International Journal of New Technology and Research (IJNTR)
ISSN:2454-4116, Volume-2, Issue-5, May 2016 Pages 52-55

Improvement of Cooling Efficiency of Plasma Spray Gun Nozzle

Raghunandan M S

Abstract— **The nozzle of a plasma spray gun is one of the important parts that have to be considered for the better improvement of life of the plasma spray gun. This study simulated the nozzle using CFD (Computational Fluid Dynamics) by considering parameters like velocity and fluid flow direction. A numerical model is developed using SIMPLE (Semi Implicit Method for Pressure Linked Equations) for fluid flow and heat transfer conditions. Due to the formation of local turbulence, a static-water region is formed which decreases the efficiency of the cooling system. The nozzle is heated up with a temperature of 8000 K at the heat source point. Due to the characteristic of the nozzle, the heat source is assumed to fall at a point and is supplied with a temperature of 8000 K. The nozzle is also susceptible to burn out due to over-heating. The seal gasket at the rear of the nozzle is also susceptible to burn out due to the heat exchange between water and the inner wall of the nozzle. This can be overcome by changing the direction of water flow and also changing the water flow velocity through the coolant channel.**

Index Terms— **Heat Transfer, Analysis, Fluid Flow Analysis, Analysis Of Plasma Spray Gun.**

I. Introduction

The plasma spray coating of metal has been one of the major breakthroughs since 1970s. Plasma spray gun uses high temperature plasma jet generated by arc discharge with typical temperatures reaching more than 15,000 K. This makes possible for the refractory materials such as oxides, molybdenum and others for using metal coating purposes [1]. This high temperature plasma jet has to be cooled efficiently in order to have long lasting life of the spray gun. But cooling the nozzle is a tedious job when we have no idea about the insight of the cooling system. This can be overcome by using CFD with modeling and analysis. The analysis can be carried out using computers with virtually no real experiments to be done. The use of CFD has enabled us to make use of the simulations which are typically higher cost for real experimentation. Some experiments have been carried out in this field with comprehensive modeling and analysis of the nozzle. The work in [2], enable us to have an insight over the cooling system and nozzle characteristics under higher temperatures.

In this study, the temperature of the nozzle exceeded 12,000 K and it was simulated using k – Omega model. In [3], a Wire-feed, High Velocity Oxygen Fuel (HVOF) thermal spray torch gives better insight of the cooling system. This includes the flow field inside and outside of the air cap and particle velocity predictions. This

validated with the experimental measurements outside the cap. Simulation of thermal spray was carried out in order to see the operating regimes that have not been possible with the experimental simulations. In [4], the better performance of the plasma thermal spray gives us an idea about the improvement of the nozzle with changing the dimension of the nozzle according to the use. Thus making cooling system to work more efficiently. This can be achieved only when the nozzle size change with different velocities. The paper [5], gives us the broader idea about the modeling of nozzle in 2D, since the fluid flow and heat field being symmetry. And also the nozzle is analyzed with various velocity profiles for the improvement of the cooling system.

The present study is focused on the cooling system of the nozzle with two-dimensional model. The nozzle is used here is the convergent – divergent nozzle with approximate dimensions. The schematic diagram of the plasma spray gun is shown in Fig.1. The simplified model is shown in Fig.2, for our convenience it is simplified as shown. This makes us possible to study the nozzle with different velocities and also changing the coolant. The coolant used here is water at 300 C (303 K). This makes the water heat exchange further more efficient due to the characteristic of the nozzle. The model shown in Fig.2 is modeled using the Gambit modeling software. The mesh has also been carried out using this software. The analysis of the model is done using Ansys FLUENT, with prescribed boundary conditions.

The model is simulated using SIMPLE method, which is one of the numerical simulation techniques. Due to the change in area of the coolant pipe, pressure drop regions are formed. This region enables water to remain in the portion of the cooling channel. This is also due to the local turbulence, which exists at the front corner of the cooling channel. Hence decreasing the cooling efficiency of the nozzle. This has to be overcome by using simple geometrical optimization within the nozzle cooling system. And also the nozzle has been assembled with the use of seal gasket. The seal gasket is at the rear of the nozzle and is susceptible to the burning due to in-sufficient cooling. This problem can be overcome by reversing the direction of coolant flow into the cooling channel of the nozzle, i.e., by changing the inlet to outlet and vice versa. This has shown us substantial cooling efficiency in this study.

Raghunandan M S, Assistant Professor,Dept. of Mechanical Engineering, GSSSIETW, Mysore

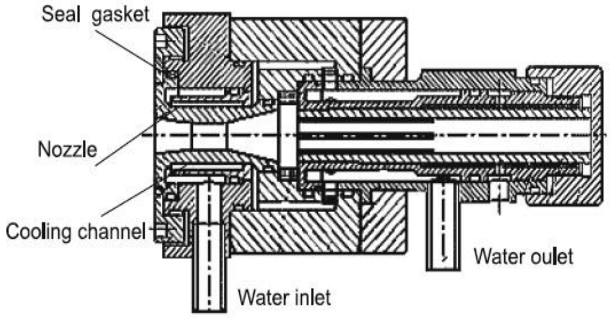

Fig.1 Schematic Diagram of Plasma Spray Gun

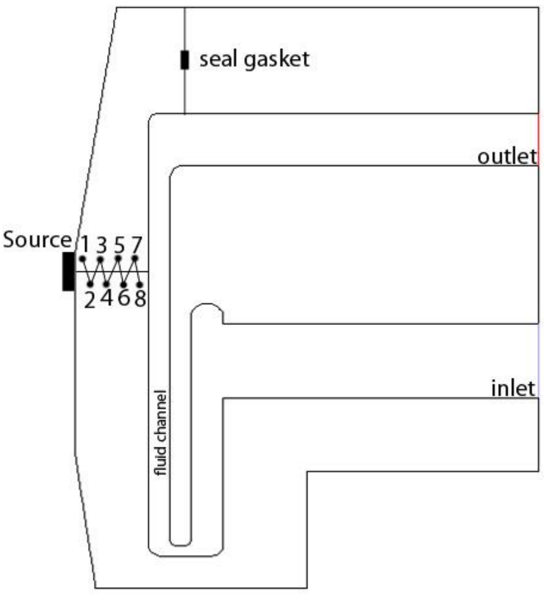

Fig.1 (a). Simplified View of the Nozzle

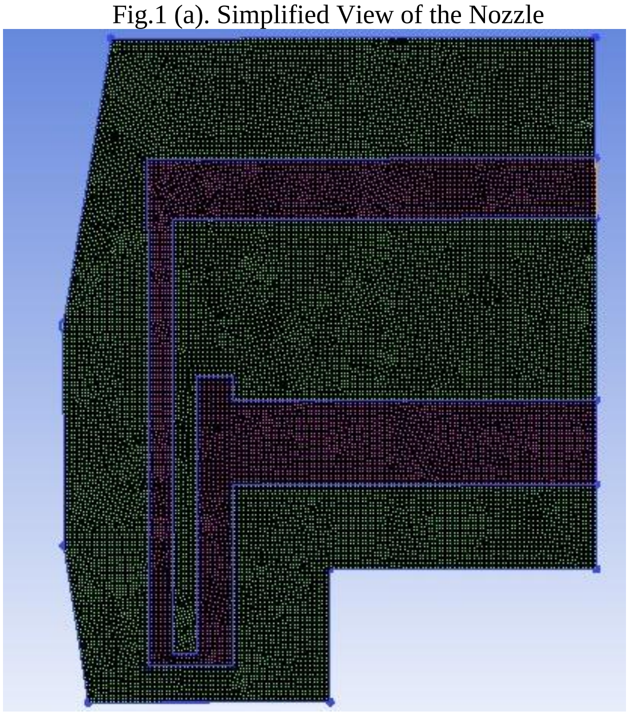

Fig.2 Simplified Model of the Nozzle with Meshing.

II. NOZZLE PARAMETERS

The nozzle under study has important parts that have to be considered,

a) Outer console which is made of copper
b) Inner console which is made of copper
c) The cooling channel, which carries coolant
d) Seal gasket which is at rear of the nozzle

The water is used in the coolant system. The water which is available to us freely was more efficient in the present study. The parameters like velocity, temperature and direction was changed in order to get higher cooling efficiency. The coolant is forced through the inlet with variable velocities and is forced out through the outlet. The front portion of the inner wall of the nozzle is in contact with the temperature source. The temperature of 8000 K is taken for the current study. The study deals with the variable velocities but with static temperature. But different coolants can be used in this study. For example, using liquid carbon-di-oxide (LCO2) is one of the ways to cool the system efficiently.

III. ASSUMPTIONS

The assumptions made in the present study are as follows:

1) The coolant is assumed to be water which is an incompressible fluid.
2) The model is simulated using k-Ɛ turbulence model.
3) The coolant flow is taken as continuous.
4) A 2D model is assumed due to the fact that the fluid field and the heat field are in symmetry.
5) SIMPLE method is used for the study.

IV. GOVERNING EQUATIONS

Continuity equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0. \quad ..(1)$$

Momentum Equation,

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) = -\nabla p + \mu\nabla^2\mathbf{v} + \mathbf{f}. \quad ..(2)$$

Turbulence Kinetic Energy equation,

$$\rho ui\,\frac{\partial \varepsilon}{\partial xj} = \frac{\partial}{\partial xj}\left|\frac{\mu eff}{\sigma k}\frac{\partial \varepsilon}{\partial xj}\right| + C1\frac{\varepsilon}{K}G - C2\rho\frac{\varepsilon 2}{K} \quad ..(3)$$

Boundary conditions

a) The inner wall of the nozzle is assumed no slip region.
b) According to the characteristic of the nozzle the temperature is assumed to fall on single spot of the nozzle.
c) The temperature is assumed to fall is 8000 K.
d) Inlet water is allowed to pass through it at 303 K.
e) Outlet pressure is set to gauge pressure.

V. THE SIMULATION WORK

The model is generated and meshed with the Gambit software. The mesh size is 0.358mm to 0.4mm for our convenience. The cooling channel experiences the pressure drop due to the change in the area of the system. Thus low pressure region is formed and local turbulence makes the water-static region to appear at the front corner of the cooling channel.

The water static regions decrease the efficiency of the cooling system and eases in the burn out of the nozzle and seal gasket.

Fig.3 shows us the water stagnant regions being formed at the front corner of the cooling channel and also for our convenience it is made of two-regions that are under study.

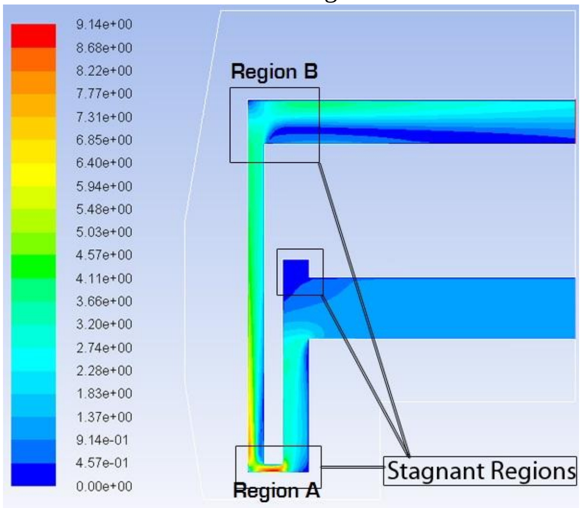

Fig.3 Velocity field and water-static regions
A detailed view of the region A is shown in Fig.4.

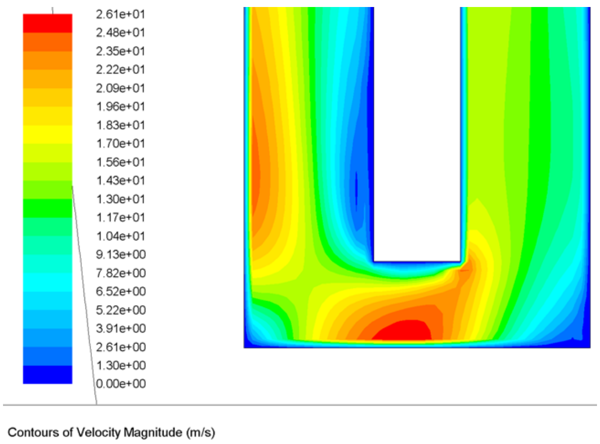

Fig.4 Detailed view of the stagnant water region.

The water is allowed to pass in to the cooling channel with the variable velocity. The velocities from 1 m/s to 5.5 m/s are used for the cooling system. The water at 303 K is used for the inlet of the coolant channel.

From Fig.4, one can envisage that the velocity water at the corner of the cooling channel is almost zero. This is water – static region. Water – static regions decrease the cooling efficiency of the system. Thus water should be allowed to pass through the corners effectively.

To make this water –static region to disappear one should optimize the structure of the nozzle. The geometrical optimization could be done by using round corners instead of sharp ones. The round corners allow the water to flow through the cooling channel easily without any change in pressure. Thus no static-water region,

Region A is focused here for the geometrical optimization. The close difference of the structural optimization can be seen in Fig.5.

a) cooling system with sharp corner.

b) cooling system with round corner.

Fig.5a shows us the path lines in sharp corners with water static region. In Fig.5b the path lines are almost uniform due to the disappearance of the water – static region.

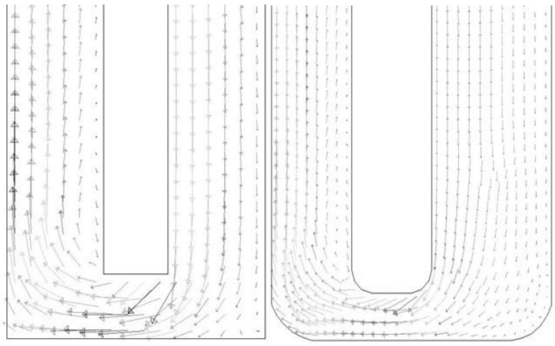

Fig.5 Difference in water flow in the corners of the cooling system of the nozzle.

VI. STRUCTURAL OPTIMIZATION

The water formation of the water – static region almost disappeared in the round corner of the nozzle. Thus changing the corners has given us substantial efficiency in the cooling system. The Fig.6 shows us the temperature distribution and heat exchange.

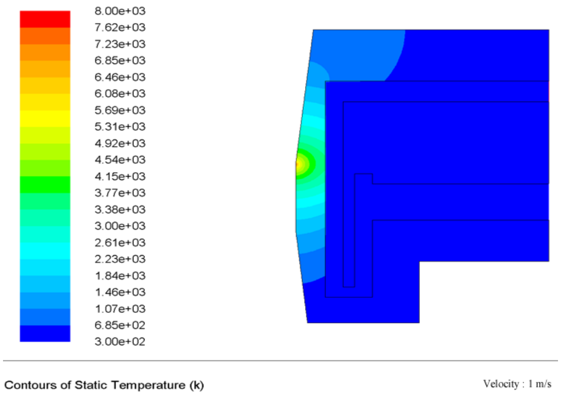

Fig.6 The structural optimization and fluid velocity of 1 m/s.

Fig.6 shows us the temperature being conducted to the outer console of the nozzle due to the fact of higher thermal conductivity of the nozzle. This high temperature over the outer console has to be cooled with water at variant velocities. Fig.6 shows us the velocity of water at 1 m/s. changing the water velocity from 1 m/s to 5.5 m/s has been conducted in this study. There is significant improvement in the cooling system due to the change of velocity of water. Fig.7 shows the water at 5.5 m/s and the temperature distribution over the nozzle.

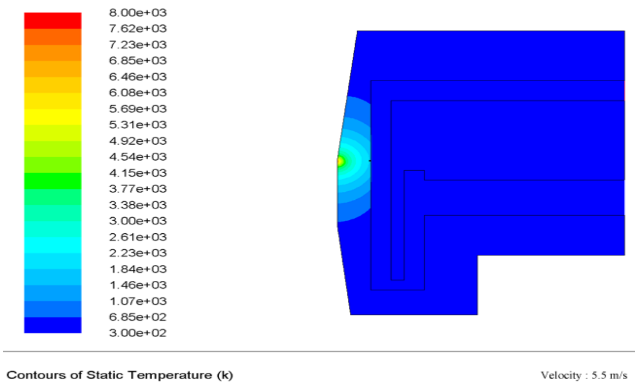

Fig.7 Temperature distribution with water flow velocity of 5 m/s.

VII. DIRECTION OF FLUID FLOW

The fluid flow direction is one of important parameter that enables us to provide better cooling to the seal gasket region situated at the rear of the nozzle. The water enters the rear of the nozzle first and then forced out through the front of the nozzle. Thus changing the temperature distribution of the nozzle and also saving the seal gasket from burn out.

Therefore, it is suggested to reverse the direction of flow of water from outlet to inlet which can cool the nozzle efficiently than the previous cooling system.

VIII. CONCLUSION

The model of plasma spray gun is numerically simulated using SIMPLE method. This method allows us to verify and analyze the fluid flow and heat transfer patterns in the nozzle. The formation of the local turbulence is mainly due to the change in area of the cooling channel. This can be overcome by changing the sharp corner to round corner at the front of the cooling system. The following conclusions can be drawn,

a) A suitable simulation technique has to be used for the study and this SIMPLE method allows us to simulate the 2D model with k-Ɛ model.
b) Water flow distribution of the nozzle is seen with varying velocities. Hence changing the velocity has yielded a significant cooling efficiency.
c) A water – static region is considered under study, namely region A. And is analyzed further for the change in geometry in this area. It can be seen that water – static region being disappeared in this region with round corners.
d) So it is suggested to use round corners for the higher water flow rate and thus higher cooling efficiency.
e) The location that was not able to cool effectively was the seal gasket region which later was cooled effectively by changing the water flow direction.

REFERENCES

[1] Kitaguchi S, Takeshima Y, Nishisaka T, et al. (1990) Japan Soc Powder PowderMetall 37:918–92

[2] Felix A. Muggli, Ronald J. Molz, Richard McCullough, and Dave HawleyImprovement of Plasma Gun Performance using Comprehensive Fluid ElementModeling: Part I

[3] Amalia R. Lopez, Basil Hassan, William L. Oberkampf, Richard A. Neiser, and Timothy J. RoemerComputational Fluid Dynamics Analysis of a Wire-Feed,High-Velocity Oxygen-Fuel (HVOF) Thermal Spray Torch

[4] Ronald J Molz, Richard J Mucculough , Sulzer Metco Better Performance of Plasma Thermal Spray.

[5] WeimingWang, Dayong Li, Jie Hu, Yinghong Peng, Yishun Zhang, Deyuan Li Numerical simulation of fluid flow and heat transfer in a plasma spray gun.

Raghunandan M S,
Assistant Professor,Dept. of Mechanical Engineering, GSSSIETW, Mysore - 570016

Master of Technology in Computational Analysis in Mechanical Science, VVCE, Mysore, 2012, Bachelor of Engineering in Mechanical Engineering, VVIET, Mysore, 2010. Received awards and certificates for achieving 100% results in Computer Aided Engineering Drawing and Elements of Mechanical Engineering Subject.