Numerical Analysis Of Interfacial Behavior With Turbulence Effect In Water Model Of A Continuous Casting Mold

[1]***Rajat Kumar Das***
Mechanical Engineering Department, I.T.E.R
Siksha 'O' Anusandhan University, SOA
Bhubaneswar,Orissa,India
rajatdas_1980@rediffmail.com

[2]***Chetana Tripathy***
Mechanical Engineering Department,
Synergy Institute of Technology,SIT
Bijupatanaik University of Technology,BPUT
Bhubaneswar,Orissa, India

[3]***SukantaKumarDash***
Mechanical Engineering Department,
Indian Institute of Technology, Kharagpur ,I.I.T.
West Bengal,India
sdash@mech.iitkgp.ernet.in

Abstract— **Over the years numerous efforts have been given by different researchers to analyze the performance of Submerged Entry Nozzle (SEN) and mold in continuous slab caster. Submerged Entry Nozzle plays a crucial role for improving the quality of steel product. The present paper numerically investigates the fluid flow pattern inside the casting mold with interfacial behavior between water and air in which different shell thickness and Submergence height to Port ratio(H/P) were considered. It was found that as Submergence height to Port ratio (H/P) goes on increasing, the interface fluctuates with a lesser amplitude. As the shell thickness grows there is more fluctuation at the free surface. Two-dimensional numerical analysis using finite volume method and modified HRIC (High Resolution Interface Capturing) scheme is used to investigate the interfacial behavior within the casting mold. In the present work instead of molten steel, water has been used as the working fluid since both fluids have almost the same kinematic viscosity.**

Keywords— Submerged Entry Nozzle (SEN), Free surface fluctuation, Turbulent kinetic Energy, Turbulent Intensity, Fluid Flow, Numerical Simulation

Abbreviations and Acronyms

C =	Volume fraction of fluid
ρ =	Density
k =	Turbulent kinetic energy
U=	Mean velocity
$u_i u_j$ =	Average turbulent stress
υ =	Kinematic viscosity
k =	Turbulent kinetic energy
ε =	Rate of dissipation of turbulent kinetic energy
σ =	Surface tension coefficient

In recent times as steel demand is growing continuously so it is high time to increase productivity as well as the quality product. Cleanliness of steel slab produced in continuous casting machine is the priorities of steel industry. Researchers from academia and industry notice that fluid flow pattern inside the continuous casting mold significantly affects quality of steel product. The fluid flow pattern inside the casting mold depends on several factors such as valve that connects tundish with submerged entry nozzle(SEN),internal geometry of SEN, submergence depth of SEN, mold dimension and casting speed. Higher casting speed is necessary for continuous casting which creates several other problems inside the mold. Higher casting speed increases interfacial fluctuation and hence level of turbulence within the mold increases which controls the quality of steel. Lots of efforts have been given experimentally as well as numerically by numerous researchers to analyse fluid flow behavior, surface disturbance, surface velocity and level of turbulence inside the casting mold. Dash et al.(2004) have studied the behavior of free surface and made a conclusion that free surface fluctuation is unsteady in nature. Excluding the initial transience phase, it was found out that the wave amplitude of the free surface varies within a certain limit. Gupta and Lahiri(1994) studied interfacial behavior by taking different parameters and concluded that vortex formation and bubble entrainment at free surface depends upon nozzle design, flow rate of water and submergence height. Chaudhary et al.(2009) studied similarity criteria between water model and steel caster. Das and Dash (2012) numerically investigated free surface behavior by taking different parameters such as water velocity, air velocity and the size of the recirculation roll. ZHANG Qiao-ying, WANG Xin-hua (2010)analyzed fluid flow pattern inside the casting mold and concluded that higher casting speed brings more surface disturbance, so lower casting speed should be used. Wu and Cheng (2008) numerically investigated and concluded that different parameters like nozzle port angle, port width, height and thickness of solidifying shell in the mold affect the fluid flow behavior and free surface fluctuation and based on which improved nozzle geometry could be arrived at. Das and Dash (2013) investigated the role of Port to Bore ratio [P/B], water velocity and size of upper recirculation roll on free surface wave and based on these parameters the nozzle was designed. Chen et al.(2012) developed a new type of SEN named self breaking SEN to reduce wave amplitude of free surface and again fluid flow inside the mold was more symmetrical and tracer dispersion was more uniform in case of new nozzle. Zheng and Zhu (2010) investigated the effect of gas flow rate, casting speed and submergence height of SEN on free surface. They developed a new approach where mean wave height and maximum wave height technique are used for estimating the free surface fluctuation. Hai-qi et al. (2010) studied fluid flow behavior inside the casting mold as well as interfacial fluctuation between molten steel and liquid slag layer by using different parameters like submergence depth, port angle, casting speed and argon gas flow rate. Xia et al.(2001) studied the flow characteristic inside the caster. Morales et al.(2012) have done both experimental and numerical work to analyze fluid flow behavior inside the casting mold by taking different types of nozzles. They made a conclusion that four ports SEN is suitable for higher casting speed. The effect of water velocity and dimensionless parameter like Froude number on free surface oscillation as well as turbulence effect inside the mold were reported in Das et al.(2014).Singh et al.(2006) conducted both experimental and numerical work about air bubble movement with different flow rate of water and air. They concluded that bubble penetration depth mainly depends on water flow rate rather than air flow rate. Jeon et al. (2010) conducted experimental work to investigate flow behavior and interfacial fluctuation inside the casting mold. They found vortex generation mechanism inside the mold by taking different parameters. Chaudhary et al. (2008) conducted experimental as well as numerical work under steady and transient state condition to analyze fluid flow pattern inside the casting mold by taking well-bottom and mountain-bottom nozzle. Rivera-Perez et al.(2014) have done both experimental and numerical work to evaluate the performance of a bifurcated submerged entry nozzle which incorporates fluid flow conditions attached to the SEN external wall. The result confirmed that the proposed new design reduces interfacial fluctuation and the appearance of vortices at the free surface was minimized. Because of the advantages with numerical simulation, the present work is based on investigation of root mean square (r.m.s) amplitude of surface wave and level of turbulence with time inside the casting mold.Figure1shows a continuous casting process.

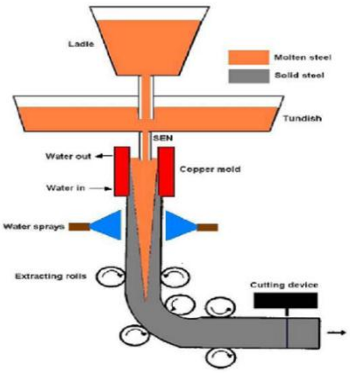

Fig.1. *Continuous Casting Process*

I. GOVERNING EQUATIONS

As the fluid flow is incompressible and viscous in nature with turbulence, so the following governing equations are used:

Continuity Equation:

$$x = \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho U_i) = 0 \quad (1)$$

Momentum Equation:

$$\frac{D(\rho U_i)}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[\mu\left\{\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right\} - \overline{\rho u_i u_j}] + \rho g + F_\sigma \quad (2)$$

Fluid flow inside the mold is turbulent in nature. So selection of a turbulence model is crucial to obtain results of fluid flow. Hence κ-є model is used for capturing fluid dynamics inside the mold which are given by the following equations:

Turbulent kinetic energy, κ :

$$\frac{D(\rho k)}{Dt} = D_k + \rho p - \rho \epsilon \quad (3)$$

Rate of dissipation of κ :

$$\frac{D(\rho \varepsilon)}{Dt} = D_\varepsilon + C_1 \rho p \frac{\varepsilon}{k} - C_2 \frac{\rho \varepsilon^2}{k} \quad (4)$$

Where

$$\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} - \vartheta_t \{\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\}$$

$$\vartheta_t = \frac{C_\mu k^2}{\varepsilon}$$

$$D_\emptyset = \frac{\partial}{\partial x_j}[\left\{\mu + \frac{\mu_t}{\sigma_\emptyset}\right\}\frac{\partial \emptyset}{\partial x_j}]$$

$$p = -\overline{u_i u_j}\frac{\partial U_i}{\partial x_j}, \ \emptyset = k \text{ or } \varepsilon$$

Constants used in the κ-є model are C_1 = 1.44, C_2 =1.92, σ_C =1.0, σ_κ=1.0, σ_ϵ=1.3, and C_μ=0.09

The present numerical simulation is a two-dimensional unsteady problem. Volume of fluid (VOF) method is a surface tracking technique applied to a fixed Eulerian mesh. It is designed for two immiscible fluids where interfacial position is of prime interest. A single set of momentum equation is shared by the fluids and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. HRIC interface capturing scheme is used for simulating free surface. The density of water is constant and the volume fraction for the second phase (air) has the following equation.

$$\frac{\partial c}{\partial t} + U.\nabla c = 0 \quad (5)$$

Where c and U are the volume fraction of fluid and the mean-velocity of fluid respectively. The grid extends to both liquid and gas phase. Entire control volume is filled by liquid if c=1 and otherwise if c=0 then entire control volume is filled by gas. Both liquid and gas are treated as single effective fluid whose properties change in space according to the volume fraction of each phase i.e.

$$\mu = \mu_1 c + \mu_2 (1-c)$$
$$\rho = \rho_1 c + \rho_2 \ (1-c) \quad (6)$$

where, subscripts 1 and 2 denote liquid and gas respectively. The effect of surface tension force per unit volume at free surface is given by the equation

$$k = -\nabla.\left(\frac{\nabla c}{|\nabla c|}\right) \quad F_{\sigma=}\ \sigma\frac{\rho k \nabla c}{0.5(\rho_1+\rho_2)} \quad (7)$$

Where σ, K, ρ are surface tension co-efficient, curvature of free-surface and average volume density respectively.

II. Boundary Conditions and Solution Methodology

A uniform water velocity is given at SEN inlet. At wall boundaries no slip condition is given. The upper part of mold is open to the atmosphere. So pressure outlet boundary condition is applied there. At the bottom of the mold, an inlet velocity boundary condition is given with a negative value. This is the drawing speed of the caster. Equations (1) to (4) are integrated on the control volume and then discretized using a finite volume technique which results in algebraic equations. These algebraic equations are solved by the multi grid solver of Fluent 6.3. The pressure velocity coupling is done by the PISO algorithm while the convective terms of the momentum and turbulence equations are discretized by the second order upwind scheme. The time step was taken to be 0.004s, for each time step iteration. For convergence of the equations the whole field residuals for mass, momentum and turbulent quantities were set to 1e-3 while for the volume fraction it was set to 1e-5. A VOF method does not need the specification of bubble size since it would inherently find the air entrapment if any in the mold and thus would create an air pocket or an air bubble.

At the interface of the bubble there would be surface tension force present and that would be acting as a body force in the momentum equation as has been shown in Eqn.(2).

III. Initial Conditions

All velocity components are set to zero at time t=0, except for the value at the inlet where normal velocity into the computational domain was prescribed to have a certain value depending on the through put we desire. In the computational domain all turbulent quantities were initialized to zero except at the inlet having a predefined value of 2% turbulent intensity and turbulent viscosity ratio of 10. The present work is a two phase problem. Water is taken as primary phase and air as the secondary phase. The properties of the two fluids are shown in table 1

TABLE I. PROPERTIES OF DIFFERENT FLUIDS

Property	Water	Air
ρ	1000(kg/m^3)	1.225(kg/m^3)
σ	0.073 N/m	
μ	0.001 pa-s	1.8x10^{-5} pa-s

V.Result and Analysis

Validation with Experimental Results

Air entrapment in liquids is an ubiquitous process in many natural and engineering applications. It is because of bulk mass transfer that occurs at the free surface. The phenomena of wave breaking and entrapment of air bubbles have been experimentally visualized by Gupta and Lahiri(1994)for a parallel port nozzle having circular opening with a port exit velocity of 1.94 m/s as it is shown in figure 2(a).A vertical scale is placed on the side wall (away from nozzle side) which helps to show the wave amplitude. A horizontal scale is placed at top of the figure 2(b) to measure the width of the mold. Experimental snapshot was taken just at the time of entrapment of air bubbles. Fig.2(b) shows numerical results of interfacial fluctuation. The time interval of numerical results for free surface was taken to be 0.04 sec with starting time of t=3.2 sec. Both experimental and numerical results closely match with each other as we see from the figure. A conclusion is drawn from both experimental and numerical work that the surface wave breaks near the trough region and then the entrapment of air bubbles occur just after the breaking of the wave. The fluid elements on free surface are always balanced by viscous, surface tension and gravity forces. Since free surface gradually becomes flatter near trough region, so the resulting surface tension force decreases due to increasing radius of curvature. So the surface tries to overturn and by turning more the surface comes to a new equilibrium position and thus enhancing surface tension force again. At the same time the gravity force pulls the liquid element down and as a result the free surface undergoes breaking and entrapment of air bubbles occur near the trough region as it is shown in figure 2(c). In real caster because of breaking of surface wave, slag enters into the mold and mixes with molten steel which controls quality of final product.The experimental investigation about free surface has been numerically modeled with grid arrangements as it is shown in figure 2(d).The free surface is located 1072 mm from the bottom of the mold with a submergence height of 150 mm. The cells around the free surface region are refined in both x and y direction in order to get better result. The present computational approach has given right trend in interpreting the behavior of the free surface and entrapment of air bubbles through it.

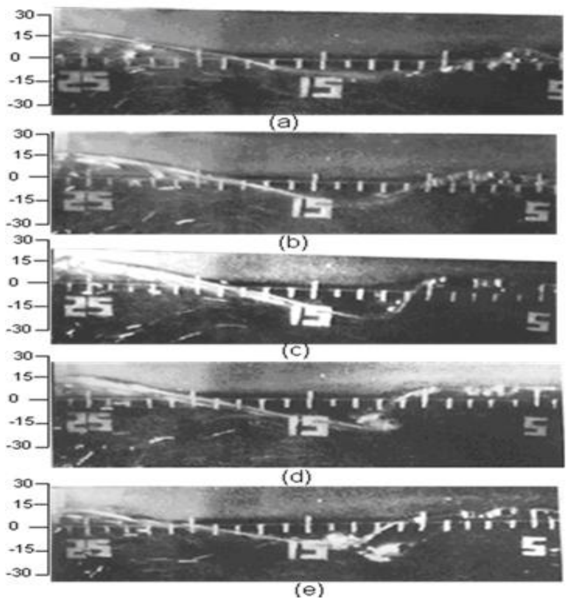

Fig.2(a).*Experimental Snapshots of free Surface fluctuation developed with Parallel Port SEN at an time interval of 0.04 sec, Port exit velocity 1.94 m/s Source: (Gupta and Lahiri 1994).All dimensions are in mm*

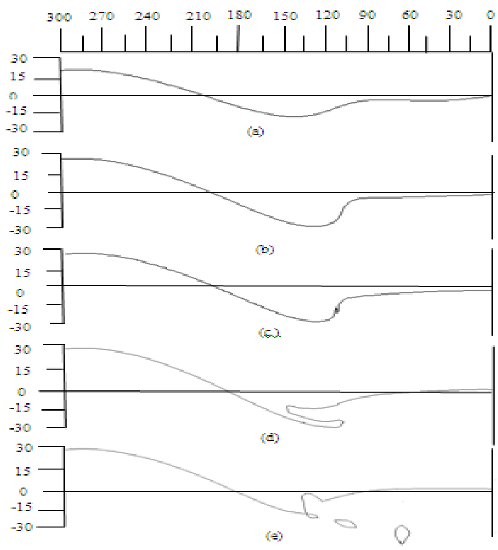

Fig.2(b).*Numerical results validated with Experimental results*

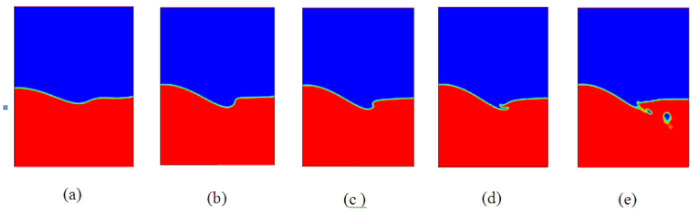

Fig.2(c). *Breaking of free surface with entrapment of air bubbles*

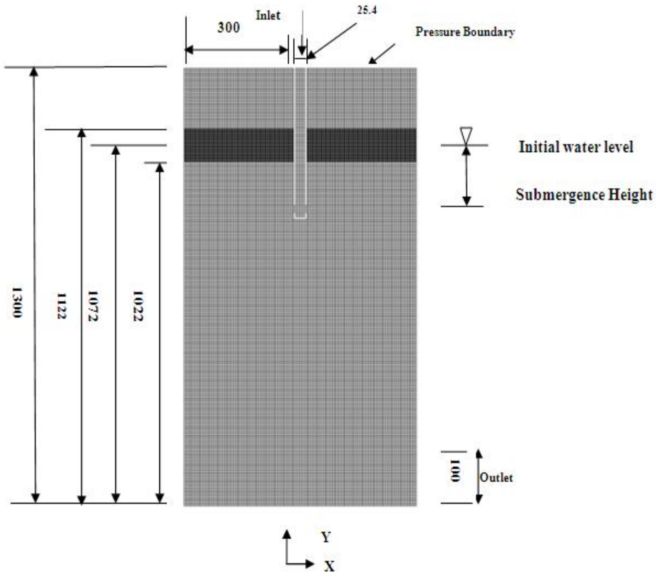

Fig.2(d). *Computational Domain with grid arrangements for simulation of Experiment. All dimensions are in mm*

VI.GRID INDEPENDENT FREE SURFACE

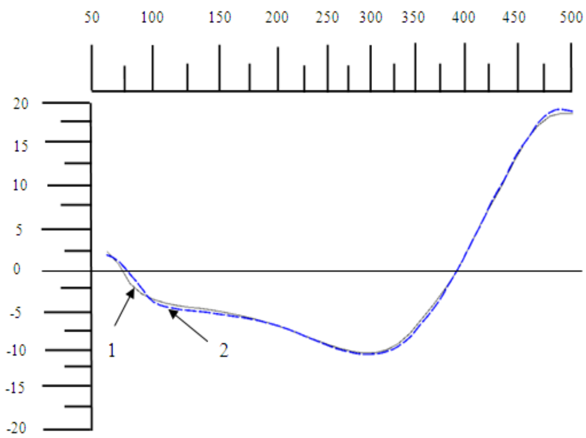

Fig. 3. *Grid Independent Tests*
All dimensions are in mm

In the present computation a cell size of 2.5x2.5 mm have been used near the free surface to capture it very sharply. Figure 4 shows grid independent test where the distance from the nozzle to side wall and interfacial fluctuation are indicated by the X-axis &Y-axis respectively. Two kinds of cell sizes, a coarser one with a cell size of 5x5 mm and a finer one with 2.5x2.5 mm have been used to compare the free surface fluctuation. It can be seen that there is not much difference between the two free surfaces.

VII. VARIATION OF INTERFACIAL FLUCTUATION ,SURFACE VELOCITY,TURBULENT KINETIC ENERGY AND TURBULENT INTENSITY WITH TIME

Water jet which passes through the nozzle strikes the side wall of the mold, resulting in recirculation roll which creates distrubance at the free surface.Figure 4 shows interfacial displacement at crest and trough for a long time.Root mean square (rms) value of fluctuation level is used to illustrate the fluctuation intensity of each monitored time.It is concluded that with increase in time unsteadiness dies out to a great extent resulting variation of crest and trough height within a certain limit.Similarly figure 5 shows variation of free surface fluctuation and surface velocity with time. Barring the initial large variation of both free surface height and surface velocity, the subsequent variations of both these qualtities become very gradual and fall witihin a limiting value.Similar variations can be seen for the turbulent kinetic energy and turbulent intensity with time as shown in figure 6.

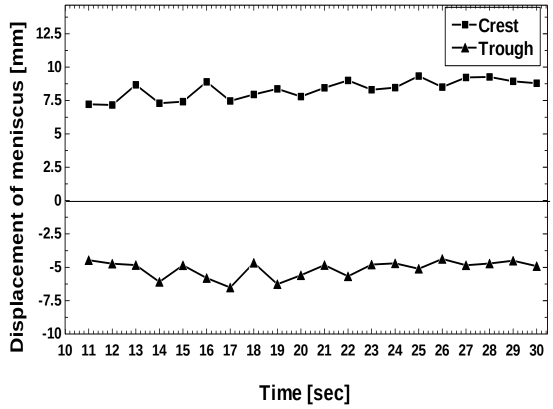

Fig.4. *Variation of Free Surface at crest and trough with time*

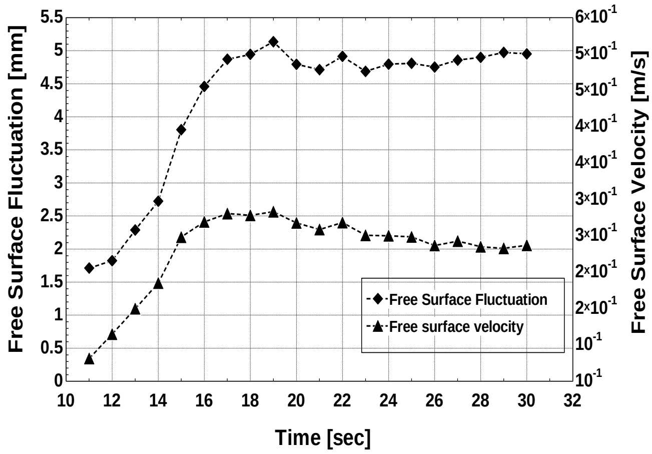

Fig.5.*Variation of Free Surface and Surface velocity with time*

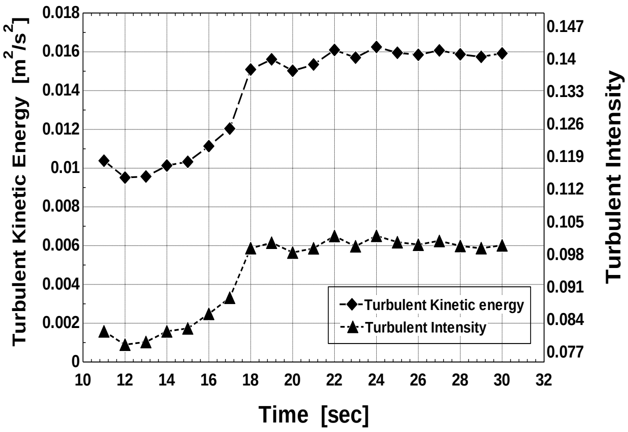

Fig.6. *Variation of Turbulent Kinetic Energy and Turbulent Intensity with time*

VIII. EFFECT OF WATER VELOCITY ON TURBULENT KINETIC ENERGY AND TURBULENT INTENSITY

With continuous increase of inlet water velocity there is chaotic behavior of fluid flow inside the mold which creates strong meniscus instability and forcing the fluid to flow along the side wall.As a result the level of turbulence as well as turbulent intensity increases which are associated with interfacial instability as it is shown in figure 7 and 8.

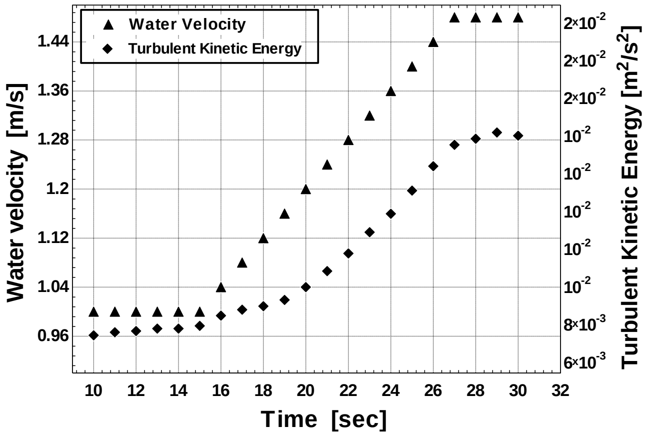

Fig.7.*Variation of Turbulent Kinetic Energy with time*

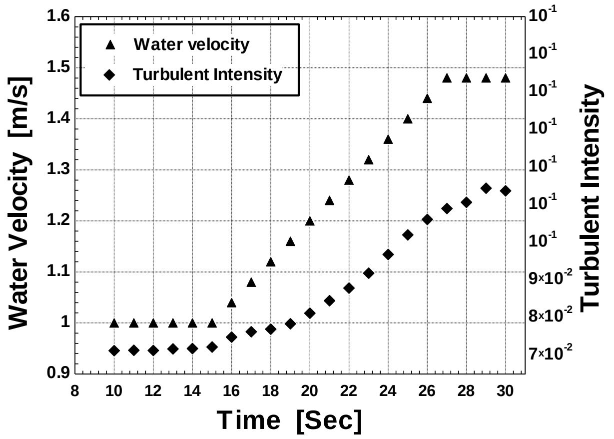

Fig.8. *Variation of Turbulent Intensity with time*

IX. EFFECT OF SHELL THICKNESS AND SUBMERGENCE HEIGHT TO PORT RATIO(H/P) ON FREE SURFACE FLUCTUATION

Attention has been given for studying fluid flow inside the mold with consideration of a solidified shell. The fluid flow of molten steel has an important effect on growth of solidified shell of slab. By considering the presence of a shell in the casting mold there is obvious shrinkage at narrow and wide faces in casting direction and it affects the characteristics of fluid flow.
Instead of taking real caster the present numerical work is based on water model. With the increase of shell thickness, the water jet passing through the nozzle strikes the side wall with a higher impact. So fluid reaches free surface with a higher momentum resulting in more fluctuation. It seems that the

TABLE2. EFFECT OF SHELL THICKNESS ON ANGULAR VELOCITY

Shell Thickness(mm)	Angular Velocity (ω) (rad/s)
10	$4.214x10^{-3}$
20	$5.455x10^{-3}$
30	$6.383x10^{-3}$
40	$8.223x10^{-3}$

increase of (H/P) ratio effectively decreases the free surface fluctuation. The fluid comes out of the nozzle as a jet and then impinges on the side wall of the mold creating an upper recirculation as well as a lower recirculation zone. Development of upper recirculation flows are responsible for free surface oscillation as it is shown in figure 9. It has been earlier discussed that the growth of the shell thickness creates an erratic nature of free surface. The growth of shell thickness increases the angular velocity of the upper recirculation roll. So the fluid reaches the free surface with a higher momentum which increases chaotic nature of free surface as it is shown in table 2.Similarly the angular velocity of upper recirculation roll decreases with increasing submergence height to Port ratio (H/P) which causes to reduce the free surface oscillation as it can be seen in table 3

TABLE3. EFFECT OF SUBMERGENCE HEIGHT TO PORT RATIO(H/P) ON ANGULAR VELOCITY

Submergence height to Port ratio(H/P)	Angular Velocity (ω) (rad/s)
2.7	$4.043x10^{-3}$
2.8	$3.445x10^{-3}$
2.9	$3.043x10^{-3}$
3	$2.88x10^{-3}$

X. CONCLUSION

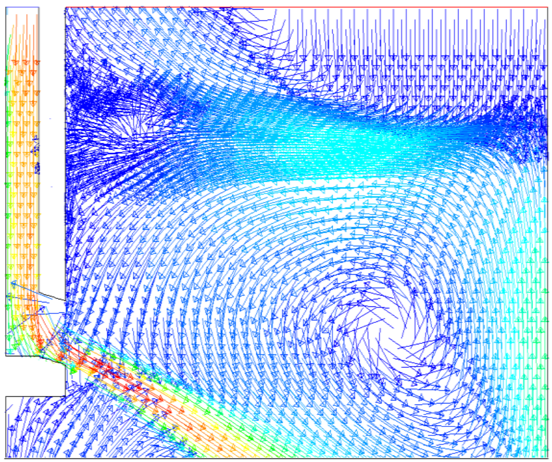

Fig.9. *Effect of Upper Recirculation Roll*

Numeri ... dy
the effe ... ht

to Port ratio (H/P) on the free surface fluctuation, surface velocity and level of turbulence inside the mold. Both experimental and numerical results closely match with each other as far as the free surface wave is concerned. It was observed that excluding the initial transience the free surface fluctuates within a certain limit for all the time. It was found from numerical analysis that by increasing submergence height to Port ratio the interfacial fluctuation decreases resulting in decrease of free surface velocity. Again as the shell thickness goes on increasing the free surface fluctuates with a greater amplitude with increasing level of turbulence inside the mold. The present two dimensional simulation is good enough for predicting the fluid flow behavior on meniscus stability while it contributes to the understanding of turbulence in the continuous casting mold.

References

[1] Dash S.K., Mondal S.S., Ajmani S.K. "Mathematical simulation of surface wave created in a mold due to submerged entry nozzle" International Journal of Numerical Methods for Heat and Fluid flow vol.14, No 5, pp. 606-632, 2004

[2] Gupta D. Lahiri A.K., "Water modeling study of the surface disturbances in continuous slab caster", Metallurgical and Material Transactions vol.25 B,pp. 227-233,1994

[3] R. Chaudhury, B.T. Rietow,B.G. Thomas, "Difference between physical Water Models and Steel Continuous Casters A Theoretical Evaluation",Materials Science and Technology (MS&T) pp. 1090-1101, 2009

[4] Das R.K.,Dash S.K.,"Free Surface Fluctuation for design of Submerged Entry Nozzle",International Journal on Theoretical and Applied Research in Mechanical Engineering Vol.-3, No1, pp.109-114, 2012

[5] Qiao-ying,Xinhua W, "Numerical Simulation of Influenceof casting speed variation on Surface fluctuation of Molten steel in Mold", Journal of Iron and Steel Research International,vol 17,No 8,pp.15-19,2010

[6] Wu D.F. and Cheng S.S., " Effect of SEN design on Surface fluctuation and solidifying shell in slab mold and its optimization",Acta,Metall. Sinica,Vol.21, No5,pp.341-350,2008

[7] Das R.K., Dash S.K., "Numerical Analysis of free surface in water model for design of Submerged Entry Nozzle'',International Journal of Advanced Computer Research, Vol. 3,No 1 pp.182-187, 2013

[8] Chen Yongfeng, Zhang Lifeng, Yang Shufeng, Li Jingshe, "Water Modeling of Self-Braking Submerged Entry Nozzle Used for Steel Continuous Casting Mold'',JOM, vol.64 No 9pp 1080-1086, 2012

[9] Zheng S, Zhu M , "Physical modeling of gas-liquid interfacial fluctuation in a thick slab continuous casting mold with argon blowing'', International Journal of Minerals, Metallurgy and Materials,Vol.17 No.6 pp.704-708, 2010

[10] Hai-qiYu, Miaoyong ,ZHU, Jun Wang, "Interfacial Fluctuation Behavior of Steel/Slag in Medium-Thin slab Continuous Casting Mold with Argon Gas Injection'',Journal of iron and Steel, Research International,Vol. 17, No 4, pp 5-11,2010

[11] Morales R.D., Tang Y, Nitzl G, Eglsaeer C and Hackl G, "Design of Submerged Entry Nozzle for Thin Slab Molds Operating at High Casting Speeds'', ISIJ International, Vol. 52,No.9,pp 1607-1615, 2012

[12] Das R.K., Tripathy Chetana, Dash S.K., "Numerical Analysis of the turbulence effects in casting mold due to Submerged Entry Nozzle''International Journal of Research in Aeronautical and Mechanical Engineering,Vol.2,Issue.1, pp 61-71, 2014

[13] Singh V, Dash S.K., Sunitha S, Ajmani S.K., Das A.K., "Experimental simulation and mathematical modeling of air bubble movement in slab caster mold''ISIJ Vol.46, No.2pp 201-218 , 2006

[14] Jeon Young Jin, Sung Hyung Jin, Lee Sangmin, "Flow Oscillations and meniscus fluctuations in a Funnel-Type Water Mold Model'',Metallurgical and Materials Transactions B,Vol.41B pp121-130, 2010

[15] Chaudhary R, Lee Go-Gi, Thomas B.G.and Kim Seon-Hyo, "Transient Mold Fluid Flow with Well- and Mountain-Bottom Nozzles in Continuous Casting of Steel'',Metallurgical and Materials Transactions B, pp 784-790, 2008