Simulation of 3D vortex jets in plasma torch application

W Kongpiboolkid[1,3*] **and R Mongkolnavin**[2,3]

[1] Department of General Education, Faculty of Science and Health Technology, Navamindradhiraj University, Bangkok, 10300, Thailand
[2] Department of Physics, Faculty of Science, Chulalongkorn University, Bangkok 10330, Thailand
[3] Plasma Technology and Nuclear Fusion Research Unit, Chulalongkorn University, Bangkok 10330, Thailand

*E-mail: watcharapon@nmu.ac.th

Abstract. The gas tunnel type plasma jet is an effective heat source for thermal processing applications such as plasma spraying. The key concept of gas tunnel plasma is its torch configuration, especially the role of the vortex gas flow. This is very important for the stability and energy density of the plasma jet produced. This work studied the flow of gas vortex in 3 dimensions using a finite element simulation. The simulation is based on solving partial differential equations where the incompressible Navier-Stokes equation is used as a governing equation that describes the laminar flow. The geometry of the plasma torch investigated is based on the design by A. Kobayashi. Key parameters investigated were gas pressure, velocity and profile of the vortex. It can be shown that the simulation produced results that are better matched to the experimental result than the calculation done in previous work. The simulation can also show detailed pictures of the vortex and its properties within the plasma chamber. This study could be useful in the design optimization of the plasma torch in the future.

1. Introduction

The gas tunnel type plasma jet generator has been proposed as a heat source suitable for any kinds of applications such as coating, synthesizing of functional material, welding and gasification of waste disposal. The generator produces a plasma jet that has high speed and high energy density. Currently, it has also gained interest in the field of biomedicine and environment where a more precise and lower energy plasma is needed. To be able to achieve this specification, the use of vortex flow is introduced to a typical gas tunnel type plasma generator. Figure 1 shows a conceptual design of a part of a gas-tunnel generator proposed by Arata and Kobayashi [1] that the gas is flowing through small cylindrical tubes tangentially placed around the chamber to produce a vortex flow in the chamber. For plasma production purposes, a gas tunnel provides several advantages. Ionization of the gas is very easy, due to the low-pressure region in the gas tunnel. Moreover, a "gas wall" with a sharp pressure gradient in the radial direction effectively stabilizes the plasma beam.

The key concept of the gas tunnel plasma jet generator is its torch configuration and the role of the vortex gas flow. These factors affect the stability and energy density of the plasma jet produced [1-3]. In previous studies of vortex flow, it was found that the pressure in the middle of the vortex is lower than the outer edge of the vortex [4-7]. The research also investigated a model of the vortex flow based on a model of water draining down the sink. This is commonly known as the free vortex flow model as shown by figure 2. The model assumes that the gas is drawn from the circumference of the cylindrical chamber into a sink at the centre axis and a radial velocity is produced. The equation of motion is expressed as follow;

$$\frac{d\rho V_r}{dt} = \frac{dp}{dr} - \frac{V_\theta^2}{r} \tag{1}$$

where p is the pressure, r is the radius, ρ is the density of the gas, V_θ is the velocity in the tangential direction, V_r is the velocity in the radial direction and t is time. By further applying the continuity equation of and the law of conservation of angular momentum to the first equation, the velocity and the pressure distribution can be solved analytically from equation (2) as follow;

$$\frac{(p/p_w)^2 - V_{w*}^2}{1 - V_{w*}^2} = \exp\{-\frac{V_{\theta w}^2}{RT}[(\frac{r_w}{r})^2 - 1]\} \tag{2}$$

where p_w is the pressure at the chamber wall and r_w radius of the chamber wall, $V_{w*}(V_{w*} = V_{rw} / V_{\theta w})$ is the ratio of the two velocities at the wall, with $V_{\theta w}$ in the tangential direction and V_{rw} in the radial direction, R is the gas constant and T is the temperature.

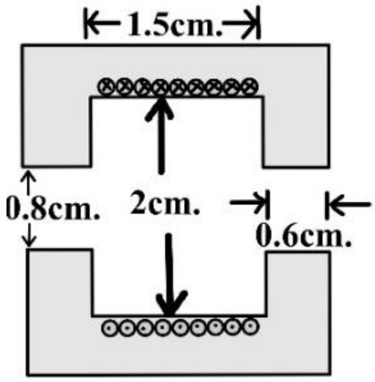


Figure 1. Conceptual diagram of a gas tunnel generator.

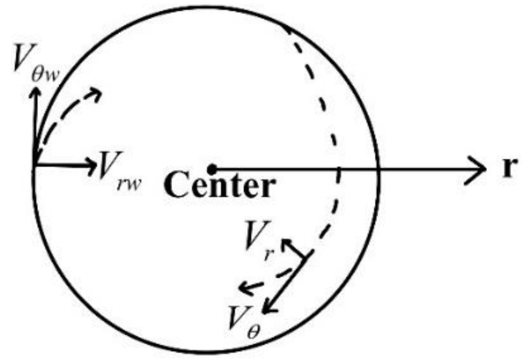
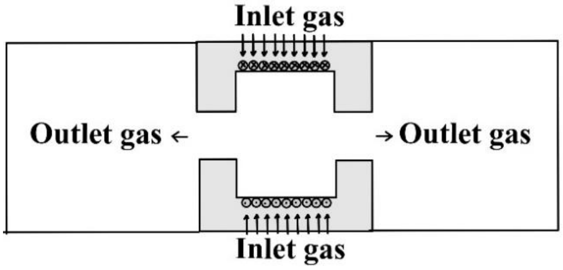


Figure 2. Vortex flow model.

However, a simple mechanical fluid flow theory cannot represent the whole picture of the vortex produced by a more complex system. The calculated tangential velocity and the calculated radial velocity do not match the experimental measurement. This study is, therefore, looking at simulating the vortex in 3 dimensions using a finite element method to look at a pressure profile and velocities that are a better match to the experimental measurement.

2. Simulation

The finite element method (FEM) is used extensively for numerically solving differential equations arising in any mathematical or engineering model. It is commonly used for analysing physical properties such as structural, heat transfer, fluid flow, mass transport, and electromagnetic potential [8-9]. In this research, the vortex flow was simulated using FEM software (COMSOL) [10]. The schematic of the vortex cavity investigated is shown in figure 3. The inlet gas nozzles are in the tangential direction on the cavity wall and evenly distributed every 90° apart. The cavity has an opening at both ends that allow the air to flow out freely to space with an ambient pressure of 1 atm. The software solves appropriate partial differential equations of discrete elements defined by a mesh of a 3D geometry as shown in figure 4.



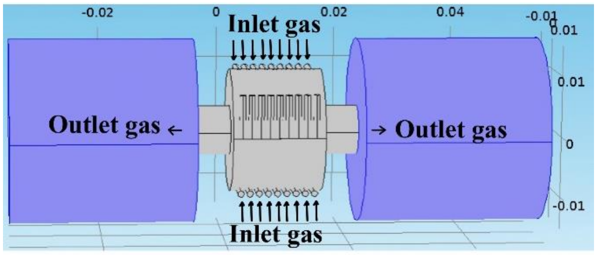


Figure 3. Schematic showing vortex nozzles of a part of the gas tunnel generator.

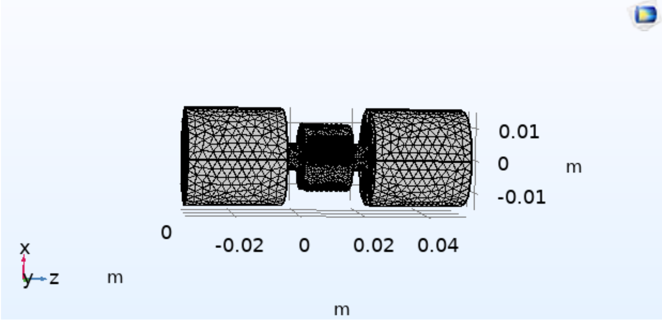


Figure 4. Schematic of a mesh used for FEM simulation.

The gas flow simulation was based on a laminar flow model where the flow begins at the inlet gas nozzles. Various gas flow rates were simulated. The compressible Navier-Stokes equation was used as the governing equation that describes the flow of air in the vortex cavity, as shown in equation (3) as well as the continuity equation as shown in equation (4).

$$\rho\frac{\partial u}{\partial t}+\rho(u\cdot\nabla)u=\nabla\cdot[-p\mathrm{I}+\mu(\nabla u)+(\nabla u)^{\mathrm{T}}]+F \tag{3}$$

$$\frac{\partial \rho}{\partial t}+\rho\nabla\cdot(u)=0 \tag{4}$$

where ρ is the density (kg/m^3), u is the velocity (m/s), p is pressure (Pa), I is an identity matrix, T is the transpose of a matrix, μ is the viscosity (kg/m.s) and F is the volume force vector (N/m^3).

3. Result and discussion

From the simulation, the relationship of relative tangential velocity and radial direction distance is shown in figure 5 (a). Even though the simulation result resembles the result obtained by the experiment done by Arata and Kobayashi [1], it does not totally agree with the analytical model proposed by them. Figure 5 (b) shows the relative radial velocity with radial direction distance. Again, a similar discrepancy between the simulation and calculation in previous research in the central region can be seen. However, both experimental results done by Arata and Kobayashi [1] and the simulation show good agreement where the radial velocity reduces towards the centre of the vortex. By simulating the vortex in 3D, the velocity in the axial direction could be deduced. Figure 5 (c) shows a variation of relative axial velocity of gas flow with the radial position. The result shows that the axial velocity has its direction out of the tunnel along the centre region. To complete the result, figure 5 (d) shows the comparison plot of relative pressure at different radial positions. The results show a similar trend between the simulation and the experimental results obtained by Arata and Kobayashi [1]. The lowest pressure can be found in the middle and on the axis of symmetry. However, the pressure found in the simulation is much higher than that of the experiment which can be explained possibly by the difference in the actual experimental setup used by Arata and Kobayashi [1]. Their experimental setup has gas diverter valves at the opening which allow the excess gas to exit freely.

The effect seen at the outer central region can be explained by a free vortex flow which is the model that assumes the fluid is drawn to a sink. This is the model that was assumed by the previous research done by Arata and Kobayashi. However, in reality, the actual vortex is produced by injecting air tangentially which is more consistent with the forced vortex flow model. Therefore, to accurately simulate the vortex that was produced in the experimental set-up, it is essential to consider both effects of the sink and the high-speed airflow. This is why the result from the FEM simulation has a better agreement with the experimental result than the theoretical model assumed before. The result also agrees with Rankine (1858) model explained by Azarpira and Zarrati [7] where he assumed that the vortex is composed of two parts of a forced vortex and a free vortex zone where the turning point defines the boundary of the two zones. In this case, the turning point is at 0.4 cm which corresponds to the radius of the opening of 0.4 cm that was in the simulation and the experiment.

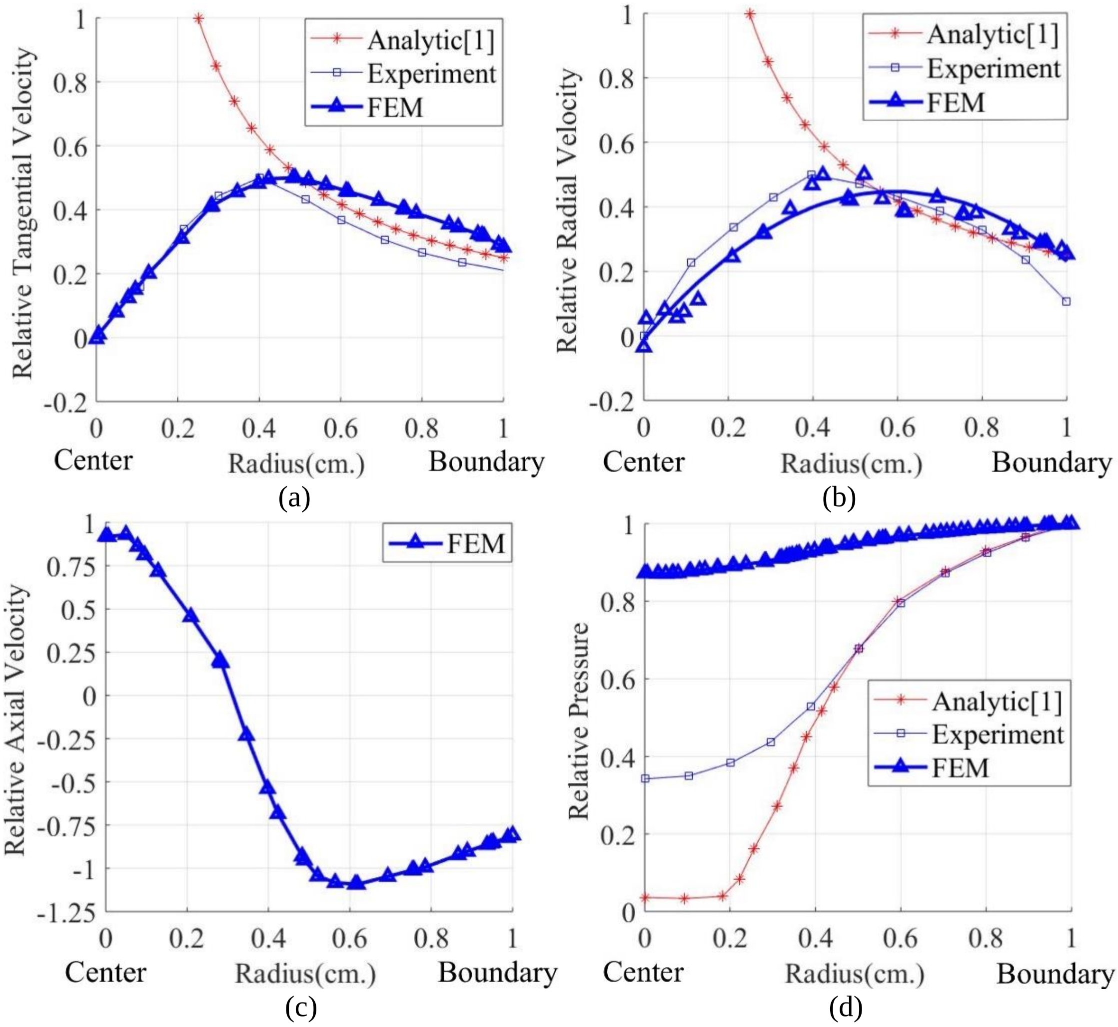


Figure 5. Plot showing variation (a) relative tangential velocity (b) relative radial velocity (c) relative axial velocity and (d) relative pressure with the radial position.

Additionally, the simulation results can also show a heatmap of the pressure profile of the vortex flow in the gas tunnel generator. Figure 6 shows the 3D profile of the velocity. It shows the gas flows through the gas tunnel generator and the direction in which the gas travels with vortex motion. The velocity inside the gas tunnel generator is higher in the cavity than that on the centre axis region.

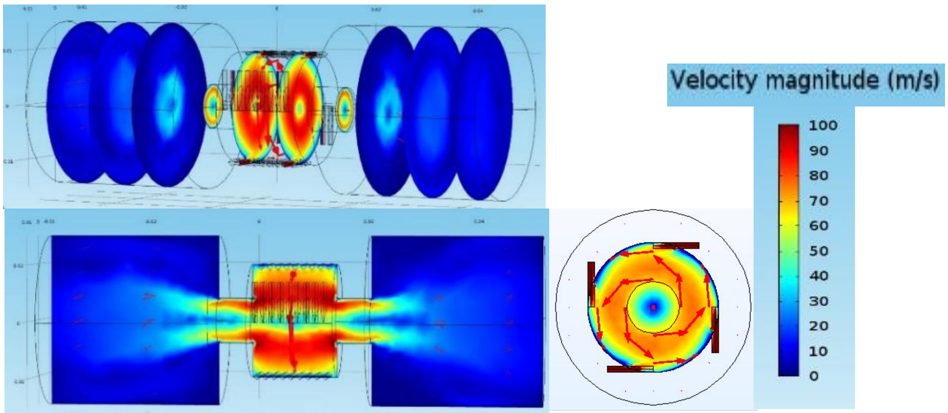


Figure 6. Heat map of velocity from simulation of laminar flow through the gas tunnel generator.

Furthering the investigation by varying the gas inlet flow rate (Q in litre/minute), the tangential velocity profile, as well as the pressure profile, can be shown in figure 7 and figure 8 respectively. It can be seen that the vortex tangential velocity increased as the gas inlet flow rate was increased. This is expected as the inlet flow rate Q relates to the initial pressure in equation (3) according to Poiseuille's Law. The pressure envelope due to the pressure gradient between the centre and the outer region inside the cavity became more distinct when the gas flow increased. This is due to the increasing difference between the pressure at the edge and the middle. As a result, it is possible to design and optimize the gas inlet flow rate and the dimensions of the plasma generator with the information obtained from the simulation. A suitable geometry and gas flow rate can be determined such that it produces the required low-pressure cavity for lower power input and a more stable plasma production.

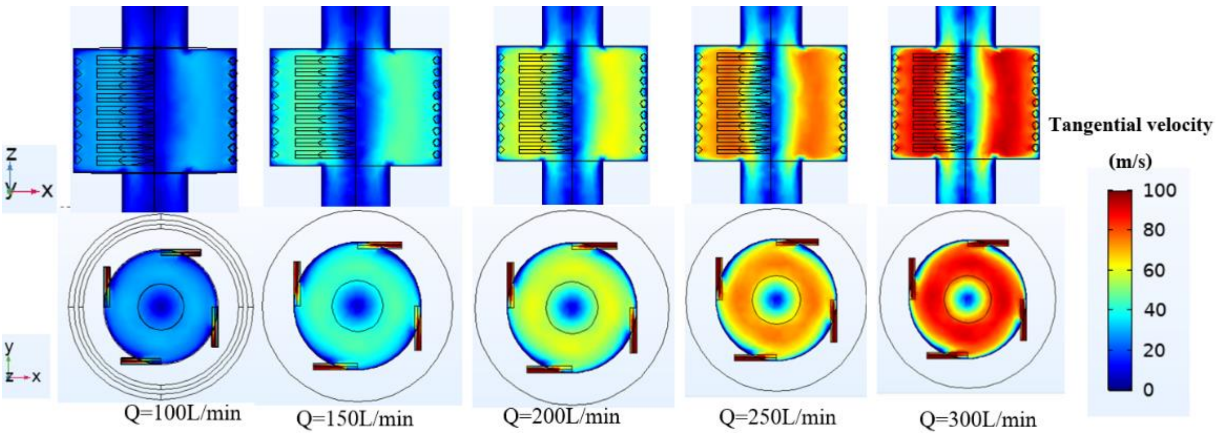


Figure 7. Tangential velocity simulation of laminar flow through a gas-tunnel generator.

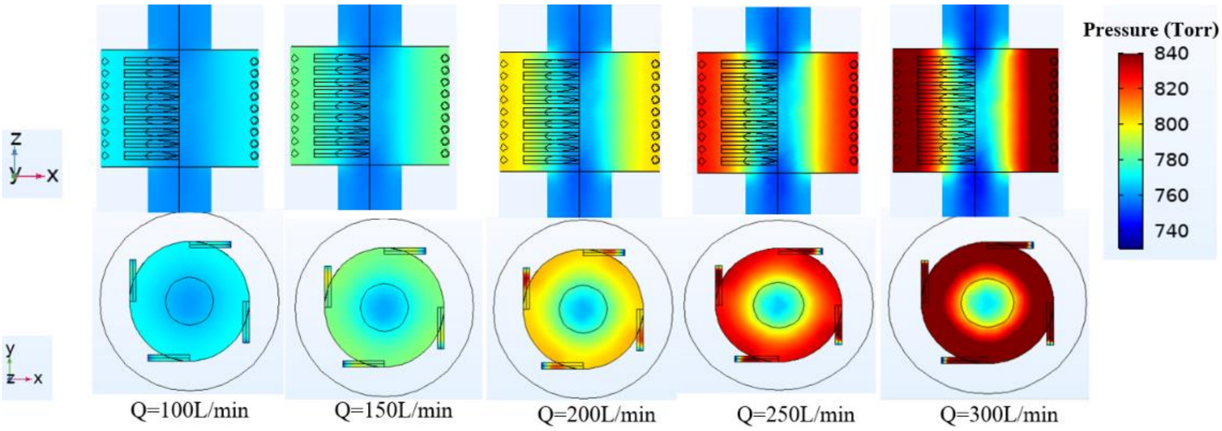


Figure 8. Pressure simulation of laminar flow through a gas-tunnel generator.

4. Conclusion

A vortex flow based on a gas tunnel type plasma jet generator designed by Arata and Kobayashi [1] has been simulated using FEM software. The compressible Navier-Stokes equation was used as the governing equation for FEM to describe the airflow in the vortex cavity. Various gas flow rates ranging from 100 L/m to 300 L/m were simulated. It was found that the velocity inside the gas tunnel generator was higher within the cavity than in the centre region. The result from the FEM simulation has a better agreement with the experimental result than the theoretical model assumed before. The discrepancy can be explained by the combined effect of the sink and the high-speed airflow that should be taken into account. With the good agreement of the result with the experiment, the simulation was extended to show a detailed picture of the vortex and its properties within the chamber. It was also found that the pressure envelope of the vortex inside the cavity became more distinct when the gas flow was increased. This study could be useful for designing and optimizing the gas tunnel type plasma torch in the future.

Acknowledgements

The authors would like to extend their great appreciation to Prof. Akira Kobayashi for his guidance and Navamindradhiraj University and Plasma Technology and Nuclear Fusion Research Unit, Chulalongkorn University for their full support of the work done in this research.

References

[1] Arata Y and Kobayashi A 1986 *J. Appl. Phys.* **59** 3038–44
[2] Arata Y, Kobayashi A, Habara Y and Jing S 1984 *Trans. JWRI* **15** 227–31
[3] Szałatkiewicz J 2014 *Pol. J. Environ. Stud*. **23** 277–81
[4] Burgers J M 1948 *A Mathematical Model Illustrating the Theory of Turbulence Advances in Applied Mechanics* vol **1** eds R Von Mises and T Von Karman (New York: Academic Press) pp 171–99
[5] Vatistas G H 1998 *J. Propuls. Power* **4** 462–9
[6] Vatistas G H 2006 *J. Aircr.* **5** 1577–9
[7] Azarpira M and Zarrati AR 2019 *Water Sci. Eng*. **12** 244–52
[8] Akter F and Deb U K 2017 *Am. J. Comput. Math.* **7** 58–69
[9] Bottani E, Rizzo R and Vignali G 2007 *Chem. Prod. Process Model.* **2** 58–69
[10] COMSOL 1998–2016 *COMSOL Multiphysics Reference Manual (Version COMSOL 5.2a)* (Sweden, Stockholm: COMSOL Inc.)