Numerical Simulation of Fluid Flow in Centrifugal Casting

Ankush Budihal[1], Bharath C V[2], Eldo Elias[3]
[1, 2, 3] Dept of Mechanical Engineering.
[1, 2, 3] Nitte Meenakshi Institute of Technology, Bangalore, India.

Abstract- *The centrifugal casting process is a common method for manufacturing the tubes, etc. Due to its high temperature and invisible mould, it is really difficult to know the mechanism of molten steel inside the mould. It is important to know the mechanism of the molten steel inside mould, since it will help the manufacturer to know more accuracy of the flow of the molten steel so that it can work for improving the productivity and quality of the products.*

The mechanism of solidification in centrifugal casting is also important since it can give manufacturer the general view of solidification process. So the basic mechanism of fluid flow (Cold flow) is simulated in this thesis.

Keywords- Numerical Simulation, Fluid, flow, Centrifugal casting, cold flow, pressure contours, VOF modling, Multiphase.

I. INTRODUCTION

Centrifugal casting has been known to the modern civilization since the year 1809. Over the years, centrifugal casting has been used for wide range of applications. In this system, the hot fluid (either a pure liquid metal or alloyed with other metallic particles), poured inside a partially filled the mould rotating at a high speed, forces the fluid towards the wall of the relatively colder mould due to centrifugal force. This strong positive radial pressure gradient imposed on the liquid metal increases the static pressure towards the mould wall and pushes up the fluid free surface in the form of a paraboloid shape. CFD analyses based on approximate volume of fluid (VOF) concept for treating the moving liquid-gas interface (free-surface) during rotation as well as the cold modelling experiments reported in literature have attempted to understand the simplified flow process only in absence of heat transfer using the VOF concept for both horizontal and vertical moulds. The actual centrifugal casting process is an extremely rapid process where it is hard to visualize the flow and hence even the modern experimental methods are very difficult and expensive. On the other hand, Mathematical analysis can be conveniently used to get an overall idea of this process. As mentioned earlier, the fluid flow in centrifugal casting is extremely complex and depends on various factors like viscosity, mould roughness, metal temperature, rate of cooling, phase change etc. However the present work carried out is only limited to observing the behavior of particle at a constant pouring temperature without any heat transfer or phase change.

II. LITERATURE REVIEW

Kestur Sadashivaiah:

Has studied the effect of various variables on flow patterns. He carried out both numerical simulation and cold model experiment based on horizontal centrifugal casting. He also investigates the flow patterns based on different viscosities of liquid steel.

R. Zagórski:

Presented a model and simulations of the centrifugal casting of a metal matrix composite reinforced with SiC in the initial stage of casting

Vinay Chandran:

Has studied the horizontal centrifugal casting process of manufacture aluminium-silicon carbide FGM by ANSYS fluent. He also compared his experiment result with the computational simulation result to verify the simulation result.

J.W. GAO:

Has carried out a numerical simulation of solidification in centrifugal casting process of functionally graded materials. He found that the angular velocity, solidification rate and the geometrical nature of the particle flow are responsible for creation of the particle concentration gradient in solidified products.

G. Chirita:

Presented that difference of production technology of a structure component with Al-Si alloys. The two production technologies involved were centrifugal casting and traditional gravity casting. The centrifugal casting technology gave a better mechanical property than the traditional gravity casting.

S C Mondal:

Reported the process capability of the centrifugal casting process. He used the computational capability index value to analyze the results.

J Boháček:

Has studied the solidification process of work rolls which are made by the centrifugal casting process. A 2D numerical simulation was carried out to investigate the average flow dynamics. A water model was also set up for comparison and the results compared favorably with the numerical model for the transition to a developed flow regime. Based on the previous investigations above, the flow pattern in the vertical centrifugal casting process is limited, so this project will give a mechanism on centrifugal casting process and it can also provide the valuable data on that.

III. OBJECTIVES

- Modeling of the complex flow system and numerical solution of the governing conservation equations of mass, momenta and energy.
- To validate the results by comparing with the measurement data.
- Flow visualisation data for Fluid Flow in a partially filled Vertical cylinder rotating about its vertical axis to be checked against simple radial equilibrium equation
- Numerical computation will be attempted for predicting the free surface contour for a given fluid rotated at a given RPM for which some experimental visualization has already been carried out in the laboratory
- CFD Analysis of a 3D flow in a container partially filled and rotating about its vertical axis considering the conservation of mass and momentum components along Cartesian directions and also one more scalar transport equation for the void fraction (=1 for Liquid Metal and = 0 for Air)
- Three momentum components, energy and void fraction with appropriate boundary conditions including the heat loss from the free surface of the liquid metal.
- Computations has been carried out using the ANSYS Fluent software on the high speed multiprocessor computation facilities at M/s ENTUPLE TECHNOLOGIES LTD, BANGALORE.

IV. METHODOLOGY

Analytical Method:

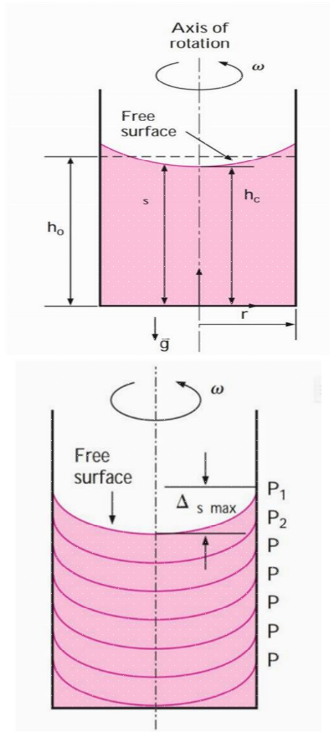

FIG. 1 Pressure based analysis

- This problem is best analyzed in cylindrical coordinates (r,Θ,z)
- The cylinder is symmetry about z axis.
- The total differential of P=(r,z) is given by

$$dp = \rho r\omega^2 dr - \rho g dz$$

- The equation for surface of constant pressure is obtained by setting dp=0
- $Z(\text{isobaric}) = (\omega^2/2g)r^2 + c$
- Thus we conclude surfaces of constant P are paraboloids of revolution
- The overall pressure variation across radius 'r' is given by

$$P = P_0 + \frac{\rho\omega^2}{2}r^2 - \rho g z$$

Numerical Method

The Problem Computed Using ANSYS (FLUENT) Code
The code solves:

- Momentum Equations along three Cartesian direction (u,v and w)
- Continuity Equation
- Scalar Equation for Void fraction (V =0 for air and V=1 for water)

The code uses Pressure Based Finite Volume Algorithm (**S**emi **IM**plicit **P**ressure **L**inked **E**quation algorithm)

Navier-Stokes Equations

Conservation Law

Navier-Stokes equations are the governing equations of Computational Fluid Dynamics. It is based on the conservation law of physical properties of fluid. The principle of conservational law is the change of properties, for example mass, energy, and momentum, in an object is decided by the input and output.

For example, the change of mass in the object is as follows

$$\frac{dM}{dt} = \dot{m}_{in} - \dot{m}_{out}$$

If

$\dot{m}_{in} - \dot{m}_{out} = 0$, we have

$$\frac{dM}{dt} = 0$$

Which means

$M = const$

Navier-Stokes Equation

Applying the mass, momentum and energy conservation, we can derive the continuity equation, momentum equation and energy equation as follows.

Continuity Equation

$$\frac{D\rho}{Dt} + \rho \frac{\partial U_i}{\partial x_i} = 0$$

Momentum Equation

$$\underbrace{\rho \frac{\partial U_j}{\partial t}}_{I} + \underbrace{\rho U_i \frac{\partial U_j}{\partial x_i}}_{II} = -\underbrace{\frac{\partial P}{\partial x_j}}_{III} - \underbrace{\frac{\partial \tau_{ij}}{\partial x_i}}_{IV} + \underbrace{\rho g_j}_{V}$$

Where

$$\tau_{ij} = -\mu\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) + \frac{2}{3}\delta_{ij}\mu\frac{\partial U_k}{\partial x_k}$$

I: Local change with time
II: Momentum convection
III: Surface force
IV: Molecular-dependent momentum exchange (diffusion)
V: Mass force

Energy Equation

$$\underbrace{\rho c_\mu \frac{\partial T}{\partial t}}_{I} + \underbrace{\rho c_\mu U_i \frac{\partial T}{\partial x_i}}_{II} = -\underbrace{P\frac{\partial U_i}{\partial x_i}}_{III} + \underbrace{\lambda \frac{\partial^2 T}{\partial x_i^2}}_{IV} - \underbrace{\tau_{ij}\frac{\partial U_j}{\partial x_i}}_{V}$$

I: Local energy change with time
II: Convective term
III: Pressure work
IV: Heat flux (diffusion)
V: Irreversible transfer of mechanical energy into heat

Boundary Conditions:

To solve the equation system, we also need boundary conditions. The typical boundary conditions in CFD are No-slip boundary condition, axisymmetric boundary condition, Inlet, outlet boundary condition and Periodic boundary condition.

For example, fig is a pipe, the fluid flows from left to right. We can use inlet at left side, which means we can set the velocity manually. At the right side, we use outlet boundary condition to keep all the properties constant, which means all the gradients are zero.

At the wall of pipe, we can set the velocity to zero. This is no-slip boundary condition.

At the center of pipe, we can use axisymmetric boundary condition.

The present problem is analyzed rotation of wall for 10000 rpm.

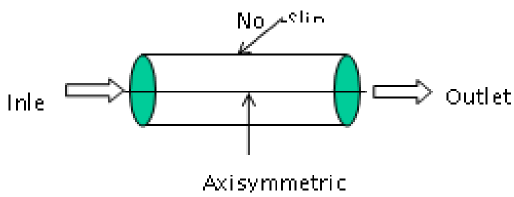

Fig. 2 Basic Cylinder.

Multi-Fluid VOF Model

When simulating multiphase flows, situations can arise in which neither the fully segregated VOF model nor the interpenetrating Eulerian multi-fluid model is adequate to describe flow dynamics. The frothy nose of an air–water slug or the motion of an air bubble through a slurry of fine particles illustrate applications in which regions of mixed flow coexist with large regions of segregated flow. For solving such problems, a hybrid model that describes the physics between the mixed dispersed phases as well as the physics at interfaces for segregated phases is built into ANSYS CFD.

V. MODLING

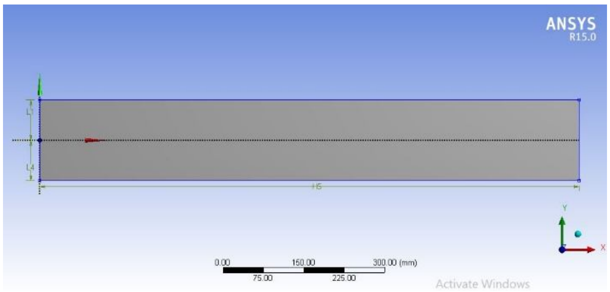

Fig. 2 2D Cylinder (ANSYS)

VI. MESHING

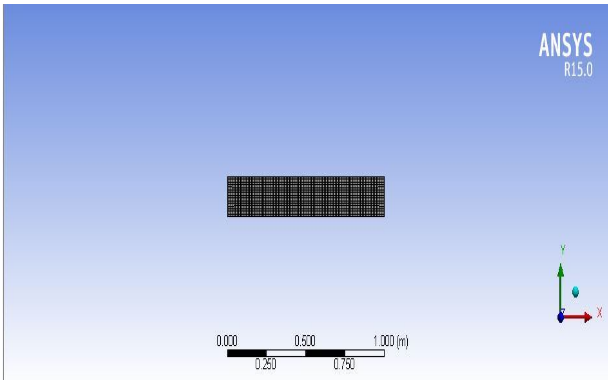

Fig. 3 Meshing of cylinder done in ANSYS.

Mesh Quality:

- Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
- Minimum Orthogonal Quality = 9.15463e-01
- Maximum Aspect Ratio = 1.32115e+01
- Mesh Size

Level	Cells	Faces	Nodes	Partitions
0	80934	162936	82003	1

VII. RESULTS

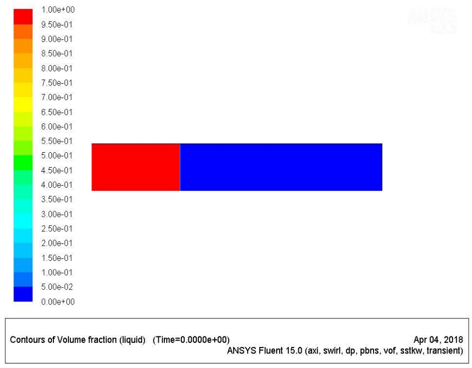

Fig. 4 Contours of Volume Fraction for Water(t=0sec)

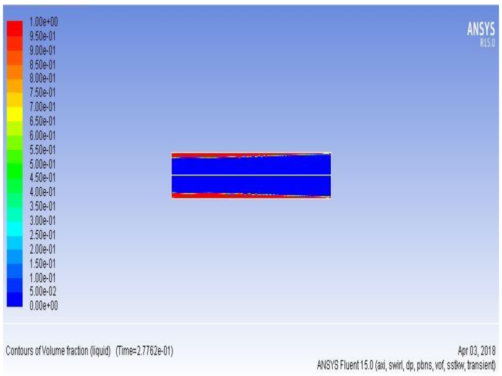

Fig. 5 Contours of Volume Fraction for Water(t=0.277sec)

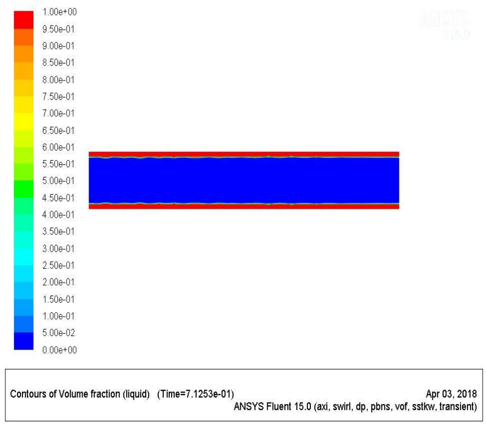

Fig. 6 Contours of Volume Fraction for Water(t=0.712sec)

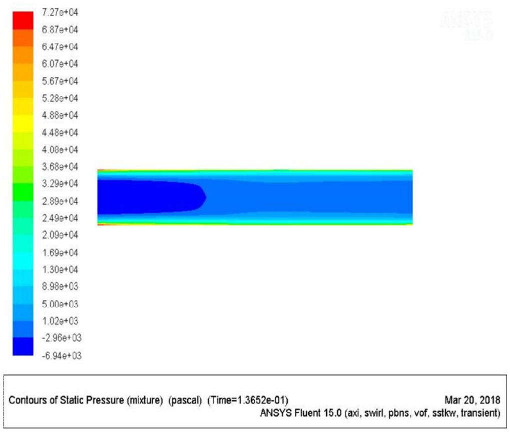

Fig. 7 Contours of Static Pressure for Water(t=0.136sec)

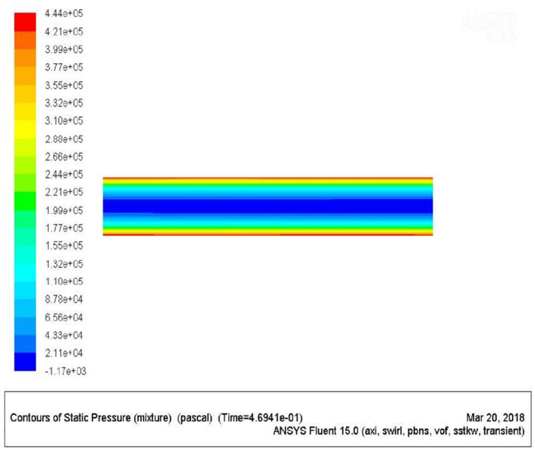

Fig. 8 Contours of Static pressure for Water(t=0.469sec)

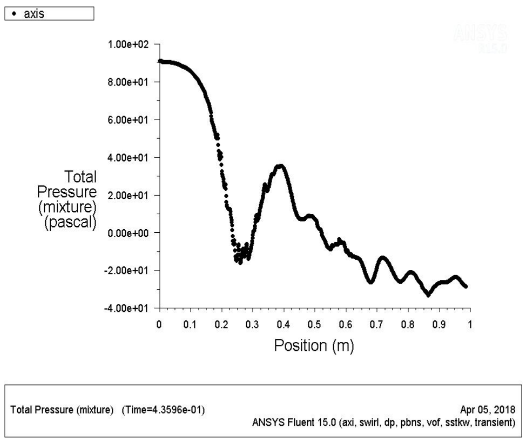

Fig. 7 Plot of Total Pressure of mixture at t=0.435sec

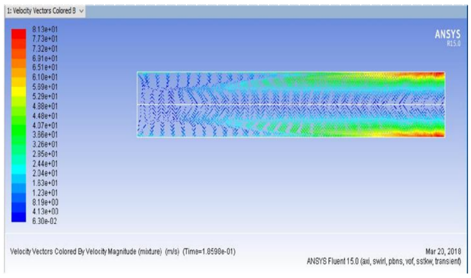

Fig. 8 Velocity Vectors at t=0.859

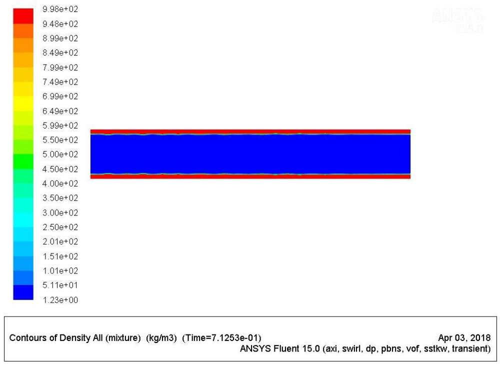

Fig. 9 Contours of Density at t=0.712sec.

VIII. BENEFITS

Cylinders and shapes with rotational symmetry are most commonly cast by this technique. "Tall" castings (in the direction of the settling force acting, usually gravity) are always more difficult than short castings. In the centrifugal casting technique the radius of the rotation, along which the centrifugal force acts, replaces the vertical axis. The casting machine may be rotated to place this in any convenient orientation, relative to gravity's vertical. Horizontal and vertical axis machines are both used, simply to place the castings longest dimension conveniently horizontal Thin-walled cylinders are difficult to cast by other means, but centrifugal casting is particularly suited to them. To the rotation radius, these are effectively shallow flat castings and are thus simple.

Centrifugal casting is also applied to the casting of disk and cylindrical shaped objects such as railway carriage wheels or machine fittings where the grain, flow, and balance are important to the durability and utility of the finished product. Providing that the shape is relatively constant in radius, noncircular shapes may also be cast.

IX. FUTURE PROSPECT

One can also try to study the behaviour for varying temperatures.

The results discussed in the present work are based on assumptions and the phase change with respect to temperature has not been discussed. It is necessary to understand this process in order to obtain more accurate results.

X. CONCLUSION

Cold Flow computation results are likely to clarify the effects of rotation on the fluid partially filling a rotating circular cylinder and also to determine the free surface of the liquid as a function of the RPM, cylinder diameter and the fluid properties. The CFD Code solves the coupled equation system (five nos.) for the three momentum components, continuity and another one for passive scalar called Volume of Fluid (VOF) to handle simultaneously the air and the liquid together with a free surface inside the cylinder .

Computation for the hot liquid metal (given solidification temperature, density, specific heat, latent heat of solidification and thermal diffusivity) rotating in a cold cylindrical mould (given thickness , density, temperature, thermal diffusivity and rotational speed) will solve the coupled equation system (six nos.) for the three momentum components, continuity and VOF to predict the flow and temperature field due to the complex transport processes in a vertical Centrifugal Casting System

REFERENCES

[1] RK. S. KEERTHIPRASAD, "Numerical Simulation and Cold Modeling experiments," vol. 42B, 2010

[2] J. O. R. Zagórski, Pouring mould during centrifugal, Krasiñskiego, Katowice: Silesian University of Technology, faculty of material science and metallurgy, 2007.

[3] D. D. V. P. A. S. P. Vinay Chandran, CFD Simulation of Centrifugal Casting of Al-SiC FGM for the, Kerala, Kerala: Department of Mechanical Engineering, MA College of Engineering, 2015.

[4] Kestur Sadashivaiah Keerthiprasad, Mysore Seetharam Murali,Pudukottah Gopalingar Mukunda, and Sekhar Majumdar.

[5] Kirti Kanaujiya, Yugesh Mani Tiwari, ISSN: 2278-9359 (Volume-4, Issue-10)

[6] Madhusudhan, "Experimental Study on Rate of Solidification of Centrifugal Casting", International Journal of Mechanical and Materials Engineering (IJMME), Vol. 5 (2010), No. 1, 101-105.

[7] Yikun Luan, "Effect of solidification rate on the morphology and distribution of eutectic carbides in centrifugal casting high-speed steel rolls", Journal of Materials Processing Technology 210 (2010) 536–541.