CFD SIMULATION OF SUPERSONIC OXYGEN JET BEHAVIOUR INSIDE A HIGH TEMPERATURE FIELD

Morshed ALAM[1], Jamal NASER[1] and Geoffrey A. BROOKS[1]

[1] Faculty of Engineering and Industrial Sciences, Swinburne University of Technology, Hawthorn, Victoria 3122, AUSTRALIA

ABSTRACT

In this study, a CFD model was developed to investigate the effect of high ambient temperature field on the supersonic oxygen jet behaviour. The results were compared with available experimental data by Sumi et al. (2006) and a jet model proposed by Ito and Muchi (1969). The CFD simulation with k-ε turbulence model including compressibility terms was found to under predict potential flow core length at higher ambient temperatures. At high ambient temperatures, the density of the ambient atmosphere is low which results in reduction of the growth rate of mixing. The k-ε turbulence model with compressibility correction can duplicate the reduction in growth rate of mixing at low ambient temperatures but fails to accurately model the decrease in growth rate of mixing at high ambient temperatures. A modified k-ε turbulence model was presented which modifies the turbulent viscosity in order to reduce the growth rate of mixing at high ambient temperatures. The CFD simulation was performed by using commercial CFD software AVL FIRE 2008.2.

NOMENCLATURE

P	pressure (N/m^2)
U	mean velocity (m/s)
u	fluctuating velocity (m/s)
k	turbulent kinetic energy (m^2/s^2)
E	total energy (J)
T	mean temperature (K)
t	fluctuating component of temperature (K)
K	thermal conductivity (W/mK)
q	conduction heat flux (W/m^2)
Pr_t	turbulent prandtl number (-)
M_τ	turbulence Mach number (-)
T_t	total temperature (K)
a	sound speed (m/s)
l	turbulence length scale (m)
S	mean shear rate (s^{-1})
C_P	specific heat (J/kg.k)
d_e	nozzle exit diameter (m)
ρ	density (kg/m^3)
μ	molecular viscosity (Ns/m^2)
μ_t	turbulent viscosity (Ns/m^2)
ε	turbulence dissipation rate (m^2/s^3)

Subscript

e	nozzle exit
a	ambient

INTRODUCTION

Supersonic gas jets are widely used in Basic Oxygen Furnace (BOF) and Electric Arc Furnace (EAF) steelmaking for refining the liquid iron inside the furnace. Supersonic gas jets are preferred over subsonic jets because of high dynamic pressure associated with it which results in higher depth of penetration and better mixing. Laval nozzles are used to accelerate gas jets to supersonic velocity of around 2.0 Mach number in steelmaking (Deo and Boom, 1993). The supersonic gas jets generate droplets upon impingement on liquid iron which is known as splashing. Droplet generation has both beneficial and detrimental effects. The droplet increases the interfacial area which in turn increases the refining rate (Subagyo, Brooks, Coley and Irons, 2003). On the other hand, it may cause wearing of refractories, skulling on the mouth of the vessels and lances which can result in loss of production (McGee and Irons, 2002, Peaslee and Robertson, 1994, Luomala, Fabritius, Virtanen, Siivola and Harkki, 2002). Therefore, it is necessary to understand the behaviour of the supersonic gas jets in a high temperature environment to determine optimum manufacturing conditions.

Various experimental and numerical investigations of the behaviour of supersonic oxygen jet after emerging from the nozzle have been reported (Sumi, Kishimoto, Kikichi and Igarashi, 2006, Imai, Kawakami, Miyoshi and Jinbo, 1968, Naito, Ogawa, Inomoto, Kitamura and Yano, 2000, Allemand, Bruchet, Champinot, Melen and Porzucek, 2001, Tago and Higuchi, 2003, Lau, Morris and Fisher, 1979) in the literature. Sumi et al. (2006) studied experimentally the behaviour of supersonic oxygen jet at three different ambient temperatures: 285K, 772K and 1002K. The results showed that velocity attenuation of the jet was restrained and the potential flow core length was extended under high-temperature condition. The potential core length was defined as the distance from the nozzle tip to the point where the magnitude of nozzle exit velocity remains unchanged. Numerical simulations of supersonic oxygen jet behaviour at high ambient temperature carried out by Allemend et al. (2001) and Tago and Higuchi (2003) also showed an increase in potential flow core length but those results were not validated against experimental data.

When a supersonic jet exits from a Laval nozzle, it interacts with the surrounding still air to produce a region of turbulent mixing as shown in Figure 1. This process results in an increase in jet diameter and decrease in jet velocity with increasing distance from nozzle exit. It became known in late seventies that the standard k-ε

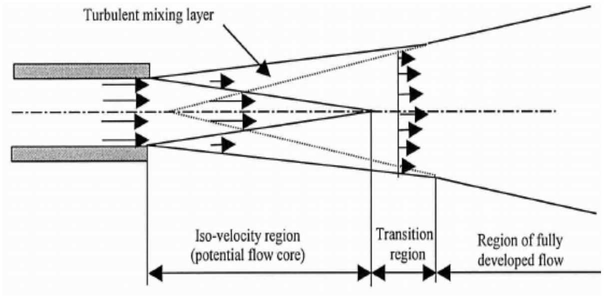

Figure 1: Regions of supersonic jet exiting from a nozzle (Allemand, et al., 2001)

model proposed by Jones and Launder (1972) give a poor prediction of the mean velocity profiles of high speed turbulent axisymmetric jets (Pope, 1978). Because in a high speed jet, the mixing of the jet with its surroundings is suppressed and the growth rate of the turbulent kinetic energy is reduced (Papamoschou and Roshko, 1988). The standard k-ε model lacks the ability to reproduce the observed reduction in growth rate of turbulent mixing region. However, some modifications of the k-ε model have been proposed (Sarkar, Erlebacher, Hussaini and Kreiss, 1991, Heinz, 2003) to take into account the effect of compressibility in reducing the turbulent kinetic energy growth rate.

In the present study, numerical simulation of supersonic oxygen jet, exiting at high ambient temperature, showed that the k-ε turbulence model including the compressibility correction proposed by Heinz (2003), underpredicts potential flow core length at high ambient temperature. This is because at high ambient temperature, the density of the ambient atmosphere is lower. Therefore the mass addition to the jet is lower which further reduces the growth rate of turbulent mixing layer. As a result, the velocity decreases more slowly (Allemand, et al., 2001). Abdol-Hamid et al. (2004) proposed a temperature corrected turbulence model to take into account the large temperature fluctuation when high temperature supersonic jet exits into cold atmosphere. But their model did not give reasonable results when used for simulating cold supersonic jet exiting into a hot atmosphere

In the present study, a simple modification to the k-ε turbulence model was proposed based on the temperature corrected turbulence model developed by Abdol-Hamid et al. (2004). The supersonic jet behaviour at high ambient temperature was simulated by using the modified k-ε turbulence model and the numerical results were validated against experimental data (Sumi, et al., 2006) and a jet model proposed by previous researchers (Ito and Muchi, 1969).

NUMERICAL ANALYSIS

Governing Equations

The numerical simulations were carried out by integrating the unsteady Reynolds-averaged Navier-Stokes (RANS) equations. The averaged mass, momentum and energy equations can be written in a conservative form as follows:

- Mass conservation equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_i}{\partial X_i} = 0 \qquad (1)$$

Where ρ is the density of the fluid, U_i is the mean velocity component in the ith direction.

- Momentum conservation equation

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial X_j} = -\frac{\partial P}{\partial X_i} + \frac{\partial (\tau_{ij} - \rho \overline{u_i u_j})}{\partial X_j} \qquad (2)$$

$$\tau_{ij} = \mu \left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i} - \frac{2}{3} \frac{\partial U_k}{\partial X_k} \delta_{ij} \right) \qquad (3)$$

Where P is the pressure of fluid, τ_{ij} is viscous stress, $u_{i,}u_j$ are the fluctuating velocity component in ith and jth direction respectively and μ is the molecular viscosity.

$-\rho \overline{u_i u_j}$ is known as "Reynold stresses" and is used to represent the effect of turbulence. The Reynold stresses are modelled according to Boussinesq approximation:

$$-\rho \overline{u_i u_j} = \mu_t \left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial U_k}{\partial X_k} \right) \delta_{ij} \qquad (4)$$

Where μ_t is the turbulent viscosity and k is the turbulent kinetic energy. Modelling of turbulent viscosity and turbulent kinetic energy have been described in following sections.

- Energy conservation equation

$$\frac{\partial \rho E}{\partial t} + \frac{\partial (\rho E U_i + P U_i)}{\partial X_i} = -\frac{\partial}{\partial X_i} \left(q_i + C_p \rho \overline{u_i t} \right) + \frac{\partial}{\partial X_i} \left(\tau_{ij} U_j - \rho \overline{u_i u_j} U_j \right) \qquad (5)$$

Where E is the total energy, C_p is the specific heat at constant pressure and t is the fluctuating component of temperature. Heat transferred by conduction, q_i is:

$$q_i = K \frac{\partial T}{\partial X_i} \qquad (6)$$

Where K is the thermal conductivity of fluid. The term $\rho \overline{u_i t}$ is known as turbulent heat flux and is modelled as

$$\rho \overline{u_i t} = \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial X_i} \qquad (7)$$

Where Pr_t is the turbulent Prandtl number. Most common values of Prandtl number is 0.9 and it is satisfactory for shock free flows up to low supersonic speeds and low heat transfer rate (Wilcox, 1998). When a cold supersonic jet (300K) enters into a hot atmosphere of around 1000K, it results in high heat transfer from the hot environment to the jet due to large temperature difference. Wilcox (1998) recommended to use Pr_t=0.5 for free shear flow and high heat transfer problems. Hence Pr_t=0.5 was used in this simulation.

Turbulence modelling

To close the RANS equations, the two equation k-ε turbulence model (Jones and Launder, 1972) was used. In this model the turbulent kinetic energy k and the dissipation rate ε were obtained from the following transport equations:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho U_j k}{\partial X_j} = -\rho \overline{u_j u_i} \frac{\partial U_i}{\partial X_j} + \frac{\partial}{\partial X_j} \left(\mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial X_j} \right) - \rho \varepsilon \qquad (8)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho U_j \varepsilon}{\partial X_j} = -C_{\varepsilon 1} \rho \overline{u_j u_i} \frac{\partial U_i}{\partial X_j} \frac{\varepsilon}{k} + \frac{\partial}{\partial X_j} \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial k}{\partial X_j} \right) - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \tag{9}$$

Where $C_{\varepsilon 1}$,$C_{\varepsilon 2}$, σ_k, σ_ε are the constants for the $k-\varepsilon$ model and their values are 1.44, 1.92, 1.0 and 1.3 respectively.

The turbulent viscosity, μ_t is defined as follows:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{10}$$

In the standard k-ε turbulence model, C_μ=0.09 is used. Heinze (2003 modified the value of C_μ to account for the effect of compressibility which is given by following equation:

$$C_\mu = 0.07 exp\left(-0.4 M_g\right) \tag{11}$$

$$M_g = \frac{|S| l}{a}$$

Where $|S|$ is the mean shear rate, l is turbulence .length scale and a is the speed of sound.

In the present study the value of C_μ was modified to take into account the effect of high ambient temperature. The idea of this modification was taken from the temperature corrected turbulence model developed by Abdol-Hamid et. al. (2004). They modified the constant C_μ according to the following equations:

$$C_\mu = 0.09 C_T \tag{12}$$

$$C_T = \left[1 + \frac{C_1 T_g^m}{1 + C_2 f(M_\tau)}\right] \tag{13}$$

Where T_g is the function of local total temperature gradient normalized by local turbulence length scale:

$$T_g = \frac{|\nabla T_t| \left(k^{3/2} / \varepsilon \right)}{T_t} \tag{14}$$

The reason for using the local total temperature gradient in their model is that total temperature is not Mach number dependent. Hence, flow features that are not related to shear layer mixing, such as flow expansion and internal shocks will not influence the model. The total temperature gradient locates itself automatically in the shear layer where modifications are intended to occur.

In order to model high speed flow, turbulence mach number M_τ was included in their modification. The turbulence mach number and the function of turbulence mach number was defined as:

$$M_\tau = \sqrt{\frac{2k}{a}} \tag{15}$$

$$f(M_\tau) = (M_\tau^2 - M_{\tau 0}^2) H(M_\tau - M_{\tau 0}) \tag{16}$$

Where a is the speed of sound, $H(x)$ is Heaviside function and $M_{\tau 0} = 0.1$. For no compressibility correction, $f(M_\tau) = 0$. The constants and the coefficients of equation (13) are listed in Table 1:

Table 1: Coefficients and constants of equation (13).

Turbulence model	m	C_1	C_2
Abdul-Hamid et al. (2004)	3	24.33	24.33
Present Study	0.6	1.2	1

The functional relationship, constants and coefficients of equation (13) was not derived analytically. Instead, it was determined by trial and error to fit the total temperature experimental data of Seinor et al. (1992) for the supersonic nozzle flow. This modification was proposed for a high temperature supersonic jet flowing into the low ambient temperature. In that case, the potential core length of the jet becomes shorter because of the increment in the growth rate of turbulent mixing layer (Seiner, et al., 1992). The standard k-ε turbulence model fails to predict the observed increase in the growth rate of turbulent mixing. This modification of k-ε model was made to increase the turbulent eddy viscosity in the shear layer by increasing the value of C_μ as a function of total temperature gradient. The higher the values of total temperature gradient, the bigger the values of C_T which in turn increases the values of C_μ. As a result turbulent viscosity increases, resulting in an increase in growth rate of turbulent mixing.

But this modified turbulence model, when used in this study, for cold supersonic jet emitting into a hot environment didn't produce reasonable results although there was a large temperature gradient. As described earlier, when a cold supersonic jet enters into a hot atmosphere, potential core length increases due to the reduction in the growth rate of mixing. The modified model proposed by Abdul-Hamid et al. (2004) can only increase the growth rate of mixing by increasing the value of C_μ. Hence, a different model is required to accurately simulate the cold supersonic jet emitting into high ambient temperature.

In this study, our objective was to develop a model which will reduce the growth rate of turbulent mixing region in order to increase the potential core length of supersonic jet. Unlike the previous model (Abdol-Hamid, et al., 2004) the value of C_μ had to be reduced by dividing the standard value 0.09 by the variable C_T to achieve the desired decrease in turbulent viscosity at the shear layer which in turn reduces the growth rate of mixing. The variable C_T was determined by using the similar functional relationship of equation (13) like the previous model (Abdol-Hamid, et al., 2004) but for the present study the coefficients and the constants were determined by trial and error to match accurately the experimental velocity distribution of Sumi et al. (2006) on the center axis of supersonic jet at different ambient temperatures. The constants and coefficients for the present model are listed in Table 1. Hence, for the present study equation (12) and (13) becomes:

$$C_\mu = \frac{0.09}{C_T} \tag{17}$$

$$C_T = \left[1 + \frac{1.2T_g^{0.6}}{1+f(M_\tau)}\right] \qquad (18)$$

Computational Domain

The computational grid used in the simulations is shown in Figure 2. The computational mesh is an axisymmetric wedge shaped grid with only one cell in circumferential direction. In order to reduce computational time, flow inside the nozzle was not included in the simulation. Flow conditions at the nozzle exit were calculated by using isentropic theory. The exit diameter of the nozzle was 9.2 mm and was considered as the inlet to the computational domain. The size of the computational domain was 100 nozzle diameters downstream from the nozzle exit and 30 nozzle diameters normal to the jet centre-line. The mesh had a total of 7800 cells. The grid density was very high at the exit of the nozzle and at the shear layer.

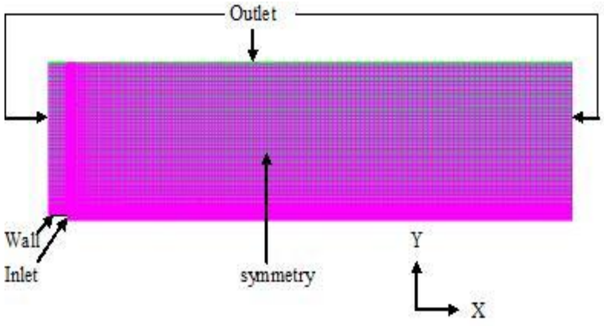

Figure 2: Computational domain and boundary condition used in the simulation.

Boundary condition

All boundary conditions were chosen to match with the experimental study of Sumi et al. (2006). Stagnation pressure boundary condition was used at the inlet of the computational domain (exit of the nozzle). The value of Mach number and total temperature were defined at the inlet. At the outlet static pressure boundary condition was used. For symmetry plane, symmetry boundary condition was used. At the wall zero heat flux was used as wall boundary condition. The values of the boundary conditions are listed in Table-2:

Table 2: Boundary conditions

Inlet	Stagnation pressure	497695 Pa
	Mach number	1.72
	Static Temperature	190 K
Outlet	Static pressure	100000 Pa
	Temperature	285 K 772 K 1002 K

Computational procedure

The unsteady, compressible continuity, momentum and energy equations were solved using segregated solver with implicit approach to calculate the pressure, velocity, temperature and density. For momentum and continuity equations, the values of the variables at cell faces were calculated using AVL SMART scheme (Anonymous, 2006)-this manual cites the original SMART scheme proposed by Gaskell and Lau,1988- which is a second order accurate TVD scheme. For energy and turbulence equations, the first order upwind scheme was used. The pressure-velocity correction was done by using SIMPLE algorithm (Patankar and Spalding, 1972). As the velocity of the flow was very high, time step used in the unsteady calculation was 1×10^{-5} s. The simulations were carried out for sufficient time until no further change was observed in the flow. The simulations were carried out using commercial CFD software AVL FIRE 2008.2

RESULTS AND DISCUSSIONS

Velocity distribution

The computed velocity using standard k-ε model, with compressibility correction of Heinz (2003), along the axis of the jet is plotted with experimental data in Figure 3. The graph shows that the potential core length of the jet increases at high ambient temperature. The agreement between CFD and experimental results is very good when the ambient temperature is 285K. But at higher ambient temperature it fails to accurately predict the velocity distribution. The percentage of deviation increases for higher ambient temperature. The average percentage of deviation of computed velocity from the experimental data is about 13% and 22% for 772K and 1002K ambient temperature respectively. The ambient temperature inside the steelmaking furnace is about 1800K and the deviation will be much larger at such high temperature. This is because the turbulence model used here does not take into account the effect of large temperature fluctuations. Figure 4 shows the velocity distribution on the center axis with modified k-ε model proposed in this study including the effect of high temperature fluctuation. The modified model more accurately predicts the velocity distribution on center axis at high as well as low ambient temperature. When the ambient temperature is 772K, the computed value differs from the experimental value by an average of less than 7%. In case of 1002K ambient temperature the average percentage of deviation of computed velocity from the experimental result is less than 9%.

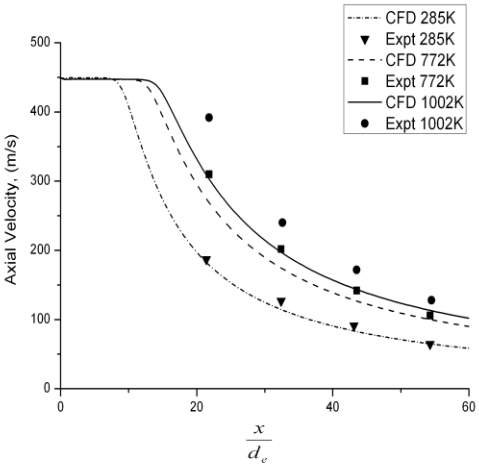

Figure 3: Velocity distribution on center axis with k-ε turbulence including compressibility correction

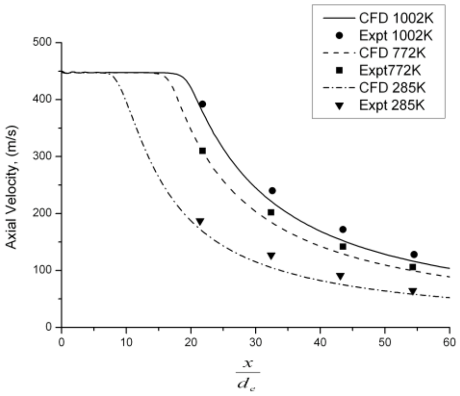

Figure 4: Velocity distribution on center axis with modified k-ε model

Temperature distribution

The calculated temperature distribution on the center axis of the jet is shown in Figure 5. The figure shows that the temperature of the gas jet increases gradually after the discharge from nozzle exit and tends to reach the ambient temperature. The computed jet temperature is in very good agreement with experimental data for 285K and 772K ambient temperature and in reasonable agreement for 1002K ambient temperature. The computed values differ from the experimental results by less than 2% for 285K and 772K ambient temperature. For 1002K ambient temperature the average percentage of deviation is about 7% except at the point $\frac{x}{d_e} = 21$. The reason for this difference is that the numerical model underpredicts the heat transfer through the turbulent shear layer from the ambient to the jet for 1002K ambient temperature. In the present model turbulent transport of heat has been calculated by equation (7) where a constant value of Pr_t =0.5 is used in all cases. But value of Pr_t should be different for different temperature gradient. Because, if the temperature gradient is large, the turbulent heat transfer rate will be high. Decreasing the Pr_t, enhances the heat diffusion capacity in the flow. The manner in which Pr_t varies with different temperature gradient is the subject of further research.

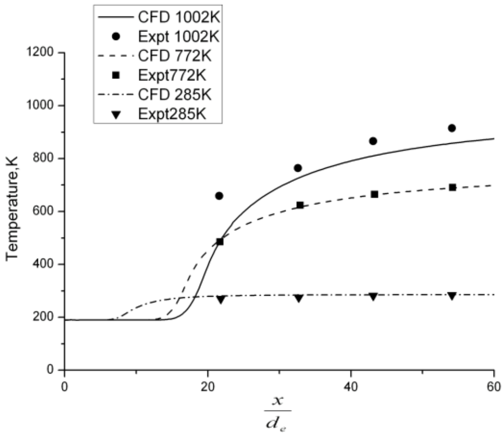

Figure 5: Temperature distribution on center axis

Comparison of jet model with numerical results

The numerical results were compared with the jet model proposed by Ito and Muchi (1969). They derived a general solution for velocity distribution which is expressed by equation (19):

$$-\frac{1}{2ln(1-U_m)} = \alpha\sqrt{\frac{\rho_a}{\rho_e}}\frac{x}{d_e} - \beta \qquad (19)$$

$$U_m = \frac{U}{U_e}$$

Where U_e is the velocity at the nozzle exit, ρ_e is the density at nozzle exit and ρ_a is the ambient density. Sumi et al calculated the value of constants α =0.0841 and β =0.06035 in their experimental study. Figure 6 shows the velocity ratio obtained from present CFD model as a function of $\sqrt{\frac{\rho_a}{\rho_e}}\frac{x}{d_e}$ for different ambient temperature. Velocity ratio obtained from equation (19) and from the experimental study is also shown in the same figure. The CFD model data is found to agree well with jet model data both at high and low ambient temperature with only 12% deviation. This means that the present CFD model can be used to predict the jet behaviour in high ambient temperatures.

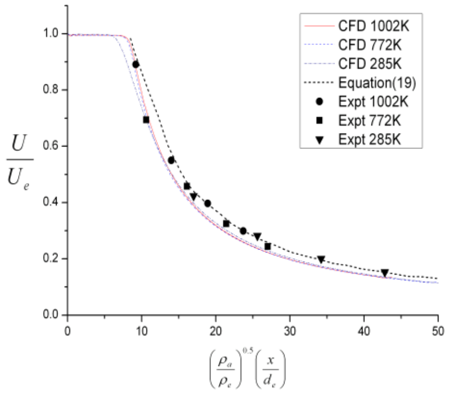

Figure 6: Comparison of velocity ratio with equation (19) as a function of $\sqrt{\frac{\rho_a}{\rho_e}}\frac{x}{d_e}$.

CONCLUSION

The behaviour of supersonic oxygen jet in a high temperature field was investigated by CFD simulation. The simulation results were compared with the experimental study done by Sumi et al. (2006). The modified turbulence model proposed in this study was found to be in very good agreement with experimental data with high ambient temperatures. But, if the total temperature of supersonic gas jet at the inlet of Laval nozzle is equal to the ambient temperature, the modified model will revert back to the standard k-ε model because the gradient of total temperature will be zero. In that case, it will under predict the coherent length as there will be no compressibility correction. Hence, further work is needed to combine this model with compressibility correction model. Also this model is not suitable for the simulation of a high temperature supersonic jet flowing into low ambient temperature. The present study will give useful information for developing theoretical turbulence model including the effect of temperature fluctuation.

REFERENCES

DEO, B. and BOOM, R., (1993), *Fundamentals of Steelmaking Metallurgy,* 21-28

SUBAGYO, BROOKS, G. A., COLEY, K. S. and IRONS, G. A., (2003), "Generation of Droplets in Slag-Metal Emulsions through Top Gas Blowing", *ISIJ International*, **43**, 983-989

MCGEE, P. and IRONS, G. A., (2002), "The penetration of oxygen lance jets in foaming slags", *Iron and Steelmaker (I and SM)*, **29**, 59-68

PEASLEE, K. D. and ROBERTSON, D. G. C., (1994), "Fluid dynamics of inclined jetting on a slag/metal bath", *EPD Congress Proceedings*, 1129-1145

LUOMALA, J. M., FABRITIUS, T. L. J., VIRTANEN, E. O., SIIVOLA, T. P. and HARKKI, J. J., (2002), "Splashing and spitting behaviour in the combined blown steelmaking", *ISIJ International*, **42**, 944-949

SUMI, I., KISHIMOTO, Y., KIKICHI, Y. and IGARASHI, H., (2006), "Effect of high temperature field on supersonic oxygen jet behaviour", *ISIJ International*, **46**, 1312-1317

IMAI, R., KAWAKAMI, K., MIYOSHI, S. and JINBO, S.-I., (1968), "Effects of Blowing Conditions on Blowing Reactions in LD converter", *Nippon Kokan Technical Report-Overseas*, **8**, 9-18

NAITO, K.-I., OGAWA, Y., INOMOTO, T., KITAMURA, S.-Y. and YANO, M., (2000), "Characteristics of Jets from Top-Blown Lance Converter", *ISIJ International*, **40**, 23-30

ALLEMAND, B., BRUCHET, P., CHAMPINOT, C., MELEN, S. and PORZUCEK, F., (2001), "Theoretical and experimental study of supersonic oxygen jets. Industrial application in EAF", *La Revue de Metallurgie*, **98**, 571-587

TAGO, Y. and HIGUCHI, Y., (2003), "Fluid Flow Analysis of Jets from Nozzles in Top Blown Process", *ISIJ International*, **43**, 209-215

LAU, J. C., MORRIS, P. J. and FISHER, M. J., (1979), "Measurements in subsonic and supersonic free jets using laser velocimeter", *J. Fluid Mech.*, **93**, 1-27

JONES, W. P. and LAUNDER, B. E., (1972), "The prediction of Lamniraization with a two-equation model of turbulence", *Int. Journal Heat Mass Transfer*, **15**, 301-314

POPE, S. B., (1978), "An explanation of the turbulent round jet/plane jet anomaly", *AIAA Journal*, **16**, 279-281

PAPAMOSCHOU, D. and ROSHKO, A., (1988), "Compressible turbulent shear layer: An experimental study", *journal of Fluid Mechanics*, **197**, 453-477

SARKAR, S., ERLEBACHER, G., HUSSAINI, M. Y. and KREISS, H. O., (1991), "The analysis and modelling of the dialational terms in compressible turbulence", *J. Fluid Mech.*, **227**, 473-495

HEINZ, S., (2003), "A model for the reduction of the turbulent energy redistribution by compressibility", *Physics of Fluids*, **15**, 3580-3583

ABDOL-HAMID, K. S., PAO, S. P., MASSEY, S. J. and ELMILIGUI, A., (2004), "Temperature corrected turbulence model for high temperature jet flow", *ASME Journal of Fluids Engineering*, **126**, 844-850

ITO, S. and MUCHI, I., (1969), "Transport phenomenon of supersonic jet in oxygen top blowing converter", *Iron & Steel Inst Japan-J (Tetsu-To-Hagane),* **55**, 1152-1163

SEINER, J. M., PONTON, M. K., JANSEN, B. J. and LAGEN, N. T., (1992), "The effects of temperature on supersonic jet noise emission", *DGLR/AIAA 14th Aeroacoustics Conference*, *Aachen*, *Germany*, **AIAA paper 92-02-046**,

ANONYMOUS, (2006), "The numerical solution procedure", *Fire CFD solver v8.5 manual*,

PATANKAR, S. V. and SPALDING, D. B., (1972), "A calculation procedure for heat, mass and momentum transfer in three dimensional parabolic flows", *Int. Journal Heat Mass Transfer*, **15**, 1787-1806