Heat and mass transfer

HEFAT2010
7th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
19-21 July 2010
Antalya, Turkey

3-D NUMERICAL SIMULATION AND ANALYSIS OF HEAT TRANSFER, FLUID FLOW AND SOLIDIFICATION INSIDE THE MOULD DURING CONTINUOUS CASTING OF STEEL

Ajam H.*, Sadat M. and Hosseini Sarvari S.M.
*Author for correspondence
Department of Mechanical Engineering,
University of Sistan and Baluchestan,
Zahedan,
Iran,
E-mail: hajam@hamoon.usb.ac.ir

ABSTRACT

A Three Dimensional simulation of mould in continuous casting process with consideration of standard k-ε model has been presented and corroborated. The main difference between this work and previous ones is that the phase change process (solidification) and turbulent flow distribution have been coupled and solved jointly instead of dividing it into “transient heat conduction process” and “steady fluid flow” that can lead to more realistic simulation. The main objective in this work is to have better understanding of the fluid flow pattern, heat transfer and solidification in the continuous casting mould. Investigating the influence of superheating degree, nozzle port angle, casting speed, comparing the temperature distribution and analyzing other parameters, can lead us to a better design in the way of optimistic working condition.

INTRODUCTION

Over the last two decades there has been a tremendous need in the manufacturing of metals such as steel/aluminium/cooper [1]. Increasing the productivity and improving the product quality are permanent necessities. Quick quenching from the liquid state, such as rapid solidification of metals and alloys which is called “Continuous Casting” (CC), has attracted great attention due to the subsequent advantages: (a) The near-net shape formation of metallic materials which will be accompanied by reductions of energy, time and labour (b) The development of new functional properties caused by structural modification, such as the formation of non-equilibrium phases (c) An improvement of mechanical properties caused by decreased segregation and the refinement of grain size [2,3,4]. Consequently, there will be no hesitation that continuous casting technology is currently the primary method of producing billets, blooms and slabs and understanding of the its analysis is playing a major role to reach the best production situations. Fortunately “Mathematical Modelling” is an effective, inexpensive tool to get information that cannot be directly measured in the steel.

NOMENCLATURE

C		Constant parameters
C_p	[J/kgK]	Constant pressure specific heat
F	[N/m^3]	Body force
g	[m/s^2]	Gravity acceleration
h	[J/kg]	Sensible enthalpy
H	[J/kg]	Total enthalpy
k	[W/mK]	Thermal conductivity and turbulent kinetic energy
L	[J/kg]	Latent heat
p	[N/m^2]	Pressure
T	[K]	Temperature
u	[m/s]	Velocity component
x	[m]	Cartesian axis direction
Special characters		
β		Liquid volume fraction
ΔH	[J/kg]	Latent enthalpy
ε	[J/kg]	Turbulent dissipation rate
μ	[kg/m.s]	Dynamic viscosity
ρ	[kg/m^3]	Density
Subscripts		
i		Indicate Cartesian axis direction
j		Indicate Cartesian axis direction
l		Laminar flow
liquidus		Indicate to become liquid
t		Turbulent flow
μ		Heat generating medium
p		Constant pressure
ref		Reference
solidus		Indicate to become solid

A typical CC process involves phase change, coupled with fluid flow and heat transfer. The majority of presented numerical solution to phase change problem are defining it as a transient, heat conduction problem. This method however is limited to the problems with small temperature interval over which the phase change is expected [5].

In this work flow and solidification have been coupled and solved together steadily. Although, because of complexity of phenomenon involving in the mould such as turbulent fluid flow and changing phase simultaneously, it is very difficult to reach an adequately exact answer but it is obvious that the answer will be more reliable.

MOULD PARAMETERS, GRID STRUCTURE AND FLUID PROPERTIES

The main operation in the mould is that, the steel is superheated enough in the furnace and stored in tundish in order to have the capability of distribution in the mould (Figure 1).

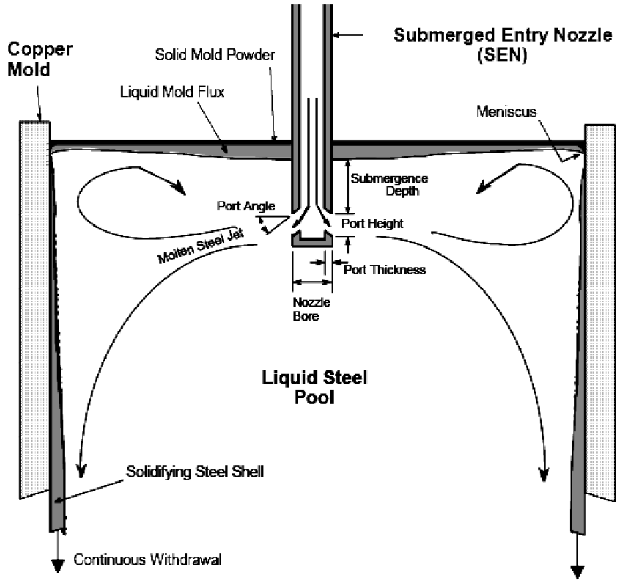

Figure 1 Mould structure

Then metal enters from the tundish through nozzles into the top of mould which includes four separate copper walls, as shown in Figure 1. The depth of the mould can vary up to 2.5 meter, depending on the casting speed and strand size. Thermal and mechanical behaviour of the mould depends on its construction and constraints including geometry, casting speed, boundary condition, etc. The mould is basically an open-ended box structure, containing a water-cooled inner lining fabricated from a high purity copper alloy. The main roll of the mould is to construct a solid shell, sufficient in strength to sustain its liquid core upon entry into the secondary cooling zone. The inner face of the copper mould is often plated with chromium or nickel to provide a harder working surface and to prevent melting due to high temperature of casting metal [6].

As shown in figure 1 there is no difference between four quarters of mould domain so the simulation can be carried out for only one quarter of it [6,7,8].

Unstructured grid has been used in order to reduce the computation as much as possible. However, solving these governing equations takes lots of time by a usual pc. The grid is unstructured vertically and horizontally and it has been compressed wherever it is essential, particularly where the flow impinges the narrow wall and returns.

Table 1 shows the liquid and solid properties of steel. The parameters of the submerged entry nozzle (SEN), the caster and some of operation condition parameters, are given in Table 2.

Table 1 Steel properties

Properties	Values	Properties	Values
Density (liquid)	7000 (kg/m^3)	Specific heat (solid)	686 (J/kg.K)
Density (solid)	7400 (kg/m^3)	Viscosity	0.005 (kg.m/s)
Conductivity (liquid)	420 (w/m.K)	Melting heat	271000 (J/kg)
Conductivity (solid)	35 (w/m.K)	Solidus temperature	1703 (K)
Specific heat (liquid)	686 (J/kg.K)	Liquidus temperature	1743 (K)

Table 2 Mould parameters and operation conditions

Parameters	Values	Parameters	Values
Mold length	1.3 (m)	Port angle	0, 15, 30 (degree)
Mold width	0.25 (m)	Port height to width ratio	0.08, 0.065 (m/m)
Mold height	2 (m)	Inlet velocity	0.242, 0.323, 0.404 (m/s)
SEN diameter	0.08 (m)	Casting speed	0.015, 0.02, 0.025 (m/s)
SEN height	0.3 (m)	Casting temperature	1800 (K)

MATHEMATICAL MODEL

As it was mentioned, CC is a phase change process. A technique called “Enthalpy-Porosity” is used in FLUENT for modelling the process. In this technique, a quantity called “Liquid Fraction”, which indicates the fraction of the cell volume that is in liquid form, is associated with each cell in the domain. The liquid fraction is computed at any iteration with consideration of enthalpy balance. The mushy zone is a region in which the liquid fraction lies between 0 and 1 and during the phase change the porosity decreases from 1 to 0 as the metal solidifies. When the metal has completely solidified in a cell, the porosity becomes zero and hence the velocities also drop to zero [9,10,11].

The code is solving the governing equations including continuity and Navier-Stokes for the entire domain for the steady fluid flow of incompressible and Newtonian fluids which are:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + (\mu_l + \mu_t)\frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) + \rho g_i + F_j \tag{2}$$

Where, i, j are coordinate direction indices, which when repeated in a term, implies the summation of all three possible terms, ρ is liquid density (kg/m^3), u_i is velocity component in x_i direction (m/s), p is pressure field (N/m^2), μ_l is laminar viscosity (kg/m.s), μ_t is turbulence viscosity (kg/m.s), g_j is the magnitude of acceleration gravity in j direction (m/s^2) and F_j can be other body forces.

But we know that the flow is highly turbulent so we have to consider a turbulent model. With the *k-ε* Model, the turbulent viscosity is given by:

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

The two supplementary partial differential equations for the transport of turbulent kinetic energy, k and its dissipation rate ε are given by:

$$\rho u_i \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + \mu_t \frac{\partial v_j}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \rho\varepsilon \qquad (4)$$

$$\rho v_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) + C_1 \mu_t \frac{\varepsilon}{k}\frac{\partial u_j}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - C_2 \frac{\varepsilon^2}{k}\rho \qquad (5)$$

In order to achieve reasonable accuracy on a coarse grid, the *k-ε* model needs special "wall functions" as boundary conditions. Several works in the past [7] have been done in order to evaluate the accuracy of different turbulence models in the complicated task of predicting flow behaviour close to a wall where there is jet impingement. In some usual codes that have been used in CC simulation by now like CFX, The non-dimensional cell size at the wall, y^+ is kept at about 30.

$$y^+ = \frac{\rho C_\mu^{1/4} K_p^{1/2} y_p}{\mu} \qquad (6)$$

It has been shown that although the predicted flow patterns for the standard *k-ε* model match well with experimental measurements, the calculation of heat transfer has some difficulty. The standard *k-ε* model greatly under–predicts heat flux so user defined wall functions with lower y^+ have to be employed to match experimental heat transfer results [7]. Fortunately in FLUENT, the log-law is employed when $y^+ < 11.225$ and the results, both in flow pattern and temperature field have good agreement with investigational results.

As it was mentioned, CC is a phase change process. A technique called "Enthalpy-Porosity" is used in FLUENT for modeling the process. In this technique, a quantity called "Liquid Fraction", which indicates the fraction of the cell volume that is in liquid form, is associated with each cell in the domain. The liquid fraction is computed every iteration with consideration of enthalpy balance. The "mushy zone" is a region in which the liquid fraction lies between 0 and 1 and during the phase change the porosity decreases from 1 to 0 as the metal solidifies. When the metal has completely solidified in a cell, the porosity becomes zero and hence the velocities also drop to solid velocity.

The enthalpy of the material, H is computed as the sum of the sensible enthalpy, h and the latent heat, ΔH and can be shown as:

$$H = h + \Delta H \qquad (7)$$

The liquid fraction, β is defined as follow,

$$\beta = \begin{cases} 0 & \text{if} \quad T < T_{solidus} \\ \dfrac{T - T_{solidus}}{T_{liquidus} - T_{solidus}} & \text{if} \quad T_{solidus} < T < T_{liquids} \\ 1 & \text{if} \quad T > T_{liquidus} \end{cases} \qquad (8)$$

The latent heat content now is written in terms of the latent heat of the material, L as $\Delta H = \beta L$

And after all the energy equation is written as

$$\frac{\partial}{\partial t}(\rho H) + \nabla \cdot \left(\rho \vec{\upsilon} H\right) = \nabla \cdot (k \nabla T) + s \qquad (9)$$

The solution for temperature is essentially iteration between the energy equation and the liquid fraction equation [9,10,11].

RESULTS

Every simulation is done in order to show the effect of involving parameters in the way of optimistic working condition:

Best Angle:

Determining the optimum port angel for the nozzle can play a major roll in designing mould. Fundamentally, we know that the conductivity of steel is very small in comparison with other metals like cooper, consequently the nozzles ports are used in order to create a flow pattern within the mould that makes the heat transfer steadier along the walls. Three special port angles have been simulated. All parameters in these cases are the same with the exception of the port angles of nozzles.

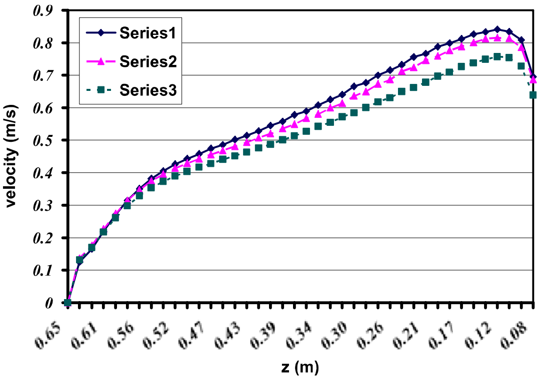

Figure 2 Velocity along the jet center line: (1) port angle=0, (2) port angle=15, (3) port angle=30

In figures 2 and 3 we see that increasing the angle, reduces jet center line velocity because of the reduction in volume of the upper vortex and hence it reduces the material concentration. On the other hand, increasing the port angle not only makes the flow pattern steadier but also it reduces heat flux concentration at the impingement point of jet.

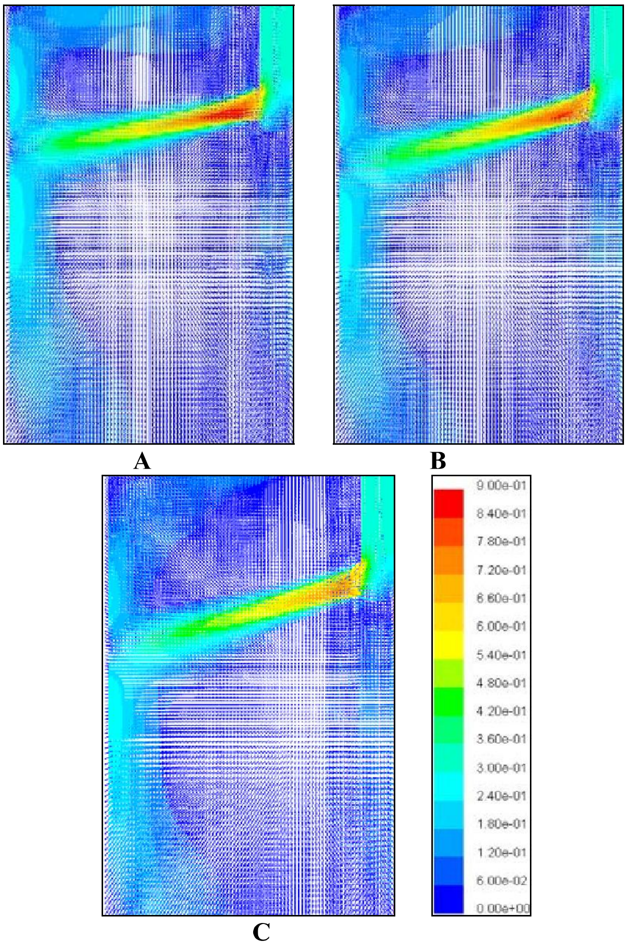

Figure 3 Vectors of Velocity for different port angles: (A) port angle=0, (B) port angle=15, (C) port angle=30

Figure 4 and figure 5 show the distribution of heat flux across the upper half of mould narrow wall. We can see that as the angle increases, not only the maximum value of heat flux reduces but also the distribution of heat flux on the narrow wall is steadier. It seems that increasing the port angle causes good effects on the involving parameters by affecting the flow pattern. But there is a constraint in designing nozzle port geometries that they must not direct the flow too deep. Naturally the coldest regions are found at the top corners near the narrow face and near the SEN [6]. As we know the solidified shell is pulled out of mould with casting speed. The steel surface can solidify a solid bridge between the SEN and the shell against the mould wall, which often causes a breakout.

This is an apprehension, because this could lead to quality problems such as cracks and other surface defects. To avoid these problems, flow must reach the surface quickly.

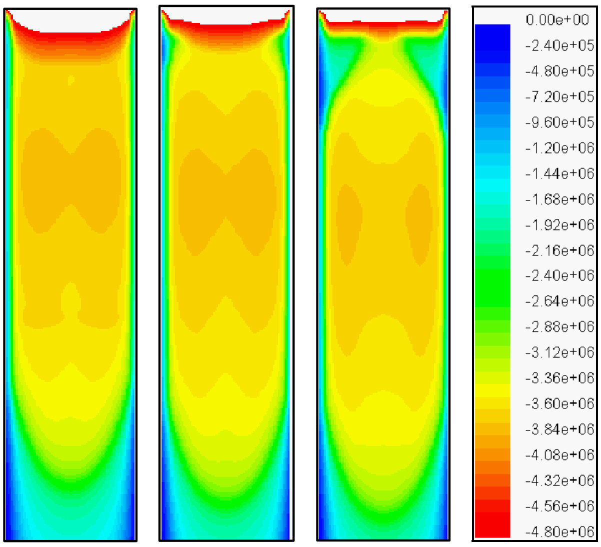

Figure 1 Heat Flux distribution and impingement point along the narrow wall: (A) port angle=0, (B) port angle=15, (C) port angle=30

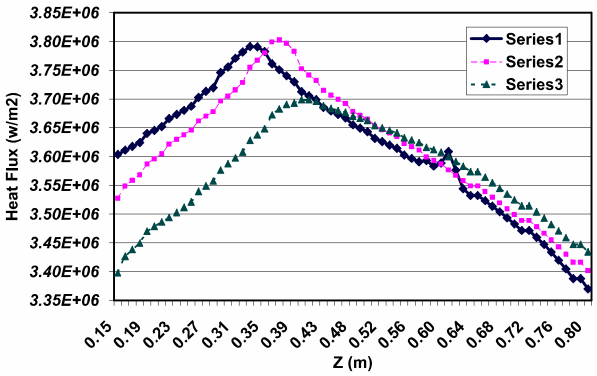

Figure 5 Heat Flux along the narrow wall center line: (1) port angle=0, (2) port angle=15, (3) port angle=30

Figure 6 shows the contours of temperature for different angles. As we see by increasing the port angle from A to B and C, the cold region in top left corner increases and hotter section in right corner progressively decreases.

Figure 7 shows the temperature allocation in top corner near the narrow wall for different angles. As we see by increasing the angle from zero to 15 and 30, not only there are 5 degrees centigrade distinctions between coldest parts but also entire temperature field decreases.

Casting Speed

As we know, in order to have more casting speed, the mass flow rate through the nozzle into the mould must be increased and there will be less time for the mould walls to form a shell from the liquid, sufficient in strength. Although this phenomenon is very important from the view of metallurgical considerations (and is not discussed in this paper), it causes a great increase in values of heat flux on both narrow and wide walls of mould that can be very important because each casting system has restricted capabilities.

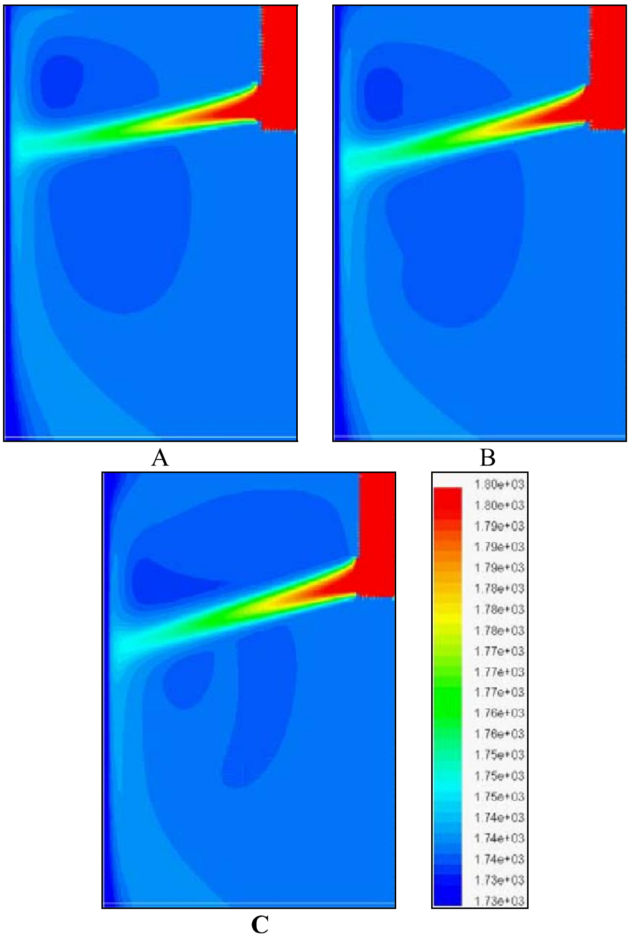

Figure 6 Contours of temperature for different angles: (A) port angle=0, (B) port angle=15, (C) port angle=30

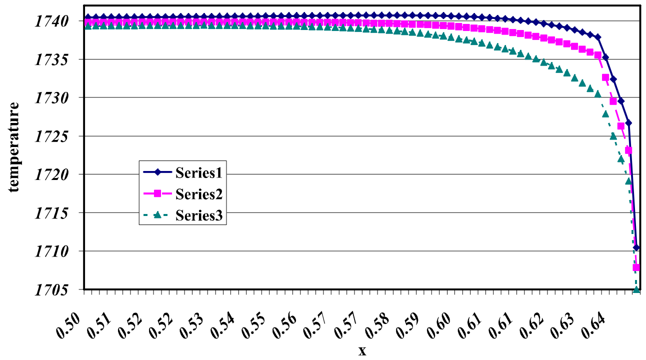

Figure 7 Distribution in top corner near the narrow wall for different angles: (1) port angle=0, (2) port angle=15, (3) port angle=30

In order to show the importance of casting speed on heat flux, the walls of mould have been kept at the same constant temperature for different casting speed. Figure 8 and 9 shows the heat flux on narrow and wide walls of mould for different casting speed. Also, it affects shell growth along the both narrow and wide walls of the mould which has been shown in figures 10 and 11. So it is very important to consider the metallurgical and economical views in choosing the best casting speed for CC system.

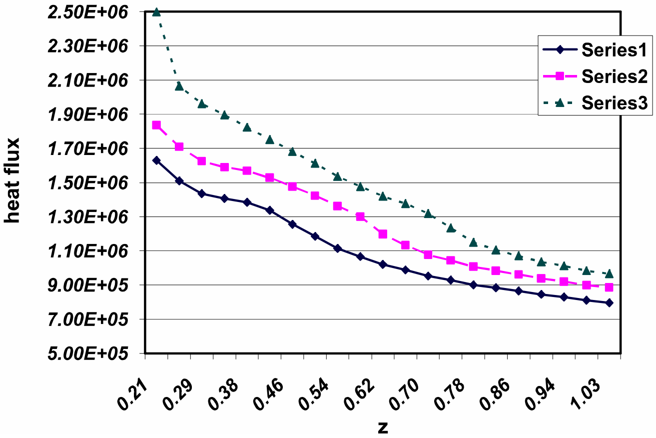

Figure 8 Heat Flux on wide walls of mould for different Casting Speed: (1) casting speed=0.02 m/s, (2) casting speed=0.025 m/s, (3) casting speed=0.03 m/s

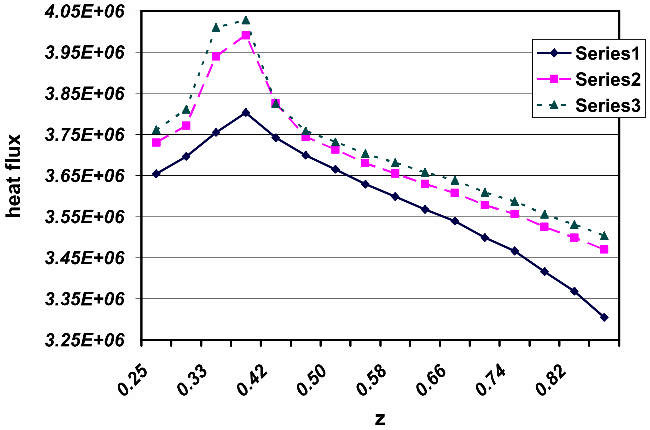

Figure 9 Heat Flux on narrow walls of mould for different Casting Speed: (1) casting speed=0.02 m/s, (2) casting speed=0.025 m/s, (3) casting speed=0.03 m/s

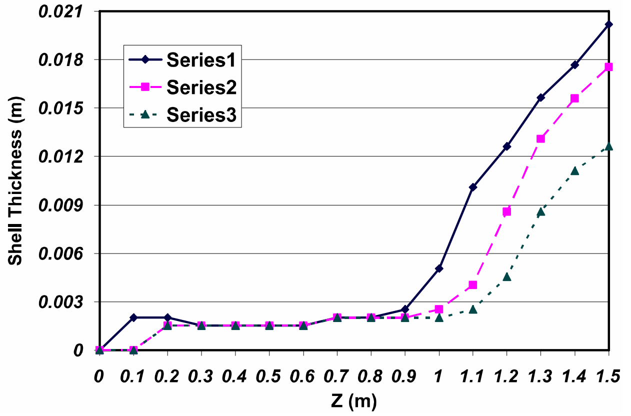

Figure 10 Effect of Casting Speed on shell thickness for narrow wall: (1) casting speed=0.02, (2) casting speed=0.025, (3) casting speed=0.03

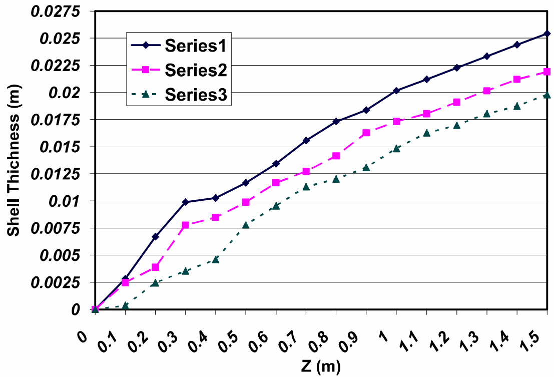

Figure 11 Effect of Casting Speed on shell thickness for wide wall: (1) casting speed=0.02, (2) casting speed=0.025, (3) casting speed=0.03

Superheating Degree

Superheating the liquid flow inlet is one of the most important designing parameters. Superheat is sensible heat which is given to the metal above its liquidus temperature to have the capability of distribution in the mould [6]. In other words, if the inlet molten steel is not superheated enough it may suddenly be frozen as soon as it reaches the Solidus temperature and stops the process. Figure 12 shows the contours of temperature for different superheating degree.

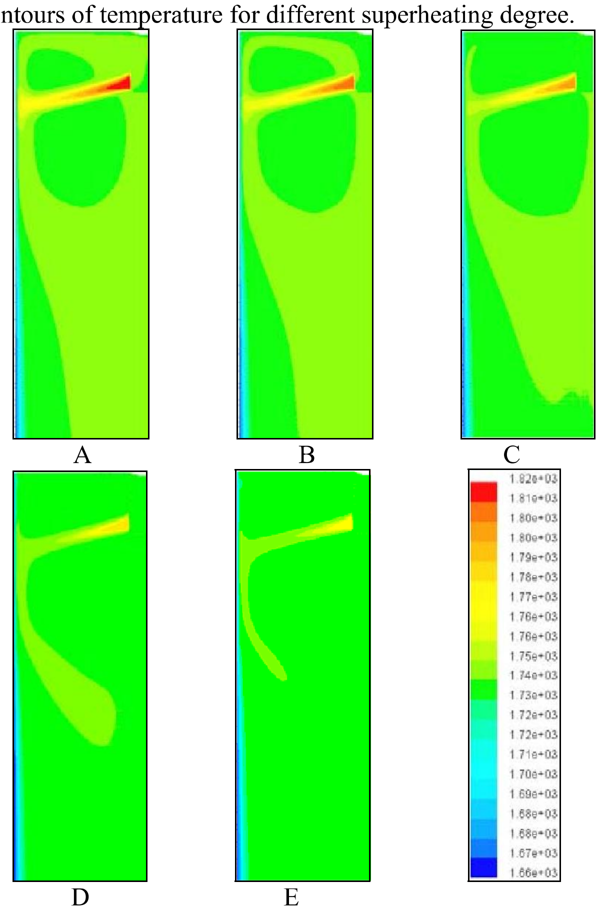

Figure 12 Temperature Distribution for different superheating degree $(^{\circ}C)$: (A) 1820, (B) 1810, (C) 1800, (D) 1780, (E) 1770

As we can see, in both upper and lower circulations, there is a great rise in temperature so the fluid will have more capability of distribution. As we see by decreasing the temperature from A to E, the Solidus Bridge that was mentioned in port angle design may cause breakout.

CONCLUSION

A three dimensional turbulent flow and solidification in a thin slab caster has been simulated. The model has coupled and solved the k-ε flow and heat transfer simultaneously and from this view point has an advantage. The accuracy of this modelling approach has been demonstrated by comparison with a simulation [8], although there are little differences because of some unidentified parameters.

Then some significant parameters have been investigated and the different working condition has been shown for them. Consequences of port angle, casting speed and superheating degree on necessary heat flux, shell thickness, and breakout probability, have been analyzed.

REFERENCES

[1] Ruhul Amina M., Mahajan Anurag, Modeling of turbulent heat transfer during the solidification process of continuous castings, *Journal of Materials Processing Technology* 174 (2006) pp 155–166

[2] Hagiwara M. and Inoue A., Production techniques of alloy wires by rapid solidification techniques, *in: H.H. Liebermann (Ed.), Rapidly Solidified Alloys conf.*, Marcel Dekker, New York, 1993, pp. 139–156.

[3] Boettinger W.J. and Perepezko J.H., Fundamentals of solidification at high rates, *in: H.H. Liebermann (Ed.), Rapidly Solidified Alloys conf.*, Marcel Dekker, New York, 1993, pp. 17–78.

[4] Herlach D.M. and Willnecker R., Under cooling and solidification, *in: H.H. Liebermann (Ed.), Rapidly Solidified Alloys conf.*, Marcel Dekker, New York, 1993, pp. 79–102.

[5] Bonacina C., Comini G., Fasano A., Primicerio M., Numerical solution of phase-change problems, *Int. J. Heat Mass Transfer 16 conf.*, (1973) pp. 1825-1832

[6] Thomas B.G., Continuous Casting: Complex Models, *The Encyclopedia of Materials: Science and Technology*, K.H. J. Buschow, R. Cahn, M. Flemings, B. Ilschner, E. J. Kramer, S. Mahajan, eds., (J. Dantzig, subject ed.) Elsevier Science Ltd., Oxford, UK, Vol. 2, 2001, pp. 1599-1609

[7] Creech D.T. and Thomas B.G., 3-D Turbulent Multiphase Modeling of Molten Steel Flow and Heat Transfer in a Continuous Slab Caster, *CFX User's conference*, Wilmington, DE, Oct. 1, 1998.

[8] Thomas B.G. and Malley R.O., Validation of Fluid Flow and Solidification Simulation of a Continuous Thin-Slab Caster, *MCWASP IX Conf.*, Shaker Verlag, GmbH, Aachen, Germany, 2000, pp. 769-776

[9] Voller V.R., Modeling of Solidification Processes, *Technical report, Mathematical Modelling of Metals Processing Operations Conf.*, American Metallurgical Society, Palm Desert, CA, 1987.

[10] Voller V.R., Brent A.D., Reid K.J., A Computational Modeling Framework or the Analysis of Metallurgical Solidification Process and Phenomena, *Technical report, Conference for Solidification Processing*, Ranmoor House, Sheffield, September 1987.

[11] Voller V.R. and Prakash C., A Fixed-Grid Numerical Modeling Methodology for Convection-Diffusion Mushy Region Phase-Change Problems, *Int. J. Heat Mass Transfer* 30, 1987. pp 1709-1720