0.05	0.1	0.15
V(m/s)	5.7	5.8	7.6	5.6

The table 4 shows the velocities of measuring points which are at X = 0, Z = -0.525m, close to the inlet of wind pipe, along the Y axis to take Y = 0, 0.05, 0.1, 0.15 m values.

Table 4 The measuring results of velocities (on X=0, Z=-0.525m)

Y(m)	0	0.05	0.1	0.15
V(m/s)	5.3	5.9	6.2	5.8

In order to determine the exit velocity of each nozzle as inlet boundary condition during the computer simulation, we measured the average exit velocities of the nozzles in the experiment which are shown in the table 5. The nozzles are numbered as $1^{\#}$, $2^{\#}$, $3^{\#}$, $4^{\#}$ and $5^{\#}$ from the blind end to the entrance of wind pipe.

Table 5 The Average exit velocities of nozzles [v(m/s)]

Number of nozzles	$1^{\#}$	$2^{\#}$	$3^{\#}$	$4^{\#}$	$5^{\#}$
Average exit velocity	47.9	45.5	42	36.5	31.1

Numerical simulation of gas flow

Control Equation. The standard k-ε turbulent model is used. FLUENT is used to simulate the complex flow. Assume that gas pressure reaches working pressure directly without blowing and refilling time of fan.

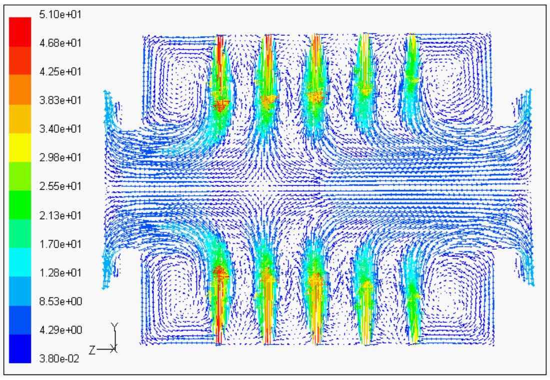
The General equation is written as [5]

$$\frac{\partial(\rho\phi)}{\partial t}+div(\rho V\phi)=div(\Gamma grad\phi)+S \tag{1}$$

In formula (1), each item is respectively unsteady, convection, diffusion and source item. The control equation consists of continuity equation, momentum equation, energy equation, k and ε equations.

Boundary conditions. The inlets of simulation model are set at the exits of nozzles and the velocity on the inlet plane is the exit one of each nozzle tested in the table 5. Gas returns back from the both sides and the locations of the gas exit are there. The quenching system is a kind of closed circulation, and the fan static pressure will be consumed by all kinds of loss and when gas reaches the exit, so exit pressure is just the working pressure 0.45MPa. In order to reduce the relative error of pressure, we take the exit as a reference pressure plane, on which the pressure is zero, so the pressure value is relative one relative to the reference pressure plane.

Figure 3 shows the X = 0 plane velocity vectors and in order to be displayed more intuitively, and the velocity and temperature distributions of the entire furnace area are represented by use of symmetrical function of software. It can be seen from the figure a vortex appears in the hot zone around the area, therefore the gas flow is slow. Exit velocities of five nozzles different results in uneven distribution of flow field, which is bound to affect the cooling uniformity of the works.



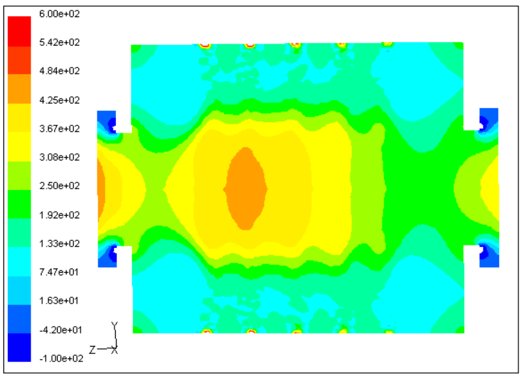
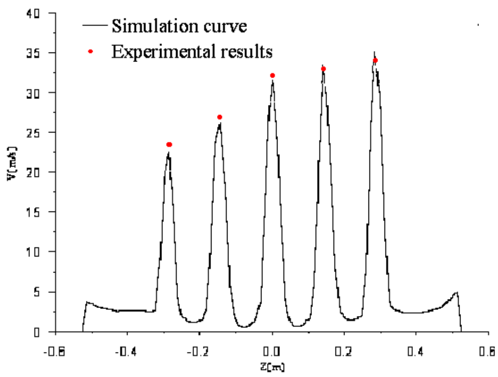


Fig. 3 Velocity vector of X=0 plane (m/s)

Fig. 4 Pressure of X =0 plane (Pa)

Figure 4 shows the pressure distribution on the X = 0 plane. We can see high-pressure area appears in the middle of the furnace chamber, which is due to the reducing of velocity resulted by the collision of jet gas from the nozzles.



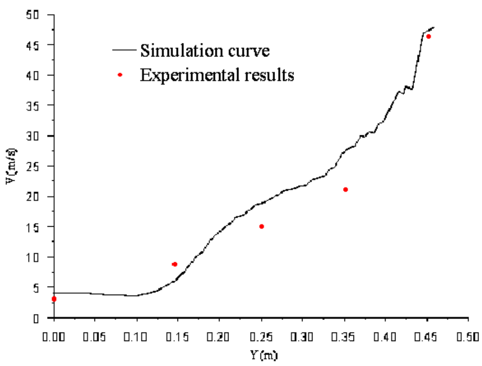


Fig. 5 Comparison of computer simulation with experimental results (X=0,Y=0.4m)

Fig. 6 Comparison of computer simulation with experimental results (X=0, Z=0.29m)

In order to compare experimental results and simulation results, we take different positions to make speed curves. Figure 5 shows velocity change at X = 0 plane, Y = 0.4m, along the Z direction.

It can be seen from the figure, the velocity is higher along the nozzle axis, but the lower rate between the two nozzles. Figure 6 shows velocity change at X = 0 plane, Z = 0.29m, along the Y direction. Figure 7 and 8 show velocity changes at X = 0 plane, Z = 0.525m and Z = -0.525m, along the Y direction.

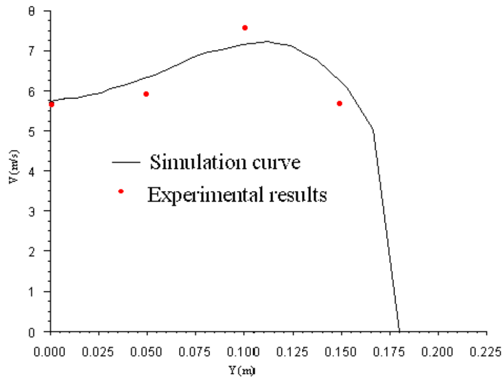


Fig. 7 Comparison of computer simulation with experimental results (X=0, Z=0.525m)

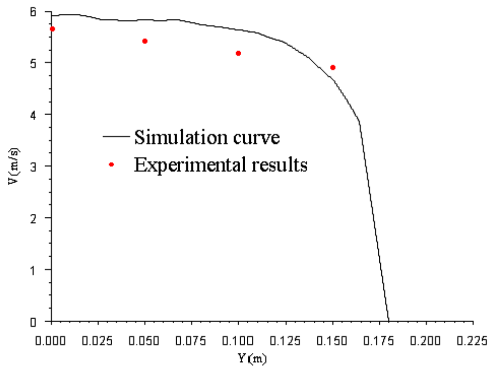


Fig. 8 Comparison of computer simulation with experimental results (X=0, Z=-0.525m)

Comparing the simulation results with the measured results, we can find they are basically consistent, and the error is within 10%, which proves that numerical simulation of flow field calculation and prediction is feasible and the results are reliable in high pressure gas quenching device. Meanwhile, there must inevitably be some errors between them. On the one hand some simplifications and assumptions are made during the simulation, on the other hand the apparatus and human errors inevitably exist during the measurement of flow speed, such as that environmental temperature changes and the instrument is not placed vertically to the incoming flow are all the causes of errors.

Summary

1) FLUENT is used to simulate the complex flow in the furnace chamber and predict the gas flow distribution which will contribute to the practical quenching process. Hot-wire anemometer is used to measure velocities.

2) The simulation and test results show that at the center of the furnace appears high-pressure low-speed zone and the vortex also appears in the area around the furnace, which means that the cooling rate of works will be slower there. Different exit velocities of five nozzles cause the uneven flow distribution, which will affect the cooling uniformity of works.

3) The error between the simulation results and the measured results is within 10%.

References

[1] C.P. Yan: *Vacuum heat treatment process and equipment design* (Machinery Industry Press, China 1998)

[2] ASM Heat Treating Society et al. Heat Treating Industry 2020 Vision-Heat Treating Industry of the Future, 1997,6. Chicago.

[3] B. Edenhofer and J. W. Bouwman: Industrial Heating, Vol.36 (1998) No.2, pp12-17

[4] Y.Z. Xu: Heat Treatment. (2000) No.1, pp20-23

[5] W.Q. Tao: *Numerical heat transfer* (Xi'an Jiaotong University Press, China 2001)