Article ID: 1003 - 6326(2004)05 - 0940 - 05

Turbulence flow and heat transfer of aluminum melt in tip cavity in process of thin-gauge high-speed casting①

LIU Xiao-bo(刘晓波)[1], MAO Da-heng(毛大恒)[2], ZHONG Jue(钟 掘)[2]
(1. Faculty of Mechanical and Electronic Engineering, Southern Institute of Metallurgy, Ganzhou 341000, China;
2. College of Mechanical and Electrical Engineering, Central South University, Changsha 410083, China)

Abstract: The uniformity of flow distribution of aluminum melt in tip cavity is a precondition to decide whether or not thin-gauge high-speed casting can be accomplished smoothly. The laws of aluminum melt flow and heat transfer in tip cavity can be found out through numerical simulation, which gives theoretical basis for solving the problem of the flow distribution of melt in tip cavity. A mathematical model with a low Reynolds number $k-\varepsilon$ model for turbulence flow and heat transfer of aluminum melt in tip cavity was developed. The finite difference method was used to calculate the flow field and temperature field of aluminum melt in tip cavity. The phenomena and characteristics of turbulence flow and heat transfer were analyzed, including the characteristics of temperature distribution of turbulence similar to that of laminar flow. The simulation results are in good agreement with the experimental results for flow velocities and temperature at the exit of tip, which verifies the validity of the simulation results.

Key words: thin-gauge high-speed casting; tip cavity; mathematical model; turbulence; heat transfer
CLC number: TG 29; TG 33 **Document code:** A

1 INTRODUCTION

In the process of twin roll casting of aluminum strip, the uniformity of flow distribution in the tip cavity has a great influence on the quality of strip[1], such as flatness, heat line/voids, surface crackle/wrinkles. These defects are all due to non-uniformity of flow distribution at the exit of tip along width direction. In the process of high-speed thin-gauge casting, the gap between rollers is decreased, while the speed of casting is increased greatly, which results in the flow state in tip cavity more complicated, and the flow resistance and the flow velocity distribution taking great changes. At the same time, the loss of heat of aluminum melt in tip cavity takes corresponding changes, which results in the temperature distribution of melt taking changes. Therefore, solving the problem of flow distribution of melt in tip cavity is a key to getting uniform velocity and temperature of melt at the exit of tip cavity, which is a precondition of realizing thin-gauge high-speed casting and getting top-quality strip.

Two methods are considered to investigate the melt flow and heat transfer in tip cavity. One is water simulation[2-6]; the other is numerical simulation[7-10], which has been widely used because it is economical, repeatable and liable to control and optimize. A 2-D mathematical model for aluminum melt flow in tip cavity was developed[7] and a 2-D mathematical model for aluminum melt flow and heat transfer in tip cavity was presented[8]. The aluminum melt flow and heat transfer in tip cavity was simulated by computer in Ref. [9]. Because the thickness of tip cavity is much less than its width and length, the lower and upper walls have a great influence on the melt flow and heat transfer, which are three-dimensional. In Ref. [10], a 3-D mathematical model for aluminum laminar flow in tip cavity was developed. While in the thin-gauge high-speed casting, the melt flow in tip cavity is turbulence[11]. A mathematical model for turbulence flow and heat transfer will be developed in this paper to investigate the melt flow and heat transfer in tip cavity by numerical simulation, which contributes greatly to obtaining reasonable and optimal geometry structure of tip cavity to realize steady, high-efficient, high-speed and continuous casting.

2 MATHEMATICAL MODEL

2.1 Governing equations

The structure of tip cavity is shown in Fig. 1. For developing the model, the assumptions of melt

① **Foundation item:** Project(G1999064106) supported by the National Basic Research Program of China; project (02081) supported by the Key Science and Technology Research Project of State Education Department; project (0250020) supported by the Natural Science Foundation of Jiangxi Province, China
Received date: 2003 - 07 - 16; **Accepted date:** 2004 - 05 - 20
Correspondence: LIU Xiao-bo, Professor, PhD; Tel: + 86-797-8312062; E-mail: liuxiaobo2000@21cn.com

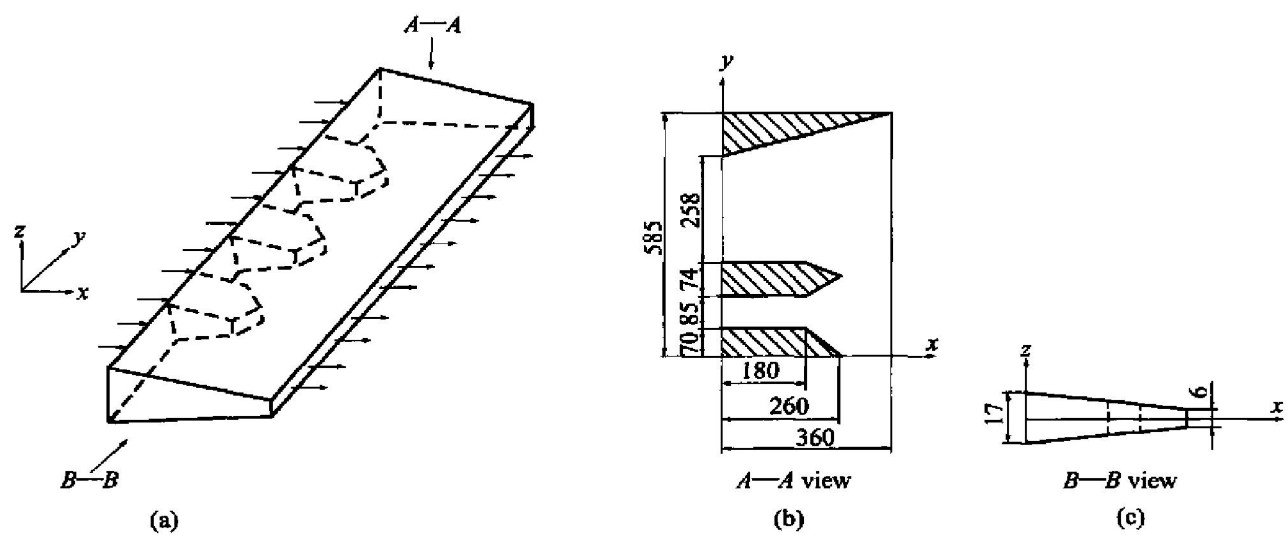

Fig. 1 Structure of tip cavity (unit: mm)

in tip cavity are made as follows:

1) The melt is incompressible and steady;

2) The melt is Newtonian fluid;

3) The flow melt is with generalized Boussinesq eddy viscosity.

Based on above assumptions, a mathematical model for melt flow and heat transfer in tip cavity can be solved by tensor in continuity equation, Navier-Stokes equation and energy equation.

1) Continuity equation

$$\frac{\partial(\rho u_j)}{\partial x_j} = 0 \qquad (1)$$

2) N-S equation

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = \rho F_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu_l + \mu_t)\left[\frac{\partial u_i}{\partial x_j}\right]\right] \qquad (2)$$

$$\mu_t = C_u \rho \frac{k^2}{\varepsilon} \qquad (3)$$

3) Energy equation

$$\frac{\partial(\rho c_p u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(k_l + k_t)\frac{\partial T}{\partial x_j}\right] \qquad (4)$$

$$k_t = \rho c_p \frac{C_u k^2}{Pr\ \varepsilon} \qquad (5)$$

The melt flow in tip cavity is turbulence, while the melt flow becomes quasi-laminar flow at the exit of tip cavity[11], and the turbulence model adopts the low Reynolds number k—ε model[12]. The k equation and ε equation are as follows:

$$\frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu_l + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \rho\varepsilon - 2\mu_l\left(\frac{\partial\sqrt{k}}{\partial x_j}\right)^2 - 2\frac{\mu_l}{k}k \qquad (6)$$

$$\frac{\partial}{\partial x_j}(\rho u_j \varepsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu_l + \frac{\mu_t}{\sigma_s}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 f_1 \frac{\varepsilon}{k}\mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - C_2 f_2 \rho\frac{\varepsilon^2}{k} + 2\mu_l\mu_t\left(\frac{\partial^2 u_j}{\partial x_j \partial x_k}\right)^2 - 2\frac{\mu_l}{k}\varepsilon \qquad (7)$$

And the u_t is optimized as:

$$\mu_t = C_u f_u \rho \frac{k^2}{\varepsilon} \qquad (8)$$

$$\begin{cases} f_u = \exp\left[\dfrac{-3.4}{\left(1 + \dfrac{Re}{50}\right)^2}\right] \\ f_1 = 1 \\ f_2 = 1.0 - 0.3\exp(-Re^2) \end{cases} \qquad (9)$$

$$Re = \frac{\rho k^2}{\mu_l \varepsilon} \qquad (10)$$

$C_u = 0.09$, $\sigma_k = 1.0$, $\sigma_s = 1.3$, $C_1 = 1.44$, $C_2 = 1.92$.

2.2 Boundary conditions

According to the characteristics of tip cavity, the boundary conditions with Cartesian coordinates are proposed as follows:

1) Conditions of inlet:

$$u_x = \bar{u},\ u_y = u_z = 0,\ \frac{\partial p}{\partial y} = 0,$$

$$k = 0.005(u_x^2 + u_y^2 + u_z^2),$$

$$\varepsilon = \frac{k^{1.5}}{0.3r},\ T = T_{in}$$

2) Conditions of exit:

$$\frac{\partial u_x}{\partial x} = 0,\ \frac{\partial u_y}{\partial x} = 0,\ \frac{\partial u_z}{\partial x} = 0,\ p = \bar{p},$$

$$\frac{\partial k}{\partial x} = 0,\ \frac{\partial \varepsilon}{\partial x} = 0,\ k\frac{\partial T}{\partial n} = q_{wall}$$

3) Conditions of wall:

$$u_{x\ wall} = u_{y\ wall} = u_{z\ wall} = 0,$$

$$\frac{\partial p}{\partial n} = 0,\ k_{wall} = 0,\ \varepsilon_{wall} = 0,\ k\frac{\partial T}{\partial x} = q_1$$

4) Conditions of symmetric area:

$$\frac{\partial u_x}{\partial y} = 0,\ u_y = 0,\ \frac{\partial u_z}{\partial y} = 0,\ \frac{\partial p}{\partial y} = 0,$$

$$\frac{\partial k}{\partial y}=0,\ \frac{\partial \varepsilon}{\partial y}=0,\ \frac{\partial T}{\partial y}=0$$

where u_x, y_y, u_z denote the components of velocity vector in x, y, z directions respectively, m/s; p is pressure, Pa; ρ is density, kg/m^3; μ_l is the kinetic viscosity coefficient of melt, Pa·s; $\overline{u}$ is the component of inlet velocity vector in x direction, m/s; $\overline{p}$ is the exit pressure, Pa; k is the turbulent fluctuation kinetic energy, m^2/s^2; ε is the dissipation rate of turbulent fluctuation kinetic energy, m^2/s^3; k_l is the thermal conductivity of melt, W/(m·K); k_t is the thermal conductivity of turbulence, W/(m·K); c_p is the specific heat capacity of melt at constant pressure, kJ/(kg·K); T is temperature, K; T_{in} is inlet temperature, K; q_l is the heat transfer coefficient, W/m^2; q_{wall} is the heat transfer coefficient between melt and walls, W/m^2; r is the hydraulic radius of inlet section in tip cavity, m; Re is the Reynolds number of turbulence; Pr is the Prandtl number of turbulence.

3 NUMERICAL SIMULATION AND ANALYSIS OF VELOCITY AND TEMPERATURE OF MELT IN TIP CAVITY

3.1 Numerical methods and solutions

The control volume integral method is used to establish the finite difference equations in this paper. The staggered grid and hyrid difference scheme are used for discretization, and the SIMPLE algorithm is used to solve the equations. The method of additional source term is used to keep the velocity in solid zone as $u_x = u_y = u_z = 0$, which makes the velocity of solid nearly zero. This method is better than Blooking-off method[13], for the velocity in solid zone is still high if Blooking-off method is used. This doesn't agree with the actual condition, and at the same time the results in the velocity field of the simulated solid zone are invalid. Compared with the body-fitted coordinate method, the calculation process of this method is much simpler.

Being the tip cavity of symmetry, a half domain is used to calculate. The calculation domain is divided into 6 240 control volumes, 26, 30 and 8 grid units along x, y and z directions respectively.

3.2 Results and discussion

The flow field, temperature field and pressure field of aluminum melt in tip cavity (as shown in Fig. 1) are calculated in this paper. The parameters used for calculation are as follows:

$\rho = 2.371\times 10^3$ kg/m^3, $\mu = 1.22\times 10^{-3}$ Pa·s, $\overline{u} = 0.049$ m/s, $\overline{p} = 0$, $c_p = 1.08\times 10^3$ kJ/(kg·K), $k = 1.03\times 10^2$ W/(m·K), $Pr = 3.5$, $T_{in} = 958$ K, $q_l = 2.13\times 10^5$ W/m^2, $q_{wall} = q(x)$[14].

The calculated OM (order of magnitude) of u_z is 10^{-3}–10^{-4} m/s, while the OM of u_x was 10^{-2} m/s. Compared with u_x, u_z is neglectable. The calculated temperature distribution of melt is symmetrical along the section centerline. The melt temperature increases gradually from the upper and lower walls to the middle, and the maximum does not surpass 3 ℃. However, the temperature distribution of aluminum melt is different from that of general turbulence. It is the same as the temperature distribution of laminar flow and takes on arc distribution, which is due to the aluminum melt with low Pr number, and the heat transfer diffuses more quickly than momentum does, which in turn makes the heat diffusion of molecule very active even in turbulence. Based on above reasons, the temperature distribution of turbulence is similar to that of laminar flow. The calculated difference of melt pressure from the upper and lower walls to the middle is very little, which is nearly zero.

Considering that the changes of velocity, temperature and pressure along x, y directions are much greater than those along thickness direction, we only investigated the characteristics of melt flow and heat transfer in $x-y$ plane.

The velocity vector diagram calculated by the model in $x-y$ plane is shown in Fig. 2(a). From the figure, we know the melt flows towards x direction between the fore half section (straight zone) of two diffluent blocks, which indicates that u_y is much smaller than u_x and they are not of the same OM. Therefore, the influence of u_y on flow is very little. The melt in rear section (wedge zone) flows towards two sides, which indicates the influence of u_y on flow becomes great. After passing the diffluent blocks, the melt gradually flows towards x direction, which indicates that the influence of u_y on flow is becoming little with it leaving diffluent blocks.

The dividing line of the melt flow between side diffluent block and side wall is the mid line of this zone. The flow adjacent to side diffluent block is much the same as that between two diffluent blocks. The melt adjacent to side wall flows along side wall. With it approaching the side wall, the u_y influences the flow very much.

The temperature contour diagram of melt flow calculated by the model in $x-y$ plane is shown in Fig. 2(b). From the figure, we know that the temperature in tip cavity gradually gets low along flowing direction. In the fore half section (straight zone) between two diffluent blocks except near diffluent blocks, the temperature distribution of melt along width direction is uniform. In the half rear section (wedge zone), the temperature distribution of melt gets uniformly proportionally low. After passing the diffluent blocks, the temperature distribution of melt gets uniformly proportionally high rapidly. Although

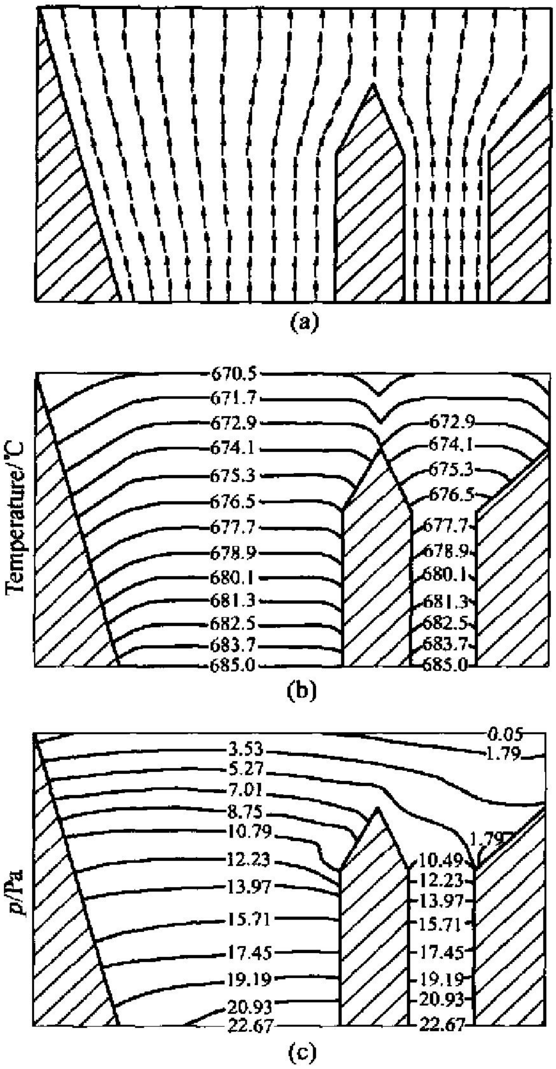

Fig. 2 Velocity vector(a) , temperature contour(b) and pressure contour(c) diagrams of melt flow in tip cavity ($x - y$ plane)

the temperature of melt in the wake zone of diffluent blocks is much lower than mid temperature, the aluminum melt is with high thermal conductivity and low *Pr* number, and at the same time the temperature diffusivity is much higher than velocity diffusivity. Therefore, the melt temperature in wake zone of diffluent blocks restitutes more rapidly than velocity does. The temperature distribution of melt in most zone between side diffluent block and side wall is much the same as that between two diffluent blocks. However, the difference between the side temperature and mid temperature becomes larger and larger with the melt flowing from the inlet to exit, which is due to heat transferring through not only upper and lower plates but also side walls.

The pressure contour diagram of melt flow calculated by the model in tip cavity ($x - y$ plane) is shown in Fig. 2(c) . From the figure, we know that the difference of pressure is larger than that in conventional casting, which can reach 22. 67 Pa. It is because that the melt flow is turbulence, which is of high viscosity. In order to overcome the resistance of viscosity (including the resistance along the way and local resistance) , the pressure suffers great losses.

The experimental and simulation results of velocity and temperature distribution of melt at the exit of tip cavity are shown in Fig. 3. Curve 2 and curve 1 denote the experimental results and simulation results respectively, which are calculated by the turbulence model developed in this paper.

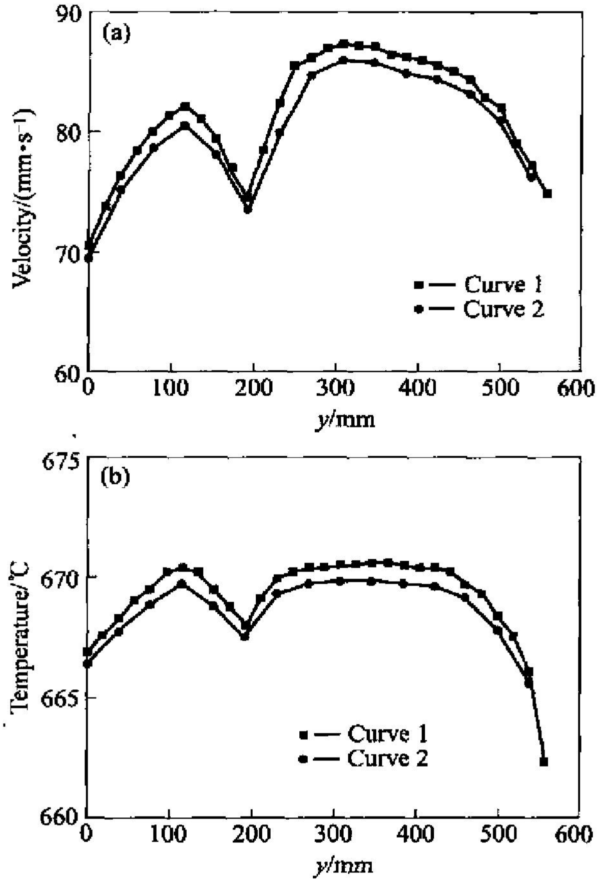

Fig. 3 Velocity(a) and temperature(b) distribution of melt curves at exit of tip cavity

From Fig. 3(a) , we know the trend of velocity distribution calculated by the model agrees with that of experiment results. The side speed of melt is higher than that of the mid speed; the mid speed of melt is higher than that of melt passed diffluent blocks. The relative error of speed of curve 1 is 13. 6%, and the curve 1 agrees with the curve 2 in general. From Fig. 3(b) , we know the trend of temperature is the same as that of velocity, and the results by the model agree with the experimental results.

In the process of thin-gauge high-speed casting, the relative error of melt speed at the exit of tip cavity should be less than 5%. At the same time, the difference of temperature along width direction at the exit of tip cavity should be less than 5 ℃. Obviously, the existing standard tip used for calculation in this paper can not satisfy the rigid technological requirement of thin-gauge high-speed casting, and the tip cavity should be redesigned. We can find out the laws of melt flow and heat transfer in tip cavity by the simulation experiments with the model developed in this paper, which gives basis to reasonable and optimal design for tip cavity.

4 CONCLUSIONS

In the process of thin-gauge high-speed casting, the melt flow at the exit of tip cavity is quasi-laminar flow. According to this characteristic, a mathematical model with a low Reynolds number $k-\varepsilon$ model for turbulence flow and heat transfer in tip cavity was developed. The finite difference method was used to calculate the velocity field and temperature field of the aluminum melt in tip cavity. The phenomena and characteristics of turbulence flow and heat transfer were analyzed, including the characteristics of the melt temperature distribution of turbulence similar to that of laminar flow. It is very helpful to obtain the essence of melt flow and heat transfer in tip cavity in the process of thin-gauge high-speed casting. The simulation results of turbulent flow and heat transfer model are in good agreement with the experimental results for flow velocities and temperature at the exit of tip. Therefore, the laws of melt flow and heat transfer through the numerical simulation can be find out with the model, which is helpful for tip-cavity-designers in making reasonable optimal design and solving problem of the flow distribution in the tip cavity that enables continuous aluminum production for thin gauge at high speed.

REFERENCES

[1] LI B Q. Producing thin strips by twin-roll casting —part I: process aspects and quality issues[J] . JOM, 1995, 5: 29 - 33.
[2] Merchant H D, Kattamis T Z, Morris I G. Continuous Casting of Aluminum Alloys, Continuous Casting of Nonferrous Metals and Alloys[M]. Merchant H D, et al ed. Norwalk: TMS Press, 1989. 60.
[3] Flowers J, Romanowsk C, Smith D. Continuous Casting Aluminum Alloy[P]. US 4751958. 1988.
[4] Beals R, Duvvuri R, Hamer S, et al . Recent innovations in advanced casting technology[A]. Light Metals [C]. San Antonio, 1998. 1161 - 1167.
[5] Hamer S, Smith D, Yildizbayrak G, et al. Twin roll casting of aluminum at 2.5 mm gague production experience and process improvements[A]. Light Metals[C]. San Diego, 1999. 931 - 937.
[6] LIU Xiao-bo, MAO Da-heng, ZHONG Jue. Effect of nozzle spacer for roll strip casting on cavity fluid field[J]. Light Alloy Fabrication Technology, 2000, 28(10): 18 - 21. (in Chinese)
[7] YU H. A fluid mechanics model of the planar flow melt spinning process under low Reynolds number conditions [J]. Met Trans B, 1987, 18B: 557 - 563.
[8] Akdeniz M V, Yildizbayrak G, Suzen F. Computer simulation of metal feeding system in the direct production of thin strip of aluminum alloy by twin-roll casting[A] . 9th Proceeding of the International Metallurgy and Meter-Ials Congress[C]. Istanbul: 1997. 1519 - 1524.
[9] Sarioglu K, Yildizbayrak G, Dunder M. Computer simulation of metal feeding system used in twin roll casting [A]. Light Metals[C]. San Diego, 2000. 663 - 666.
[10] LIU Xiao-bo, DUAN Ji-an, DENG Gu-ling, et al. 3D flow analysis of nozzle cavity for twin roll casting[J]. Journal of Xiangtan Mining Institute, 2000, 15(1): 43 - 47. (in Chinese)
[11] LIU Xiao-bo, CHEN Li-an. Flow state analysis of aluminium melt in nozzle cavity[J]. Hunan Nonferrous Metals, 2001, 17(5): 12 - 14. (in Chinese)
[12] Launder B E, Sharma B I. Application of energy dissipation model of turbulence to the calculation of flow near a spinning disc[J]. Letters in Heat and Mass Transfer, 1974, 1: 131 - 138.
[13] Patankar S V. Numerical Heat Transfer and Fluid Flow [M]. Guo Kuan-liang transl. Anhui: Anhui Science and Technology Press, 1984. 149. (in Chinese)
[14] LIU Xiao-bo, DENG Gu-ling, MAO Da-heng. Analysis of combined heat transfer in nozzle cavity boundary for aluminium roll casting[J] . Light Alloy Fabrication Technology, 2001, 29(5): 17 - 19. (in Chinese)

(Edited by YUAN Sai-qian)