iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 Issue: III Month of publication: March 2018

DOI: http://doi.org/10.22214/ijraset.2018.3128

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue III, March 2018- Available at www.ijraset.com

Study of Effect of Metal Delivery Tube on Flow characteristics of Fluids and Melt in Convergent-Divergent nozzle used for Gas Atomization Process using CFD techniques

Sanjay Phatige[1], Anand S. N[2]
[1,2]Dept. of Mechanical Engineering, REVA University, Bangalore- 560064

Abstract: Atomization is one of powder production techniques because of high production rates and ability to make alloy powders of desired composition. In gas atomization process, liquid metal is broken into droplets to form powders upon solidification. Gas-metal interaction influences the break-up of liquid stream into droplets. The idea is to transfer kinetic energy from a high velocity jet-gas expanded through a nozzle, to a stream of liquid metal, resulting in fragmentation and break up into metal droplets. Gas atomization process is one of the widely used powder production technique and nozzles play an important role in the gas atomization process. The geometry of nozzles governs the gas to metal interaction. The selection of nozzle type and the flow geometry is the most important preset parameter for atomization process. The design of an atomizing nozzle determines degree of contact of liquid metal with the atomizing gas.

Presence of Metal Delivery Tube (MDT) coaxially throughout the flow passage i,e full length of the nozzle will affect flow characteristics of the fluid along radial and axial direction. This helps to analyze flow behavior of gas when MDT is present in the nozzle during gas atomization process.

In the present analysis, an attempt was made to analyze effect velocity/Mach number and temperature of the fluid and melt when MDT is present inside the nozzle during the atomization process using the CFD software Fluent.

Key words: Gas Atomization, Convergent nozzle, MDT, CFD, Fluent

I. INTRODUCTION

A. Gas Atomization process

Gas atomization process helps in production of a wide range of ultra fine spherical metal powders which have very attractive material properties. In gas atomization process, atomization pressure plays an important role in determining particle size and surface morphology [1]. Atomizing nozzles have co-axially placed Metal Delivery Tube (MDT), which carry molten metal to atomizing zone [2, 3]. The flow properties of gas are considerably affected by the presence of MDT. In this process, inert gas (N2) is used an atomizing medium. Atomization technique has been extensively reviewed by Beddow [4], Grant et al [5] and Lewley [6].

Gas atomization process is one of widely used powder production technique. In gas atomization process, molten metal is melted and liquid metal is broken into individual particles to form powders. Gas atomization is one of the powder production method/technique. In this method, kinetic energy of a high velocity (greater than Mach 1) impinging inert gas jet disintegrates continuous metal flow into droplets. Nozzles are used to achieve the atomization through break-up of molten metal stream into droplets by fast flowing gas [7]. The heart of the gas atomization process is the nozzle. The set-up for gas atomization of liquid metals using a nozzle assembly is shown in figure- 1

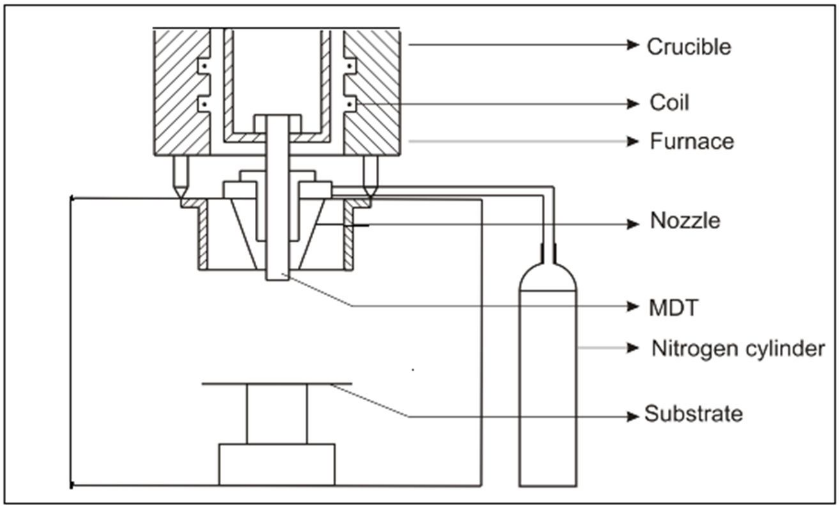

Fig. 1: Atomization set- up using Nozzle

CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. CFD predicts accurately practical flow parameters. CFD method is being increasingly used in order to study fluid flow and heat transfer problems which otherwise is a complex task to achieve by experimental investigation. CFD is the art of replacing differential equations which governs fluid flow with the set of algebraic equations which in-turn can be solved with the aid of digital computers to get approximate solution. The well-known discretization methods used in are CFD Finite Difference, Finite Element Method (FEM) and Finite Volume Method (FVM). The fundamental equation of any CFD problem is Navier-Stokes equations, which define any single-phase fluid flow. [8]

CFD techniques are used in gas atomization process to analyze flow characteristics of gas at nozzle exit and atomizing zone. In addition heat transfer aspects of gas and MDT can be analyzed by this technique.

The convergent divergent nozzle is fitted with co-axially fitted MDT to carry the molten metal to the atomization zone.

II. METHODOLOGY FOLLOWED

Analysis of CD nozzles nozzles with presence of MDT for parameters like flow and heat transfer aspects was carried out. CFD trials were conducted to determine gas and melt flow characteristics with specific process parameters. A nozzle with a co-axially placed MDT was considered for analysis.

A. Conducting CFD trials by applying desired input parameters of gas, melt and MDT (for ex: velocity and temperature)
B. Analyzing CFD trial plots and comparing with experimental results
C. Mach number was plotted with the presence of MDT along radial and axial direction of gas flow from nozzle exit. Comparison of Mach number with reported Pitot tube experiments results was carried out for CD nozzle.
D. Velocity plots were obtained with the presence of MDT for analysis of gas velocity at nozzle exit and atomizing zone. Velocity was compared with analytical values obtained from first principle
E. Velocity vector plots were obtained with and without presence of MDT. This was to analyze presence of shockwaves at the nozzle exit which affects gas atomization process.
F. Temperature plot was obtained to determine temperature of gas and MDT at nozzle exit.
G. Free convection and forced convection analysis for inert gas and MDT was carried out based on temperature plots. This was to determine pre-heating requirements of gas and MDT in order to prevent chilling of gas and melt and choking of molten metal in side MDT.

III. CFD TRAILS TO OBTAINED FOR CD-NOZZLE

In the present investigation study, CFD trials have been carried on CD-nozzle for plotting Mach number, velocity and temperature. Geometric modeling and meshing of CD-nozzle were carried out and results were plotted using fluent software. The CD-nozzle parameters considered for simulation of greater than 2 Mach number, supersonic) are shown in table- 1.

Boundary conditions applied were Pressure inlet, Pressure outlet, Wall and Symmetry. Viscous model was defined as Spalart-Allmaras for the fluid flow, solver as Fluent 5/6 and mesh type as QUAD [9]. Inlet pressure of 0.3 MPa (3 Bar) and inlet temperature values of 300K were given as input. The iteration values were set was about 1000 initially and iterated until the solution converges [10]. Then results were plotted for pressure. Courant number controls the time step used by Fluent during inner iterations performed during each time step.

Sl No	CD Nozzle dimensions and parameters	Values
1	Inlet diameter	23.35 mm
2	Outlet diameter	24.12 mm
3	Throat diameter	16.00 mm
4	Distance between inlet and nozzle throat	14.78 mm
5	Distance between nozzle throat and outlet	19.71 mm
6	Inlet pressure	0.3 MPa
7	Gauge pressure	0.1 MPa
8	Exit Mach	2.4 Mach
9	Temperature of gas at inlet	300 K
10	MDT dimensions	ID = 6mm, OD = 8mm, t =1 mm
11	MDT length	L = (35+3) m
12	Nozzle protrusion length to obtain maximum aspiration pressure	3 mm
13	Inert gas used	Nitrogen

Table 1 Process parameters for CD nozzle

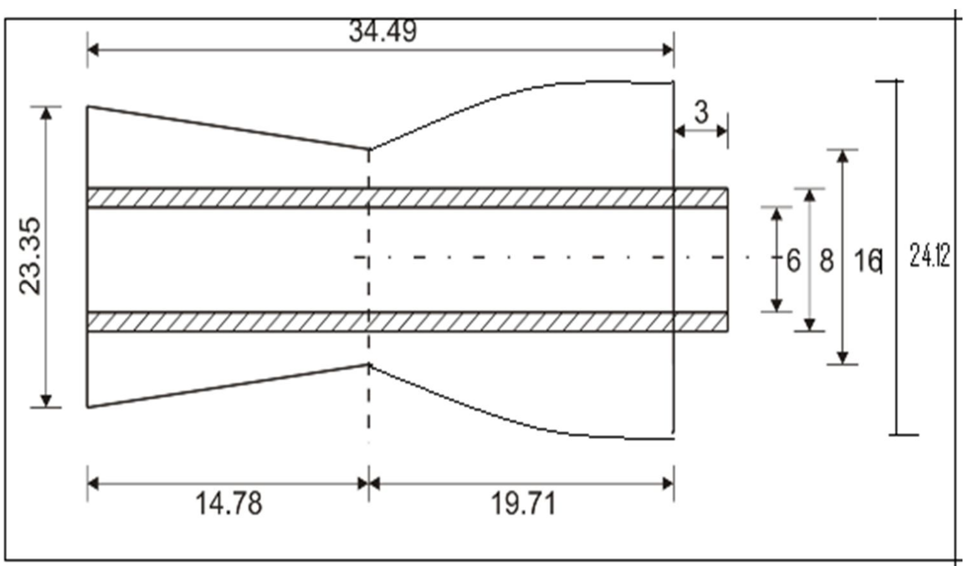

Fig-2 : CD nozzle with co-axially fitted MDT

Geometrical mesh of both nozzle and atomizing zone were generated using Gambit software shown in fig-3. The boundary conditions applied were pressure inlet, pressure outlet, wall and symmetry. The geometric mesh created was used in Fluent software for analysis and plotting of CFD results [11].
The boundary conditions applied for CD nozzle are as mentioned in table - 2

Sl no	Boundary condition details	Values
1	Boundary condition applied at inlet	PRESSURE_INLET
2	Boundary condition applied at outlet	PRESSURE_OUTLET
3	Boundary condition applied for atomization zone	PRESSURE_OUTLET
4	Boundary condition applied at the axis	Symmetry
5	Boundary condition applied at nozzle wall	Wall
6	Boundary condition applied for MDT	Wall

Table 2: Boundary condition details for CD nozzle

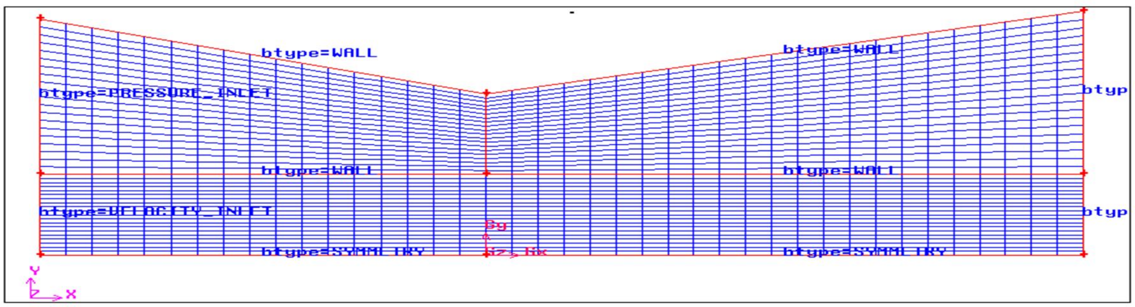

Fig. 3 - CD nozzle with MDT (Meshed, using Fluent)

IV. RESULTS OBTAINED USING CFD TRAILS

A. Analysis of Mach number – Radial direction

CFD trials are conducted with presence of co-axially placed MDT inside the nozzle and following cases were considered for the analysis.

Mach number plot along radial direction from nozzle tip

Mach number plot along the axial direction from nozzle tip towards the substrate in the atomization zone.

Figure 4 shows Mach number plot which was obtained for CD nozzle when MDT was placed co-axially placed in the nozzle. Here Mach number increases from 2.2 to 2.3 and later decreases 2.1 towards nozzle wall in radial direction. Mach number of 2.3 is obtained towards nozzle wall.

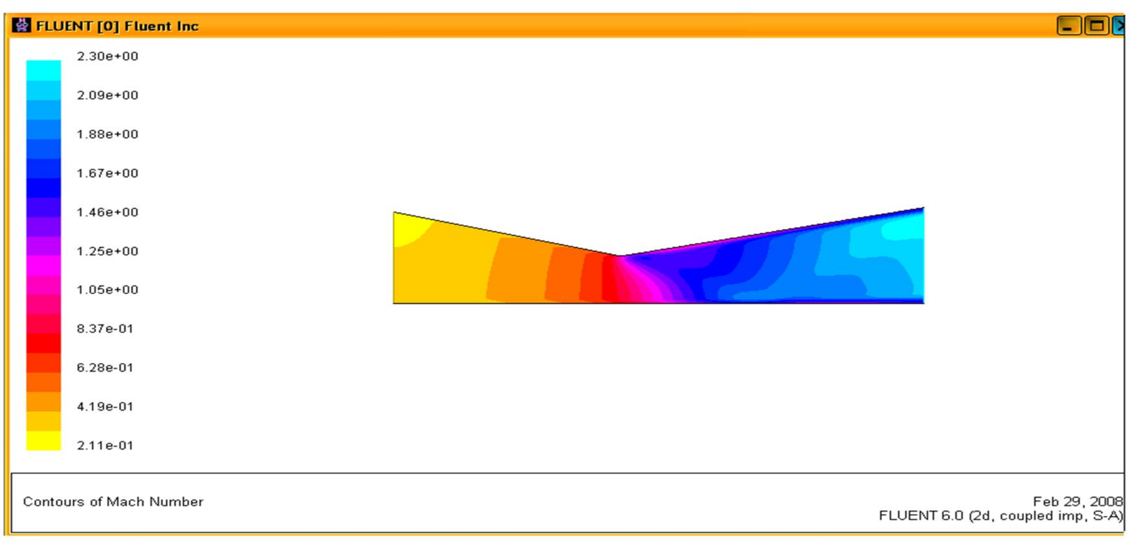

Fig. 4 Mach number plot for CD nozzle with MDT (radial)

Figure 5 shows CD nozzle with co-axially placed MDT and points where Mach number was measured along radial direction at the nozzle exit [12].

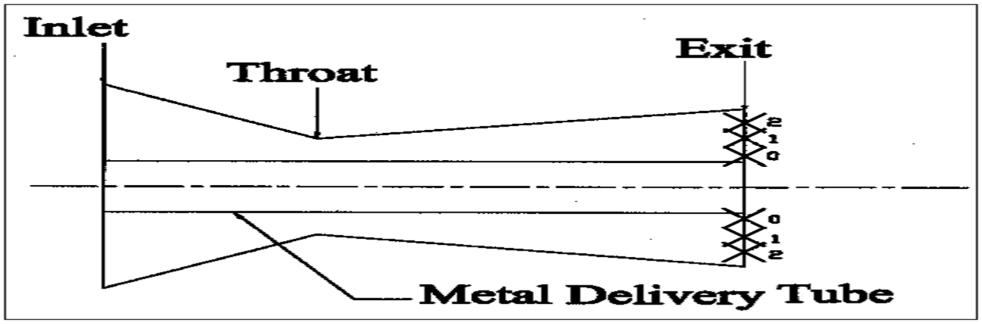

Fig. 5 Points for Mach number measurement experimentally - with MDT

Figure 6 shows Mach number comparison plot for wind tunnel experiments [12] and CFD trials along radial direction for CD nozzle with presence of MDT. It could be seen that for wind tunnel experiments [12], Mach number was maximum (closest to designed value 2.3 Mach) at center of annular region and decreases (2.1 Mach) towards nozzle periphery.

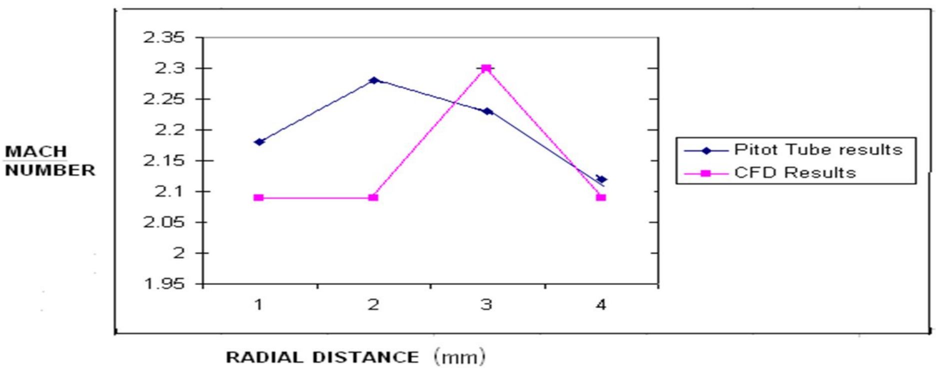

Fig. 6 Comparison of Mach number between wind tunnel and CFD results – radial direction with MDT

For CFD trials, nearly same trend was observed. Mach number increases from Mach number 2.1 to 2.3 and decreases to Mach number 2.05. In both cases (experimental and CFD trials), Mach number first increases and decreases radially at nozzle exit.

B. Analysis of Mach number – Axial direction

Figure 7 shows CD nozzle with a co-axially placed MDT where model and mesh were generated using Gambit and later boundary conditions were applied. This is exactly similar to the nozzle condition as explained in figure 7 but it has co-axially placed MDT.

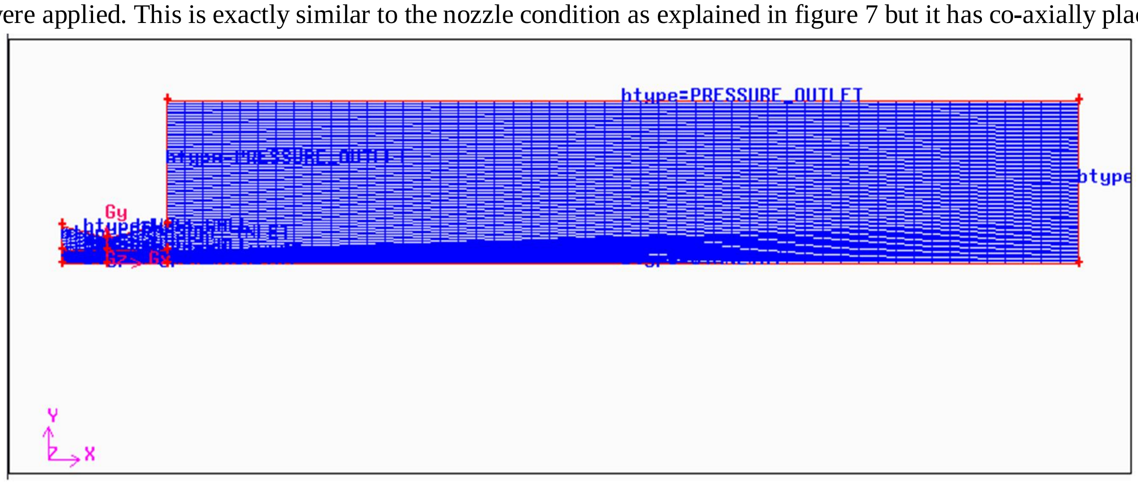

Fig. 7 CD nozzle with MDT for plotting of Mach number along axial direction (meshed)

Figure 8 shows Mach number distribution plot for CD nozzle using CFD with MDT along axial direction. The plot shows presence of shock waves. Mach number of 2 is obtained at nozzle exit and decreases beyond nozzle exit and it varies towards substrate. The Mach number also varies due to presence of shock waves.

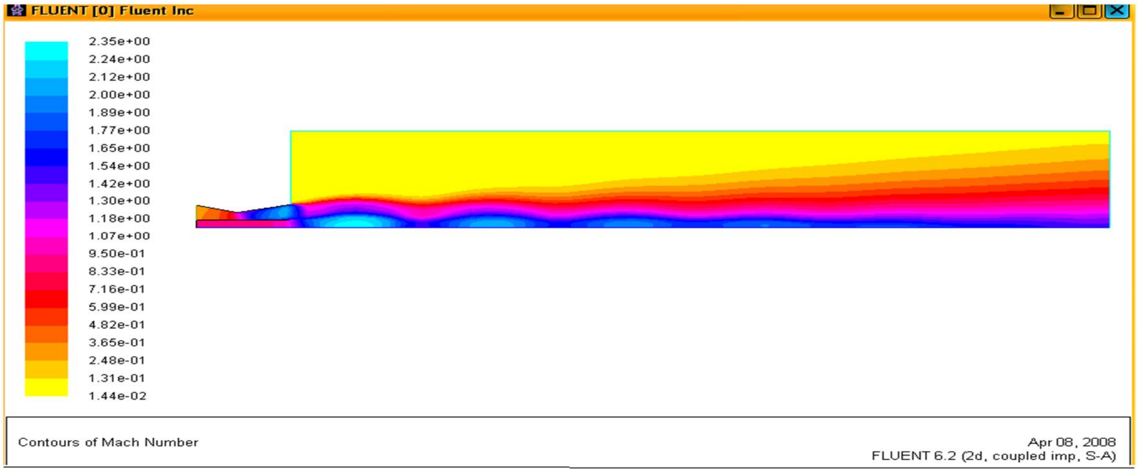

Fig 8 Mach number plot for CD nozzle with MDT along axial direction

Figure 9 shows location of points where Mach number was measured and Mach number plot from wind tunnel experimental values set up respectively for CD nozzle with MDT [12].

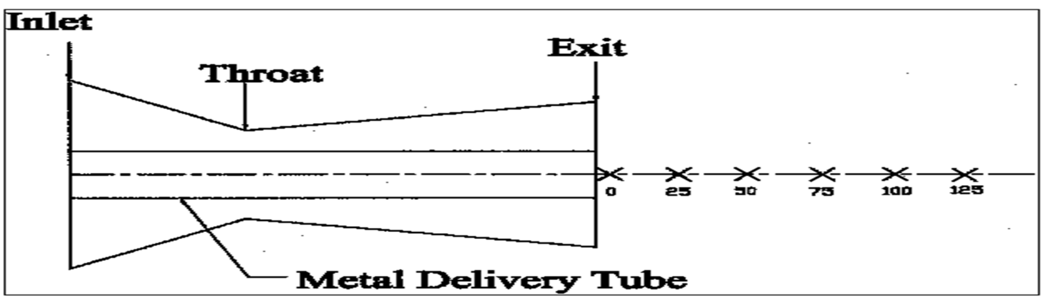

Fig. 9 Points at which velocity was measured with MDT in axial direction

Figure 10 shows Mach number comparison plot for wind tunnel experiments [12] and CFD trials. Mach number increases up to 3 at a distance of 275 mm indicating continued expansion in the case of wind tunnel experiments. This is similar to condition where over expanded flow was observed without MDT. In the case of CFD trials, Mach number decreases (2 to 1.3) along axial distance as velocity decreases.

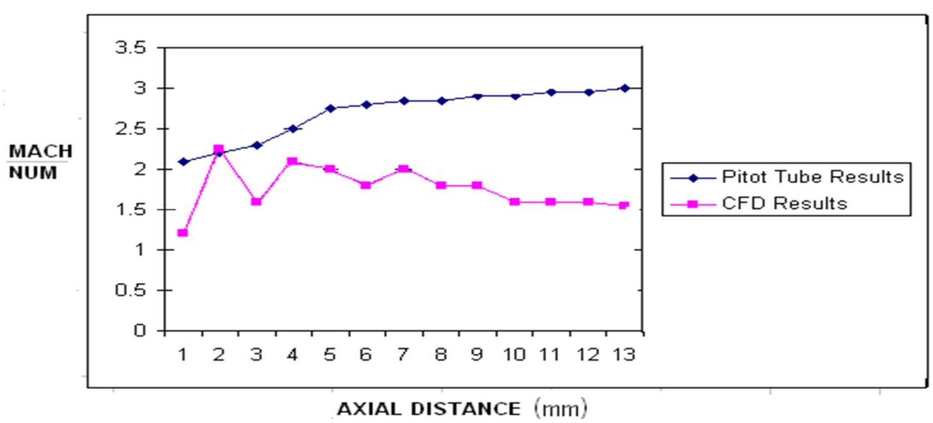

Fig. 10 Comparison of Mach number variation between wind tunnel and CFD results in axial direction – CD nozzle with MDT

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue III, March 2018- Available at www.ijraset.com

To sum up, wind tunnel results [12] showed an increase in Mach number along axial direction from desired Mach number of 2 to 3 and continue expansion of gas. But CFD results show that Mach number varies inside the nozzle from subsonic to supersonic (Mach number 2.35) and decreases with axial distance towards the substrate (Mach number 2.25 to 1.50).

C. *Temperature Analysis*

Methodology adopted for heat transfer simulation of Aluminum alloy

Input parameters for model formulation [12]

1) Alloy considered for gas atomization was Al-Si-Mg (A356 alloy)
2) MDT material used: Stainless steel (SS316
3) Atomizing inert gas used: Nitrogen (N2
4) Density of metal : 2700 Kg/m3
5) Stagnation temperature : 300 K
6) Cp/CV of gas = 1.4

This section describes set up of simulation environment for molten metal flow inside MDT. First pull velocity was defined for simulation of metal flow inside MDT for gas atomization. Later inlet and outlet velocity of molten metal, input temperature, heat transfer co-efficient, pre-heating temperature of molten metal was defined. This problem defines setup and solution procedure for a fluid flow and heat transfer analysis. The geometry contains a 2D axi-symmetric MDT [13]. The metal to be atomized was heated to its pouring temperature. There was a steady state injection of liquid metal at pre-defined velocity (v1), at pouring temperature of liquid metal and then it is pulled out of MDT at a different velocity (v2). MDT was maintained at a temperature which is equal to pouring temperature of molten metal in order to avoid chocking of MDT.

D. *Process parameters considered for heat transfer analysis*

Table 3 shows boundary conditions/process parameters considered for heat transfer analysis

Sl No	Process Parameters	Values
1	Head of liquid metal (h1)	100 mm
2	Inlet velocity of molten metal [v1 = sqrt (2*g*h1)]	1.4 m/s
3	Outlet velocity of molten metal [v2 = sqrt (2*g*h2)] CD nozzle: (h2 = h1 + Length of MDT h2= 100 + 38 = 138 mm)	1.68 m/s
4	Temperature of Al alloy (850 deg, pouring temperature)	1125K
5	Solidous temperate of metal (Melting temperature 650 deg C)	925 K
6	Mass of metal used	4 kg
7	Thermal conductivity (h)	h = 800 W/ (m*m-K)
8	MDT dimensions	ID = 6 mm, OD = 8 mm, t =1 mm
9	Model solver used for GAMBIT	Segregated
10	Nature of flow used	Steady state flow
11	Turbulence model	Spalart-Allmars
12	Initial nozzle wall/MDT temperature	300K

13	Initial gas temperature at inlet	300K
14	Boundary condition applied to liquid metal	Velocity inlet

Table 3 Process parameters for heat transfer analysis

1) *Free convection (FC) condition*
a) MDT was preheated to 1125K (pouring temperature of Al alloy)
b) Liquid metal temperature: 1125K

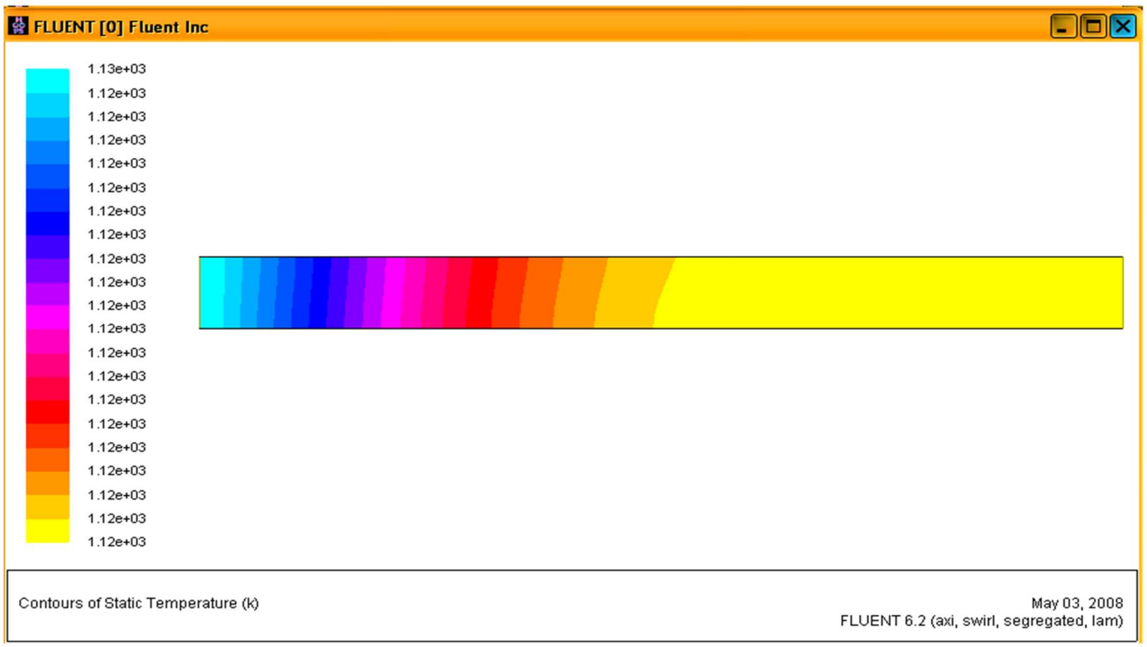

Fig.11. Temperature plot for condition FC_CDN3: MDT tip temperature of 1120K

MDT temperature should be equal to liquid metal temperature in order for free flow of molten metal and to avoid choking of nozzle. The temperature of melt at nozzle exit was found to be at 1120K which is nearly equal to pouring temperature of melt. This temperature of melt allows flowing through MDT. This is the most favorable condition for gas atomization to occur.

2) *Forced convection (FRC) conditions:* Melt will be initially at its pouring temperature at entry of flow passage. But when it flows through MDT, some portion of heat is convected out to the gas. Hence temperature of melt decreases along its flow passage. This condition necessitates preheating requirement of melt. Gas was passed at temperature of 300K at nozzle inlet. For this condition, temperature was found to be 160K at the nozzle exit. This temperature will cause chilling of melt and chocking of MDT. Hence gas has to be preheated to get its room temperature of 300K.
a) MDT was maintained at 1125K
b) Gas was passed at inlet temperature of 575K (gas was preheated)
c) Liquid metal was poured at inlet temperature: 1125K

For FRC condition shown in figure 12, temperature of gas was found to be 300K at the nozzle exit. This will be the desirable condition for gas atomization. If the gas was not pre-heated, it will cause chilling of melt causing choking of the MDT in the nozzle.

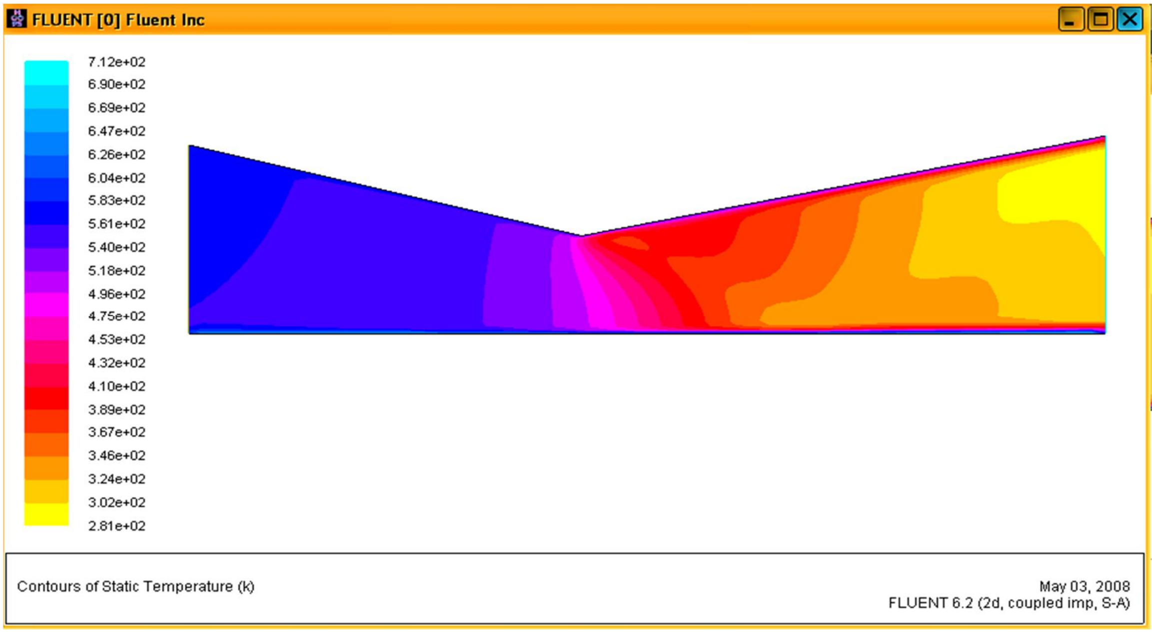

Fig 12 Temperature plot for condition FRC, exit gas temperature of 300K

E. Temperature Distribution With And Without Mdt

Figure 13 shows gas temperature distribution plot for CD nozzle with and without presence of MDT obtained through CFD trials. This was almost similar to condition reported in literature [1, 6, 12].

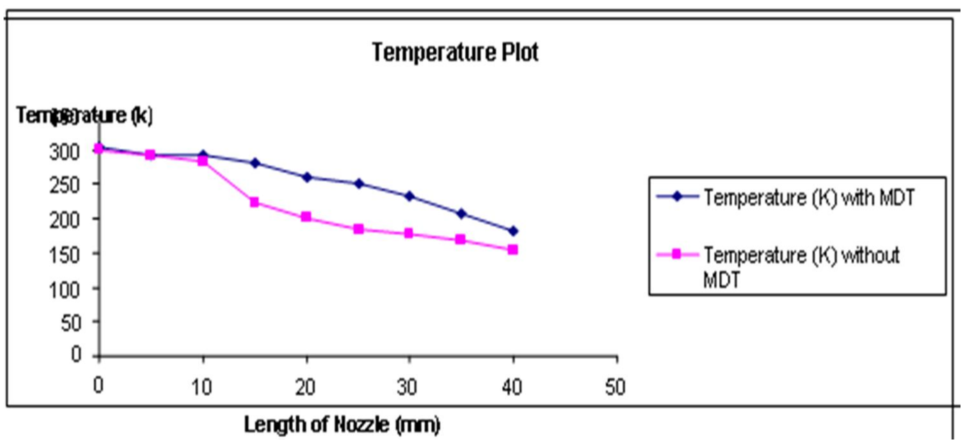

Fig.13 Variation of gas temperature as a function of length of the nozzle – CFD trials

The metal is initially at its pouring temperature at the entry of flow passage. But when it flows through MDT, some portion of heat is convected out to the gas. Hence the temperature of melt falls along flow passage. So temperature of melt and MDT will be least at the exit of nozzle due to maximum velocity of gas encountered at nozzle exit. So gas temperature will be as low as 160 K. Due to this, freezing of melt stream inside MDT would take place within seconds [13]. This necessitates the importance of preheating requirement of gas and MDT for gas atomization process.

V. CONCLUSIONS

Following values were observed when MDT is present in the CD nozzle

A. Radial Direction

Mach number increases from Mach number 2.1 to 2.3 and decreases to Mach number 2.05. In both cases (experimental and CFD trials), Mach number first increases and decreases radially at nozzle exit.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue III, March 2018- Available at www.ijraset.com

B. Axial direction

The Mach number distribution plot for CD nozzle using CFD with MDT along axial direction shows presence of shock waves. Mach number of 2 is obtained at nozzle exit and decreases beyond nozzle exit.

C. Temperature

Free convection condition: MDT temperature should be equal to liquid metal temperature in order for free flow of molten metal and to avoid choking of nozzle and this temperature of melt at nozzle exit should be 1120K.

D. Temperature

Forced convection condition: Gas need to be pre-heated to prevent chilling of melt causing choking of the MDT in the nozzle and this temperature is 575K.

E. Temperature

Temperature of melt falls along flow passage length of the nozzle (300K to 160K), which tells that gas need to be pre-heated to prevent chilling of melt and chocking of nozzle.

REFERENCES

[1] Grant P.S, “Progress in Material Science”, Vol 39, 1995, P 497-545
[2] Eun-Bo Shim, Jong-Youb Sah, “Numerical analysis of fluid flow in a confined type gas atomizer”, Vol 2, 1997
[3] J.K, Beddow, “Production of Metal powders by atomization”, Heyden, 1978.
[4] N.J. Grant, B.H. Kear and M. Cohen, “1st international conference on Rapid solidification and processing; Principles and technologies”, 1978
[5] A, Lawley, “International Journal of powder metallurgy”, 13 (1977), 169
[6] Unal A, “Influence of nozzle geometry in gas atomization of rapidly solidified aluminum alloys”, Materials Science and Tech, Vol 4, October 1988
[7] Grant P.S, “Spray-forming”, Progress in Materials Science, Pergamon, UK, Vol 39, 1995
[8] https://en.wikipedia.org/wiki/Computational_fluid_dynamics
[9] Lubanska H, “Correlation of spray ring data for gas atomization of liquid metals”, J Metals, Vol 22, 1970, pp 45-49.
[10] Anderson I.E, Figliola R.S, “Fundamentals of high pressure gas atomization process control” Adv. Powder Metall, Vol 5, 1991
[11] http://www.fluent.com/software/gambit/index.htm & Gambit user guide
[12] N S Mahesh, “Some studies on Spray casting of Aluminum-Silicon alloys”, PhD thesis, Department of Mech. Engg, Bangalore University, 2003
[13] Tuyet Le and Hani Henin, “ Effect of nozzle geometry and position on gas atomization”, International journal of Powder Metallurgy”, Volume,32, 1996