Numerical Simulation of Complex Flow field in Quenching Furnace with Mixture of Nitrogen-spray water Eject Quenching under Normal Pressure and High Velocity

Baodong SHAO, Heming CHENG, Jianyun LI, Ziliang LI, Lijun HOU, Jie HOU, Lifeng WANG

***Abstract*—**The flow field in quenching furnace designed by us is complex, and the fluid flow in furnace is the mixture of Nitrogen and spray water under normal pressure and high velocity. The two-phase flow based on mixture model is simulated by CFD software FLUENT. The velocity and pressure (static and dynamic) fields in furnace at different case (such as different inlet velocity, different volume percent of water droplet and different velocity differences between Nitrogen and spray water) are given. The less is the volume percent of water droplet, the more is the maximum velocity at outlet. The simulation results show that the flow field in furnace is complex as analysis. The results also show that the value of slip velocity between two phases has more contribution to the mixture velocity of the two-phase, and the furnace designed can accelerate the velocity of flow field.

***Keywords*—**Nitrogen-spray water mixture, Gas quenching, Two-phase flow, Numerical simulation.

I. Introduction

As a quenching media, the thermal conductivity of gas is usually smaller than that of oil and water, but gas can adjust quenching process according to workpiece shape and specific requirement of material, and the thermal conductivity of gas can be enhanced to the level of oil and water by increasing velocity and pressure. Compared with water and oil quenching, gas quenching has many advantages, such as good uniformity, little deformation, no quenching spot, no environment pollution, easy to control techniques parameter, and low cost. Because vacuum high pressure gas quenching has more requirements for equipment and cost, ordinary pressure and high velocity gas quenching may have wider applications. CHENG et al. [1] designed the equipment of ordinary pressure high velocity gas quenching by themselves and it has same quenching effect as vacuum quenching. Heck et al. [2] studied the gas flow field and simultaneous heat transfer process in gas quenching by numerical simulation for surface treatment of cylindrical sample geometry. The evaluation of optimized flow conditions and nozzle arrangements to achieve a maximum overall heat release to maximize the quenching efficiency and a local smooth distribution of the cooling process for avoidance of spatial hardness variations by derivation of an optimized nozzle arrangement and appropriate operation conditions of the gas jet array with respect to the three dimensional sample geometry of a cylinder to be quenched.

Some researches [3, 4] showed that the cooling performance of mixed two gases is better than single pure gas, thus it is very important to research quenching of mixture of two gases. At present, studies of mixture quenching are limit to mixed gas of He, N_2, CO_2, Ar or H_2, and some of gas is expensive, so cheap and high capability mixed gases are needed. Quenching by spray water jetted by air is an effective processing heat treatment technology. The heat transfer mechanism of spray water jet quenching is very complex, present research is still explored. MEI et al. [5] studied spray cooling heat transfer coefficient on high temperature surface, and analyzed the effects of spray water density, high surface temperature, droplet dimension and droplet velocity on spray cooling heat transfer coefficient. GAO et al. [6] investigated numerically a droplet impinging on a constant temperature flat surface by the method of Volume-of-Fluid (VOF), and the results showed that when the initial size is smaller and the impact velocity is bigger, the cooling effectiveness will be higher. LIU [7] experimentally studied the boiling critical heat flux by mist cooling, and found the correlations between CHF and the mist flow conditions, which consist of mass velocity of spray water, mean diameter of spray water droplets and mixed impact velocity of droplets-air.

Manuscript received Jule 9, 2010: Revised version received July 18, 2010. This work was supported in part by the Yunnan Key Application Basic Research Foundation (No. 2007A0015Z)

Baodong Shao is with Kunming University of Science and Technology, Kunming, China (e-mail: shbd_1221@163.com).

Heming Cheng, is with Kunming University of Science and Technology, Kunming, China (e-mail: chenghm650093@yahoo.com.cn).

Jianyun Li is with Kunming University of Science and Technology, Kunming, China (e-mail: jean_lee_km@yahoo.com.cn).

Ziliang Li is with Kunming University of Science and Technology, Kunming, China (e-mail: Lziliang@yahoo.cn).

Lijun Hou is with Kunming University of Science and Technology, Kunming, China (e-mail: houlijun978@yahoo.com.cn).

Jie Hou is with Kunming University of Science and Technology, Kunming, China (e-mail: houjie01107@yahoo.com.cn).

Lifeng Wang is with Kunming University of Science and Technology, Kunming, China (e-mail: wanglifeng0314@163.com).

ZHANG et al. [8] investigated the steady-state characteristics of the spray cooling system, and analyzed the effects of the heat flux value, the spray distance, the sink temperature, the subcooling degree of the return liquid on the heat transfer performance and the evaporation efficiency of the spray liquid. DU et al. [9] simulated the spray quenching process by establishing a nonlinear FEM model comprising variable physical property parameters and phase transformation. MEHMET SHALA et al. [10] studied the unstructured staggered mesh methods for fluid flow, heat transfer and phase change. VASOS PAVLIKA [11] gave a numerical design technique associated with the design of axisymmetric ducts for incompressible rotational flow. FRIEDRICH K. BENRA and SIAVASH H. SOHRAB [12] studied the modified theory of laminar flow around rigid and liquid cylinders. MAHDI HAMZEHEI et al. [13] experimentally studied the visualization of fluid flow and measurement of temperature variation at various conditions in the open channels. DAMELYS ZABALA et al. [14] studied the solute concentration effect in diffusion coefficients for a moving boundary mass transfer model which represents carbon dioxide diffusing through a n-decane liquid phase inside a glass capillary tube. SERKAN ÖZGEN et al. [15] studied the hydrodynamic stability problem of two-fluid boundary using a numerical method.

Because the thermal conductivity of Nitrogen is bigger than that of air and it is safe, abundant, cheap, inertia, if it is mixed with spray water, the mixture will has favorable cooling effect. Therefore, ordinary pressure and high velocity Nitrogen and spray water jet quenching is numerically simulated in this paper, and the variety of flow field in quenching furnace with pressure and velocity of inlet and Nitrogen-spray water mixture ratio.

II. Numerical model

The configuration of quenching furnace is shown in figure 1. The mixture of Nitrogen and spray water is jetted into the outside canister of furnace with certain velocity and pressure, and then into the inner canister through hole of inner canister, finally into outlet through collection loops, where the workpiece is cooled. Because the mixture ratio of Nitrogen and spray water at inlet can be adjusted dynamically, the cooling performance can be adjusted between pure Nitrogen and water quenching to meet the requirement for different microstructure.

The flow field is meshed by hexahedron structural grid, and 3D grid of flow field in furnace is shown in figure 2. In figure 2, the workpiece at outlet is considered. There are about 2500 000 elements and 400 000 nodes. The boundary layer is considered at all the solid boundary, such as the collection loop, holes inside of inner canister, holes under inner canister, inner and outside canister.

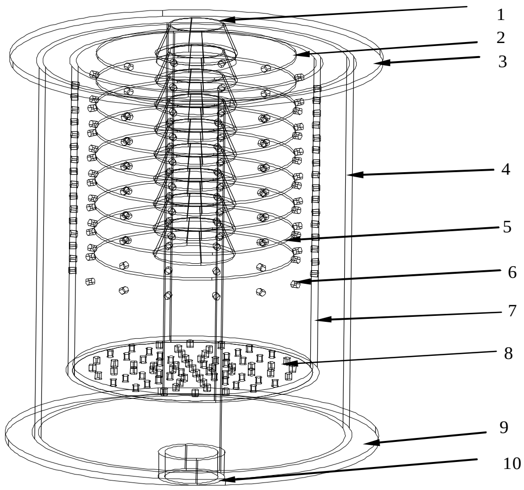

Fig.1 Diagram of quenching furnace

1-outlet, 2-cover, 3-upper cover, 4-outside canister, 5-collection loop, 6-hole in side of inner canister, 7- inner canister, 8- hole under inner canister, 9-down cover, 10-inlet

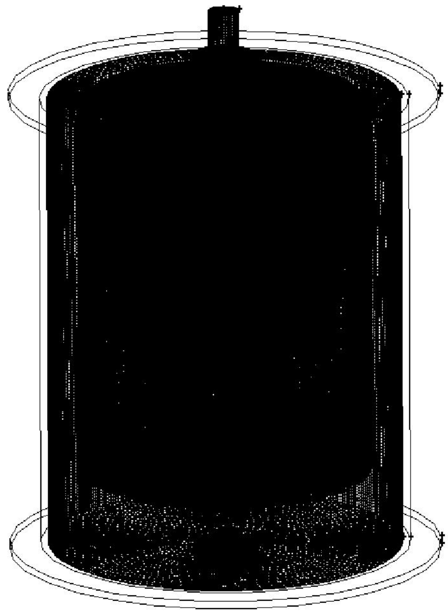

Fig.2 3D meshed model of quenching furnace

III. Control equation

The two-phase flow in furnace is simulated by Mixture model. The continuity equation for the mixture model is

$$\frac{\partial}{\partial t}(\rho_m)+\nabla\cdot(\rho_m\vec{v}_m)=\dot{m} \tag{1}$$

Where $\vec{v}_m$ is mass mean velocity, which can be expressed as

$$\vec{v}_m=\frac{\sum_{k=1}^{n}\alpha_k\rho_k\vec{v}_k}{\rho_m} \tag{2}$$

Where ρ_m is mix density, which can be expressed as

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \tag{3}$$

Where α_k is the volume percent of No. k phase.

The momentum equation for the mixture model is

$$\frac{\partial}{\partial t}(\rho_m \vec{v}_m) + \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot \left[\mu_m \left(\nabla \vec{v}_m + \nabla \vec{v}_m^{T}\right)\right] + \rho_m \vec{g} + \vec{F} + \nabla \cdot \left(\sum \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k}\right) \tag{4}$$

Where n is number of phase, $\vec{F}$ is volume force, μ_m is viscosity of the mixture

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \tag{5}$$

$\vec{v}_{dr,k}$ is slip velocity of the second phase k.

$$\vec{v}_{dr,k} = \vec{v}_k - \vec{v}_m \tag{6}$$

Relative (slip) and excursion velocity:

The relative velocity is defined as the velocity of secondary phase (p) relatived to the velocity of primary phase (q).

$$\vec{v}_{qp} = \vec{v}_p - \vec{v}_q \tag{7}$$

The relationship of the xcursion and relative velocity is

$$\vec{v}_{dr,p} = \vec{v}_{qp} - \sum_{k=1}^{n} \frac{\alpha_k \rho_k}{\rho_m} \vec{v}_{qk} \tag{8}$$

The algebra slip formula is adopted, which supposes that the relationship of relative velocity is algebra, and the local equilibrium among phases can achieve at short scale. The format of relative velocity is

$$\vec{v}_{qp} = \tau_{qp} \vec{\alpha} \tag{9}$$

Where $\vec{\alpha}$ is the acceleration of the secondary phase particle, τ_{qp} is the relaxation time of particle. According to Manninen theory, the format of τ_{qp} is

$$\tau_{qp} = \frac{(\rho_m - \rho_p) d_p^2}{18 \mu_q f_{drag}} \tag{10}$$

Where d_p is the diameter of secondary phase particle (or air bubble, or liquid drop), the function f_{drag} from Schiller and Naumann:

$$f_{drag} = \begin{cases} 1 + 0.15\,\mathrm{Re}^{0.687} & \mathrm{Re} \le 1000 \\ 0.0183\,\mathrm{Re} & \mathrm{Re} > 1000 \end{cases} \tag{11}$$

The acceleration $\vec{\alpha}$ is

$$\vec{\alpha} = \vec{g} - (\vec{v}_m \cdot \nabla \vec{v}_m) - \frac{\partial \vec{v}_m}{\partial t} \tag{12}$$

The simplest algebra slip formula is excursion flux model, in which the acceleration of particle is given by gravitation or centrifugal force, and the relaxation time is modified to consider the other particles.

The volume equation of secondary phase p can be deduced from the continuity equation of secondary phase p,

$$\frac{\partial}{\partial t}(\alpha_p \rho_p) + \nabla \cdot (\alpha_p \rho_p \vec{v}_m) = -\nabla \cdot (\alpha_p \rho_p \vec{v}_{dr,p}) \tag{13}$$

IV. PROCESS OF NUMERICAL SIMULATION

The numerical simulation is conducted by CFD software Fluent6.3. The diameter of inlet is 40mm, and the velocity of initial mixture is about 100m/s, and the viscosity of mixture is about 0.0005, so the approximate Re number is about 8000, and the k-ε (2 equations) model is used for turbulent analysis. By calculating cursorily, the Mach number is larger than 0.3, so the mix fluid is treated as incompressible. During simulation, Nitrogen is primary phase, and spray water is secondary phase, and spray water is treated as droplet which diameter is 0.01mm. At inlet, different velocities of Nitrogen and spray water are set to give a mixture boundary condition. At outlet, the pressure is normal air pressure. The iterative number is 1000.

V. RESULTS OF NUMERICAL SIMULATION

Because of the symmetrical furnace, the figures are the plane cut at centre along axes. Fig.3-Fig.7 is velocity and pressure fields when the velocity of Nitrogen and spray water at inlet of furnace is 170m/s and 150m/s, respectively, and the water droplet volume percent is 10%. Because the density of spray water is larger than that of Nitrogen, the velocity of spray water is quicker than that of Nitrogen, though the initial velocity of Nitrogen is larger than that of spray water. Fig.8-Fig.13 is velocity and pressure fields when the water droplet volume percent is 1%.

From Fig.3-Fig.5, when the initial velocity is about 150m/s, the outlet velocity is about 450m/s. The simulation results show that the quenching furnace can improve the velocity of quenching media. Fig.4 and Fig.5 show the static and dynamic pressure in flow field, respectively. At outlet, the static pressure is least, but the dynamic pressure is maximum.

Compare the maximum velocity of Fig.3 and Fig.8, when the volume percent of water droplet is less, the maximum velocity at outlet is more. Because that the inlet velocity of Nitrogen is larger than that of spray water.

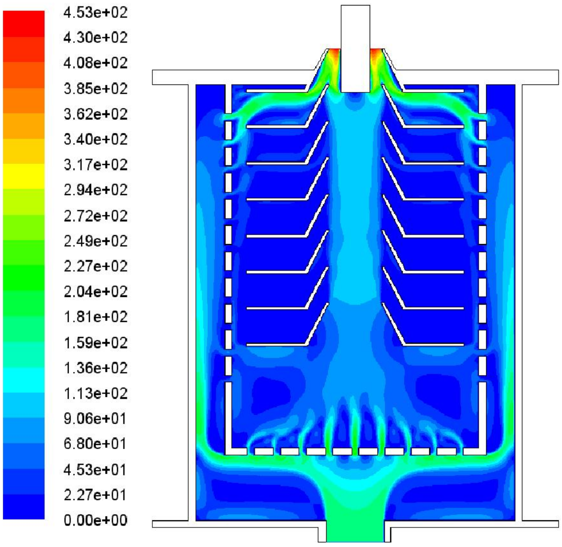

Fig.3 Velocity magnitude field of mixture (10%)

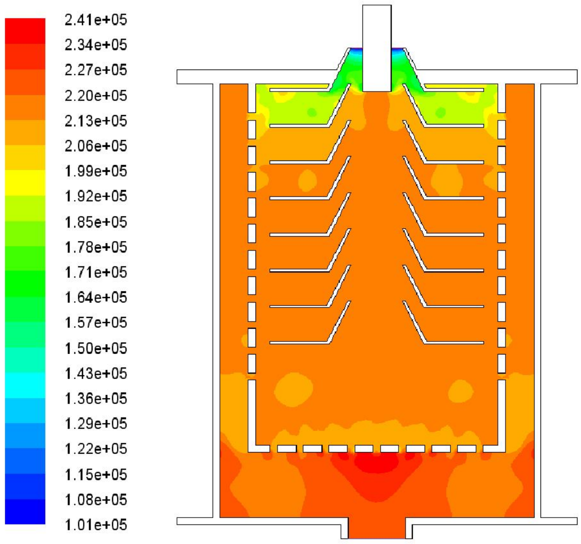

Fig.6 Static pressure field of mixture (10%)

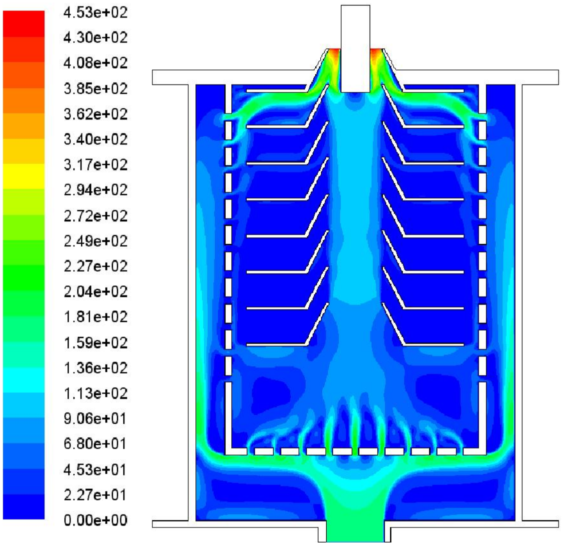

Fig.4 Velocity magnitude field of Nitrogen (10%)

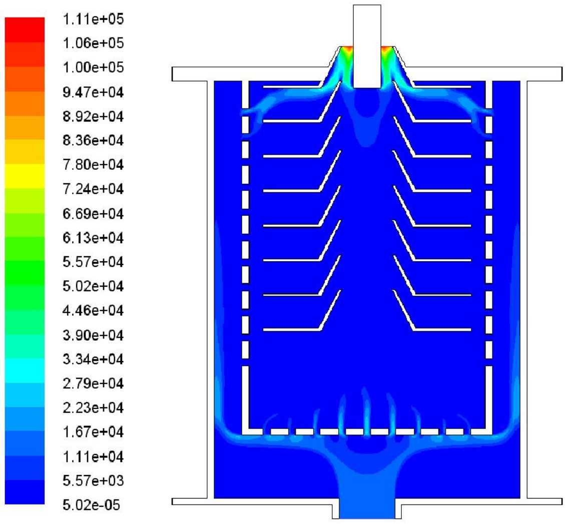

Fig.7 Dynamic pressure field of mixture (10%)

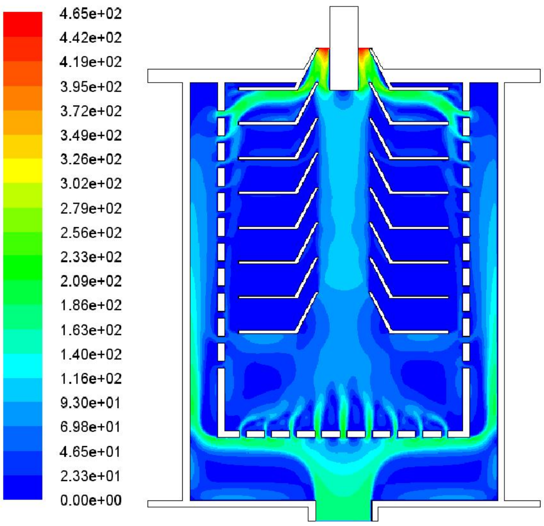

Fig.5 Velocity magnitude field of spray water (10%)

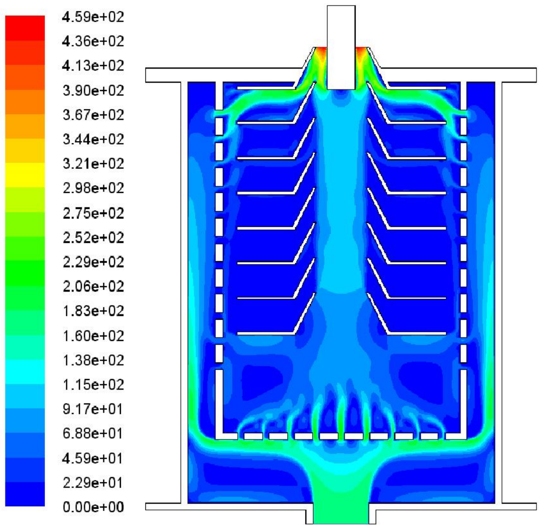

Fig.8 Velocity magnitude field of mixture (1%)

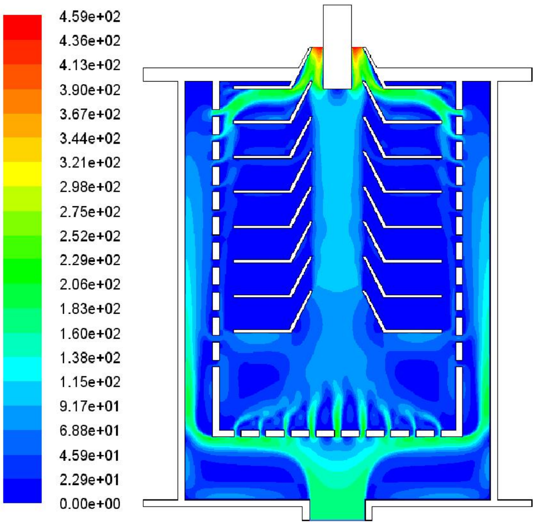

Fig.9 Velocity magnitude field of Nitrogen (1%)

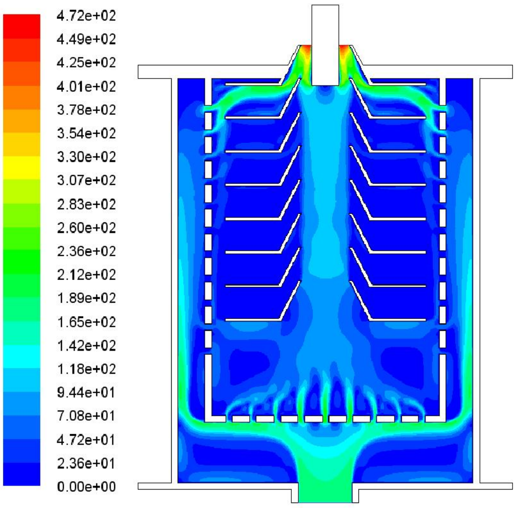

Fig.10 Velocity magnitude field of spray water (1%)

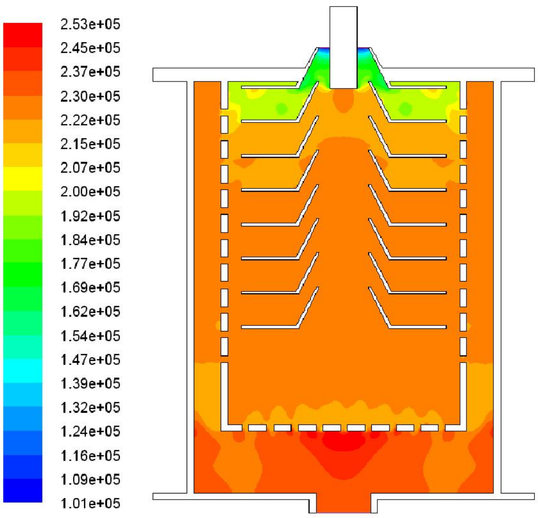

Fig.11 Static pressure field of mixture (1%)

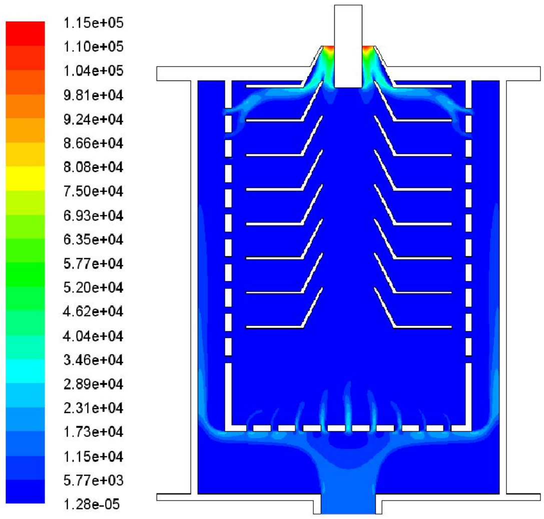

Fig.12 Dynamic pressure field of mixture (1%)

Fig.13-Fig.17 show the velocity and pressure fields when the velocity of Nitrogen and spray water at inlet of furnace is 150m/s and 100m/s, respectively, and the water droplet volume percent is 10%. Fig.18-Fig.22 is velocity and pressure fields when the water droplet volume percent is 1%.

Compare with Fig.3, the maximum velocity magnitude in Fig.13 is less. One reason is that the inlet velocity is less, and another reason is that the slip velocity is larger. The larger slip velocity between two phases, the more kinetic energy cost. The results also show that the outlet velocity is larger than that of inlet, which means the furnace designed can quicken the velocity of flow field.

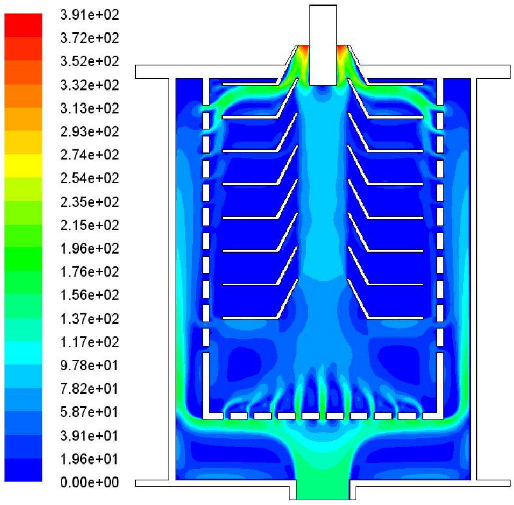

Fig.13 Velocity magnitude field of mixture (10%)

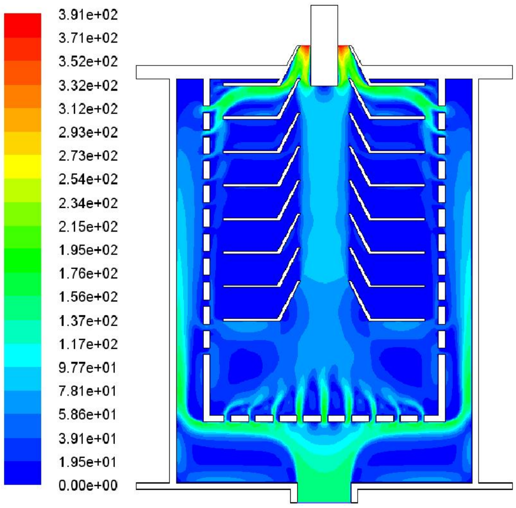

Fig.14 Velocity magnitude field of Nitrogen (10%)

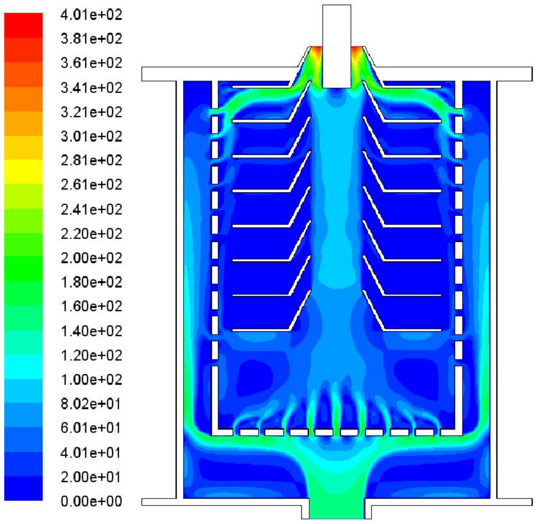

Fig.15 Velocity magnitude field of spray water (10%)

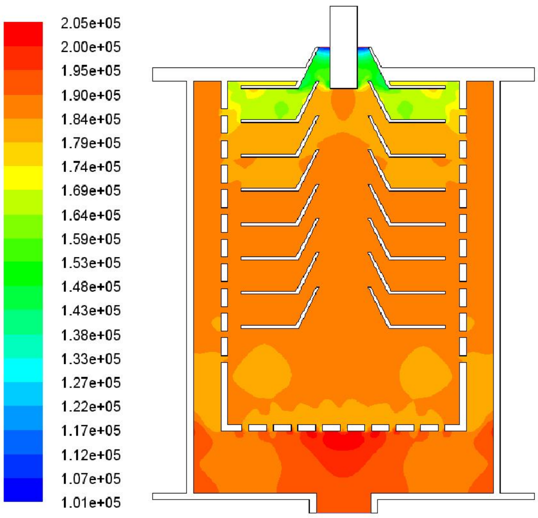

Fig.16 Static pressure field of mixture (10%)

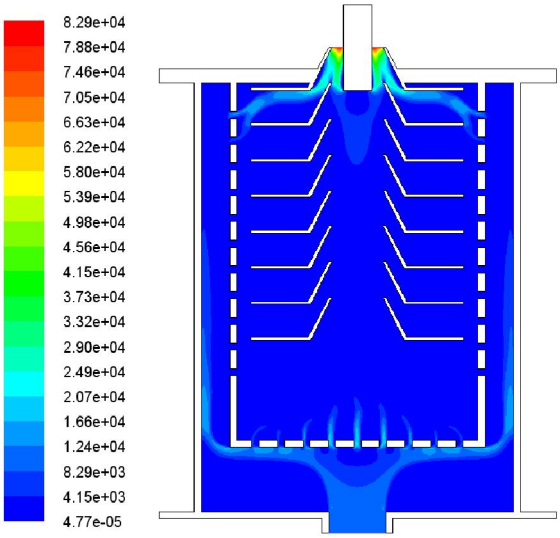

Fig.17 Dynamic pressure field of mixture (10%)

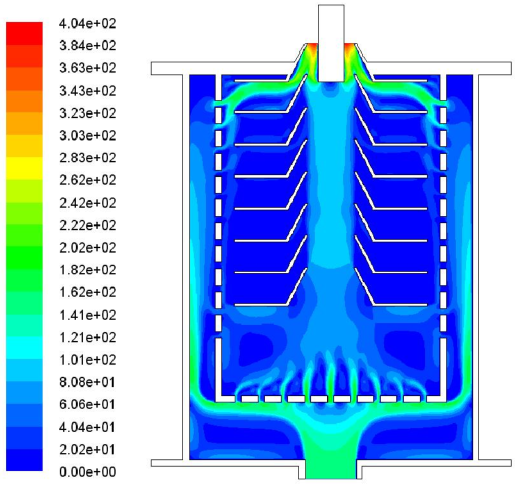

Fig.18 Velocity magnitude field of mixture (1%)

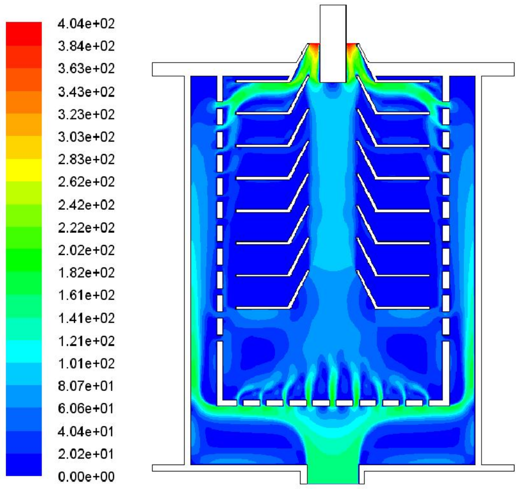

Fig.19 Velocity magnitude field of Nitrogen (1%)

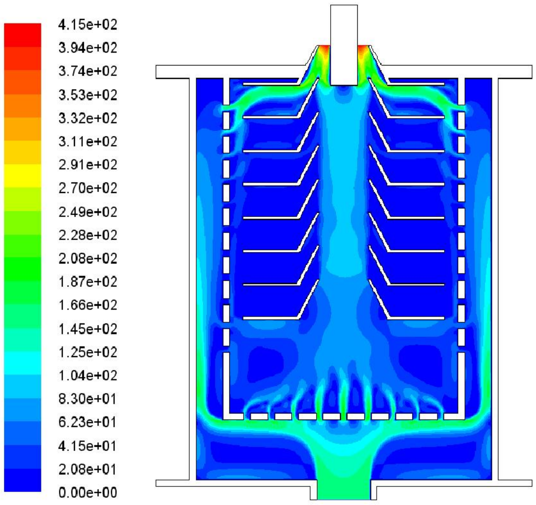

Fig.20 Velocity magnitude field of spray water (1%)

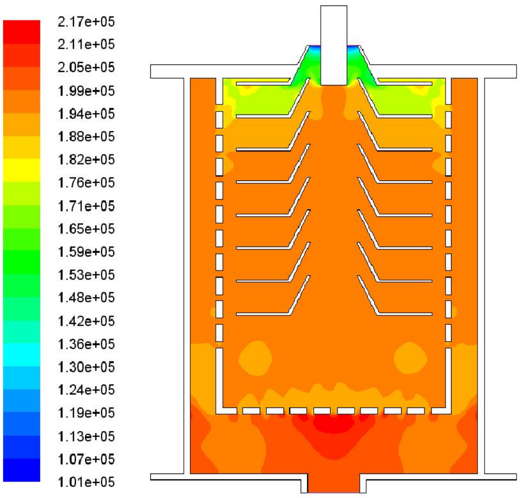

Fig.21 Static pressure field of mixture (1%)

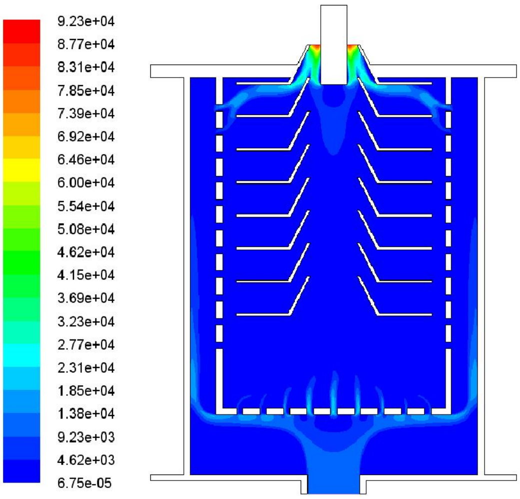

Fig.22 Dynamic pressure field of mixture (1%)

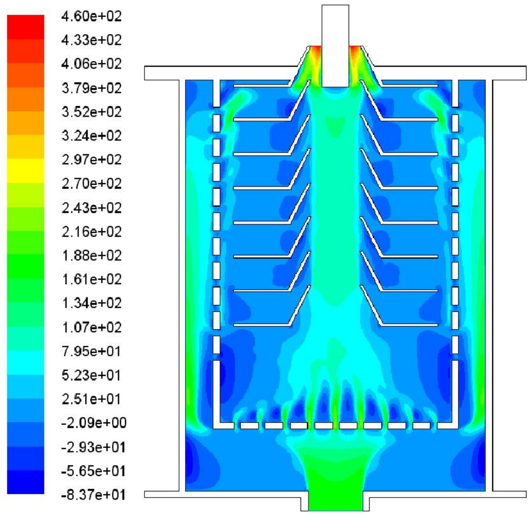

Fig.23 Z direction velocity field of mixture (1%)

Fig.23 is the Z direction velocity field of mixture when the velocity of Nitrogen and spray water at inlet of furnace is 170m/s and 150m/s, respectively, and the water droplet volume percent is 1%. Z direction is the symmetric axial direction of furnace, is also the flow direction of fluid in furnace. Fig.23 shows that the Z direction velocity is negative in corner and beside collection loop inside furnace, which means the mixture is refluent.

VI. Conclusion

The flow field in furnace for mixture of Nitrogen and spray water ejecting quenching under normal pressure and high velocity is complex, somewhere has refluent phenomena.

In two-phase flow problem, the value of slip velocity between two phases has more contribution to the mixture velocity of the two-phase.

When the other condition is same, the less is the volume percent of water droplet, the more is the maximum velocity at outlet.

The numerical simulation results show that the furnace designed can accelerate the velocity of flow field.

For future work, it would be interesting to investigate when the angle of the collection loop is changed, how the flow field will be changed.

Acknowledgment

Baodong Shao thanks the financial support of the Yunnan Key Application Basic Research Foundation (No. 2007A0015Z). Without the financial support, the work would not have been possible process.

References

[1] CHENG Heming, XIE Jianbin, LI Jianyn, "Comparison of properties of 9SiCr alloy steel in various quenching media after quenching", *Journal of Aeronautical Materials*, Vol.24, No.4, 2004, pp. 14-17.

[2] Heck U., Fritsching U., Bauckhage K, "Fluid flow and heat transfer in gas jet quenching of a cylinder", *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol.11, No.1, 2001, pp. 36-49.
[3] F. Faura, A. Campo, B. Zamora, "A mixture of pure gases that produce maximum heat transfer characteristics for quenching", *J. Mater. Eng. Perf.*, Vol.7, No.3, 1998, pp. 420-424.
[4] C. Laumen, S. J. Midea, T. Lubben, F. Hoffmann and P. Mayr, "Measured heat transfer coefficients by using hydrogen as quenchant in comparison with helium and nitrogen", *In SM heat treatment conference. Accelerated Cooling/Direct Quenching of Steels*, Indianapolis, Indiana, USA, 1997, pp. 199-206.
[5] MEI Guohui, MENG Hongji, WU Rongyang, et al, "Analysis of spray cooling heat transfer coefficient on high temperature surface", *Energy for Metallurgical Industry*, Vol. 23, No. 6, 2004, pp. 18-22.
[6] GAO Shan, QU Wei, YAO Wei, "Flow and heat transfer of droplet impinging on hot flat surface during spray cooling", *Journal of Engineering Thermophysics*, Vol. 28, Suppl. 1, 2007, pp. 221-226.
[7] LIU Zhenhua, "Experimental study on the boiling critical heat flux of mist cooling", *Journal of Shanghai Jiaotong University*, Vol. 31, No. 2, 1997, pp. 61-65.
[8] ZHANG Hongxing, MIAO Jianyin, YAO Wei, et al, "Steady-state characteristics of a closed-loop spray cooling system", *Chinese Space Science and Technology*, No. 4, 2009, pp. 61-68.
[9] DU Fengshan, ZHANG Fang, HUANG Huagui, et al, "Numerical simulation of spray quenching process for large-scale forgings", *Metal Heat Treatment*, Vol. 33, No. 5, 2008, pp. 35-38
[10] MEHMET SHALA, MAYUR PATEL, KOULIS PERICLEOUS, "Unstructured staggered mesh methods for fluid flow, heat transfer and phase change", *WSEAS Transactions on Fluid Mechanics*, Vol.1, No.5, 2006, pp. 431-438
[11] VASOS PAVLIKA, "A numerical design technique associated with the design of axisymmetric ducts for incompressible rotational flow", *WSWAS Transactions on Fluid Mechanics*, Vol. 1, No. 11, 2006, pp. 982-993
[12] FRIEDRICH K. BENRA and SIAVASH H. SOURAB, "Modified theory of laminar flow around rigid and liquid cylinders", *WSEAS Transactions on Fluid Mechanics*, Vol. 1, No. 6, 2006, pp. 533-541
[13] MAHDI HAMZEHEI, HASSAN RAHIMZADEH, MOHAMMAD SAEEDI, "Visualization of fluid flow and measurement of temperature variation at various conditions in the open channels", *WSEAS Transactions on Fluid Mechanics*, Vol. 1, No. 9, 2006, pp. 900-906
[14] DAMELYS ZABALA, AURA L. LÓPEZ DE RAMOS, "Solute concentration effect in gas-liquid interface displacement modeling", *WSEAS Transactions on Fluid Mechanics*, Vol. 1, No. 69, 2006, pp. 686-691
[15] SERKAN ÖZGEN, "Mode coalescence in two-fluid boundary layer stability", *WSEAS Transactions on Fluid Mechanics*, Vol. 1, No. 69, 2006, pp. 692-697

Baodong Shao, come from Heilongjiang province, China, 1971-12-21.

He has been a teacher of Kunming University of Science and Technology since 2001, the published articles are listed below:
Baodong Shao, Lifeng Wang, Jianyun Li, "Application of thermal resistance network model in optimization design of micro-channel cooling heat sink", *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 19, No. 3/4, 2009, pp. 535-545
Baodong Shao, Zhaowei Sun, Lifeng Wang, "Optimization design of microchannel cooling heat sink", *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 17, No. 6, 2009, pp. 628-637
Baodong Shao, Zhaowei Sun, "Non-equilibrium molecular dynamics simulation of the thermal conductivity of crystals film", *Aircraft Engineering and Aerospace Technology*, Vol. 78, No. 2, 2006, pp. 138-141