RESEARCH ARTICLE OPEN ACCESS

Simulation of Stir Casting Process Using Computational Fluid Dynamics

M. V. S. Pavan Kumar*, M. V. Sekhar Babu**, Dr. VSNV Ramana***
*(M.Tech Student, Dept. Mechanical Engineering, GMRIT, Rajam, Andhra Pradesh, India.)
** (Associate Professor, Dept. Mechanical Engineering, GMRIT, Rajam, Andhra Pradesh, India).
*** (Associate Professor, Dept. Mechanical Engineering, GITAM University, Visakhapatnam, AP, India).

ABSTRACT
Stir casting process is one of the methods to produce Metal Matrix Composites (MMCs). But the Particle Distribution of Non-Homogenous material is the greatest problem facing now days to produce MMCs. The present simulations were conducted how the speed of the stirrer effects the Particle Distribution of Non-Homogenous material. The Simulations were performed using Computational Fluid Dynamics. In this experiment Copper is used as Semi Solid Metal (SSM) and Silicon-Carbide is used as solid metal. The simulations were carried out by varying the stirrer speed 200 rpm and 400 rpm while keeping other parameters constant such as Viscosity 4.4 mPa-s and Blade Angle 60 degrees. The results show that better distribution obtained for the speed 400 rpm.
***Keywords* -** MMCs, Simulation, Stir Casting,

I. INTRODUCTION

Metal matrix composites (MMCs) are a range of advanced materials providing properties which are not achieved by conventional materials. These properties include increased strength, higher elastic modulus, higher service temperature, improved wear resistance, decreased part weight, low thermal shock, high electrical and thermal conductivity, and low coefficient of thermal expansion compared to conventional metals and alloys [1,2]. The excellent mechanical properties of these materials and the relatively low production cost make them very attractive for a variety of applications in automotive and aerospace industries. According to the type of reinforcement, the fabrication techniques can vary considerably. There are different techniques are available for the production of MMCs, among those stir casting is one of the technique. Stir Casting involves the addition of particle reinforcement into molten metal or semi solid metal (SSM) by means of agitation. The advantage of stir casting lies in a lower processing temperature leading to a longer die life and high production cycle time. Although stir casting is generally accepted as a commercial route for the production of MMCs, there are however technical challenges associated with producing a homogeneous, high density composite. Effectiveness with which mechanical stirring can incorporate and distribute the particles throughout the melt depends on the constituent materials, the stirrer geometry and position, the speed of stirring [3-4]. In this study viscosities of pure copper at different temperatures are taken from [5-6] resulted using torsion oscillation viscometer. Unfortunately, in normal practice the effect of the stirring action on the flow patterns cannot be observed as they take place in a non-transparent molten metal within a furnace. As such, and because of the fact that direct measurements of metal flow characteristics can be expensive, time consuming and dangerous. By taking this into consideration we can approximate the flow characteristics at such high temperatures using simulation software. Now a day's Computational Fluid Dynamics (CFD) is used for simulation due to low research cost, short research period and detailed description to fluid dynamic behaviour, computational fluid dynamics have received rapid development in recent decades. In this study, commercial CFD software was used to simulate stir casting process [7].

II. COMPUTATIONAL DOMAIN

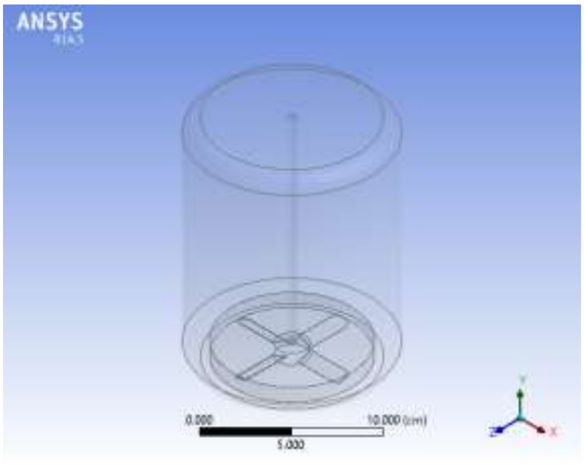

Figure 1: Geometry of Stirred Tank

To perform simulations, a stirred tank domain was created using Design Modeler of Ansys work bench. 10 cm Diameter and height of 16 cm tank, four bladed stirrer with shaft created using sketching and modeling. Fluid domain is created by subtracting blade assembly from the tank. To specify rotation motion in simulation a separate inner zone created surrounding the rotating blades. Finally parameterization also done to change the blade angle and radius of blade.

III. MODELLING AND NUMERICAL SIMULATION

The simulations were performed using Multiphase Mixture model available in the commercial CFD software, ANSYS Fluent 14.5. The continuity, momentum equations are solved for phases and coupling between phases is obtained pressure and interphase exchange coefficients in solver.

3.1 Simulation Set up

3.1.1 Mesh: The meshes were constructed in the commercial CFD Software ANSYS MESHING 14.5. Ansys meshing is one of the critical aspects of engineering simulation. Too many cells result in longer solver runs, and too few may lead to inaccurate results. ANSYS meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. Fluid dynamics simulations require very high-quality meshes in both element shape and smoothness of size changes. There are different types of mesh types are available. Present in our simulations tetrahedral mesh was generated. For volume meshing, a tetrahedral mesh provides more automatic solution with the ability to add mesh controls to improve the accuracy in critical regions, conversely hexahedral mesh provides a more accurate solution but it is more difficult to generate. Tetrahedron consists of six degrees of freedom (DOF), four faces, six edges, four vertices.

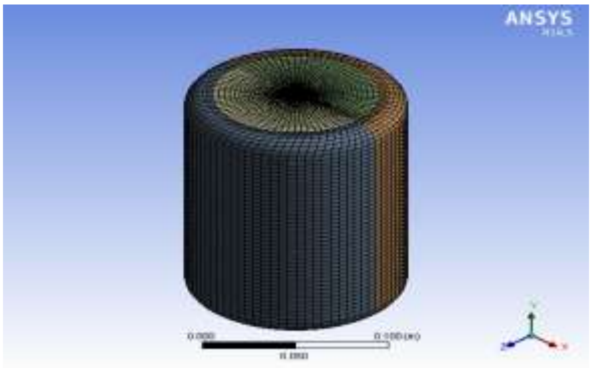

Figure 2(a) : CFD Meshes for stirred tank.

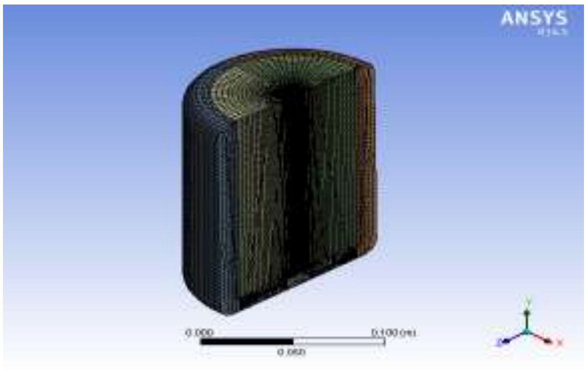

Figure 2(b) Half-sectional view for stirred tank with mesh

3.1.2 Physical Models and Boundary Conditions:

A viscous laminar model and a mixture model of two phases were the models used to perform simulations. The realizable k-ε turbulence model was used to resolve the turbulent solid-liquid flow field. Moving wall motion used to specify the rotational motion for the stirrer in Y-direction. No slip shear condition and smooth condition is used for wall. A simple pressure-velocity coupling scheme method and Least squares cell based Gradient option was used for gradient calculations at cell interfaces and first order up wind discretization scheme was used for momentum, volume fraction and turbulence equations. The unsteady state solver was utilized to solve for all flow variables. Out of the total volume of the tank, 5 percent volume of the tank filled with Silicate Carbide particles at the bottom of the tank, remaining volume filled with Copper Semi Solid Metal. Two regions were created in the stirred tank, one region patched to Copper and another region is patched to Silicon-Carbide.

3.2 Run for Calculation:

Before going to run for calculation here we varying one parameter Speed and kept constant other two parameters i.e. Blade Angle and Viscosity of Semi Solid Metal with time.

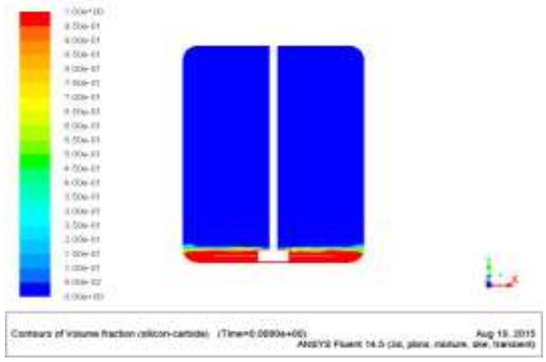

Figure 3: Initial Volume Fraction of Silicon-Carbide at' 0' seconds

The Fig 3 shows the volume fraction of Silicon-Carbide over Copper Semi Solid Metal at zero seconds. In Run for calculation we opted for 100 iterations of 60 seconds period of time. After Completion of run for calculation the results were obtained for two conditions.

IV. RESULTS

For the condition

Viscosity= 4.4 mPa-s, Speed=200 rpm, Blade Angle = 60 Degrees.

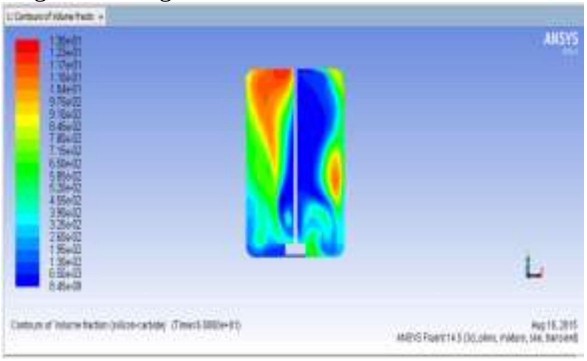

Figure 4 : Contours of volume fraction of Silicon-Carbide for 200 rpm

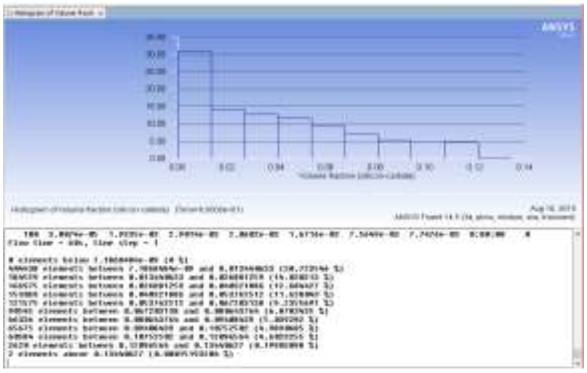

Figure 5: Percentage of Volume fraction of Silicon-Carbide Presence at different regions.

For the Condition

Viscosity= 4.4 mPa-s, Speed=400 rpm, Blade Angle = 60 Degrees.

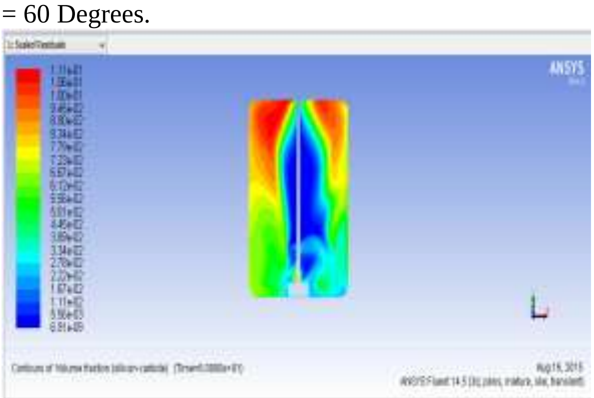

Figure 6 : Contour of volume fraction of Silicon-Carbide for 400 rpm

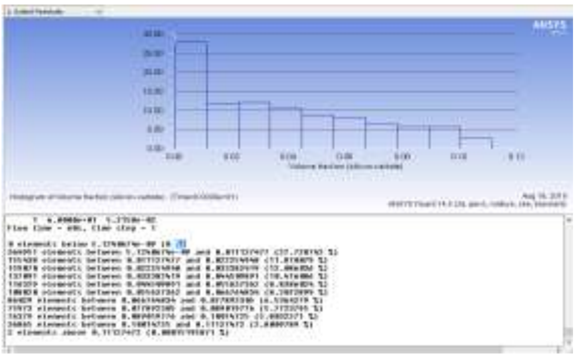

Figure 7: Percentage of Volume fraction of Silicon-Carbide Presence at different regions.

Results of simulations for stirrer speed 200 rpm and 400 rpm are shown in figures 4 to 6. Percentage of volume fraction i.e., region in blue color in figure 4 is more compared to the percentage of volume fraction in figure 6. It indicates that as speed increases from 200 to 400 rpm the percentage of volume fraction decreases. Regions shown in colors other than blue, are more in figure 6 compared to figure 4. It indicated that the decrease in low percentage volume is pushed to high percentage volume. Comparison of figures 5 and 7 supports the above observations i.e., the shift of particles from low volume fraction zone to high volume fraction zone.

V. CONCLUSION

Stirring process in Stir casting is successfully simulated using CFD Software. By studying the results of the simulation it is concluded that at 400 rpm the percentage of low volume fractions are less compared to the low percentage of volume fractions at 200 rpm of stirrer speed. Similarly the percentage of high volume fractions is high at 400 rpm compared to the high percentage of volume fractions at 200 rpm. By this simulation results it was concluded that the parameter stirrer speed plays a significant role in the distribution of Silicon-Carbide particles over the Copper Semi-Solid Metal.

REFERENCES

[1] S. Naher, D. Brabazon, L. Looney, Simulation of the stir casting process, Journal of Materials Processing Technology 143–144 (2003) 567–571.

[2] S. Naher, D. Brabazon, L. Looney, Computational and experimental analysis of particulate distribution during Al–SiC MMC fabrication, Composites: Part A 38 (2007) 719–729.

[3] G. Bartos, K. Xia, in: Proceedings of the Fourth International Conference on Semi-Solid Processing of Alloys and Composites, University of Sheffield, UK, 1996, p. 290.

[4] S. Zhang, F. Cao, Y. Chen, Q. Li, Z. Jiang, Acta Material Composite Silica 15 (1) (1998) 88.
[5] O. K. Echendu and B. C. Anusionwu, An Investigation of the Viscosities of Various Groups of Liquid Metals, The African Review of Physics (2011) 6:0005.
[6] Mao Tan, Bian Xiufang, Xue Xianying, Zhang Yanning, Guo Jing, Sun Baoan, Correlation between Viscosity of molten Cu–Sn alloys and phase diagram, Physica B 387 (2007) 1–5.
[7] ANSYS Workbench, Fluent 14.5 Users Guide.