Solid Freeform Fabrication 2018: Proceedings of the 29th Annual International
Solid Freeform Fabrication Symposium – An Additive Manufacturing Conference

OPTIMIZATION OF INERT GAS FLOW INSIDE LASER POWDER BED FUSION CHAMBER WITH COMPUTATIONAL FLUID DYNAMICS

Yu Chen[1, a], Guglielmo Vastola[1,b] and Yong Wei Zhang[1,c]

[1]Institute of High Performance Computing, 1 Fusionopolis Way, #16-16 Connexis North Tower, Singapore 138632

[a]chenyu@ihpc.a-star.edu.sg, [b]vastolag@ihpc.a-star.edu.sg, [c]zhangyw@ihpc.a-star.edu.sg

Abstract

It is crucial to maintain a uniform and fast enough inert gas flow inside build chamber to obtain high-quality final products (e.g. low porosity) without oxidation. The current study investigated the behaviors of the inert gas flow inside a chamber with CFD simulations, as well as its evaluation and optimization. The gas flow pattern inside the chamber was evaluated in terms of the uniformity of velocity across the build plate. It was shown that the gas channels and locations of inlet openings significantly affected the flow inside the chamber. So the design of gas channels/inlets and flow rates was carefully adjusted to generate uniform gas flow across the chamber to remove emissions from the melt pool efficiently. Furthermore, the re-circulation of emission inside chamber was significantly reduced to keep the chamber walls clean and minimize the damage to the optical surface. In conclusion, CFD benefits in improving quality of products and reducing life-cycle cost for laser powder-bed fusion process (L-PBF).

Introduction

Selective laser melting process is becoming one important method of powder-bed-fusion based additive manufacturing technologies. However, there are still a lot of unsolved problems within the process, which result in imperfectness in final printed products, such as void, pore and residual stress. So it is essential to predict and avoid these unexpected problems during printing. Inert gas flow over powder bed is one of the techniques ensuring the quality of the final products and efficiency of the printing process. On one hand, the gas protects the metal from oxidation at high temperature during melting under laser. Secondly, the flowing gas carries particles spattering from melt pool away from the chamber, avoiding second deposition on the powder bed and blockage of laser. Thirdly, the emission and metal vapor from melt pools can be carried away by gas flow instead of staying inside chambers then cool down and stick on chamber walls and optical surfaces.

Ferrar et al. (2012) found that poor gas flow induces higher porosity of printed material, resulting in lower compressing strength. Anwar & Pham (2016, 2017) investigated the influence of gas flow velocity and laser scanning direction on tensile strength of part, which is affected by the removal and motions of spattering particles. Ladewig et al. (2016) investigated the mechanism how the by-products of laser melting (spatters and welding plume) induce the lack of fusion and finally the defects of part. As the gas flow is important, Philo et al. (2015, 2017) utilized CFD to numerically simulate the gas flow inside RENISHAW AM250 machine, and compared with experimental results. They showed the capability of CFD in prediction and evaluation of gas flow during design phase. Besides, they also showed the sensitivity of flow uniformity to some design parameters, such gas inlet rail diameter, nozzles' shape etc. However, there is a lack of explanation of underlying mechanism for these phenomenon and design guidelines to obtain better gas flow. So, it motivated us to study the gas flow inside the RENISHAW AM250 chamber and underlying fluid dynamics, and to summarize design guidelines through step by step optimization of gas channels inside chamber.

Method and Simulation Setup

A virtual RENISHAW AM250 chamber and its gas inlet rail and outlet were built up in ANSYS for simulation and evaluation of the inside gas flow (see Figure 1). The gas firstly enters a 550mm long inlet rail with diameter 40mm, then turns into an array of 13 cylindrical nozzles with 12mm in diameter and 18mm in length, of which the interval is 20mm. The center of the nozzle array is located 67.5mm above the bottom wall of the chamber, where the 271mm*271mm sized build plate is located.

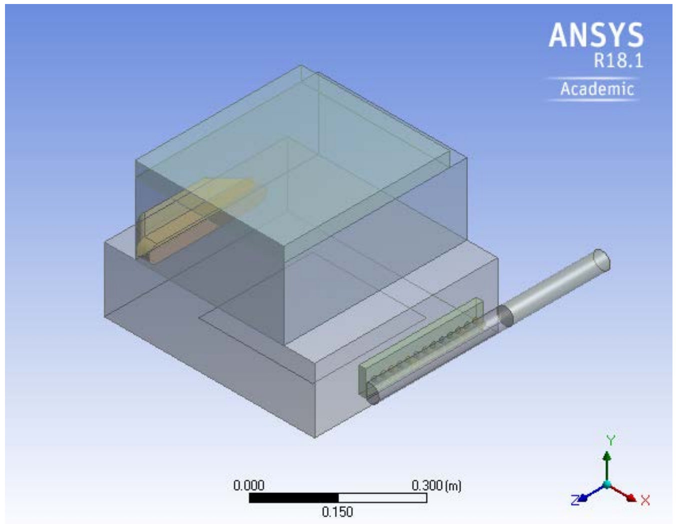


Figure 1, Sketch of the RENISHAW AM250 chamber

CFD simulation of gas flow inside the chamber was conducted using ANSYS Fluent software with steady, incompressible, turbulent flow assumption. Standard Navier-Stokes equations and k-e turbulent model were chosen to investigate the gas flow inside print chamber. The inlet was defined as “mass flow rate” of 0.0051kg/s with 2% turbulence intensity was set at the inlet, while “outflow” boundary condition for the outlet. So the velocity of gas coming out from the nozzles is around 5m/s. No slip boundary condition with standard wall function was utilized on other walls of the chamber.

The computational domain was meshed based on Tetrahedron and Hexadedron dominated cells, while 10 inflation layers was added on the surface of bottom wall surface are to capture boundary layer flow precisely. Before examining the gas flow, grid number independence study was conducted to choose the proper mesh resolution for the current study and verify the accuracy of the method. Since the gas flow over the build plate is the main concern, the mean, maximum and minimum velocity and variance of velocity on the plane 3mm above the build plate chosen as criteria for convergence of results. As shown in Figure 2, the value of velocity reaches a good asymptotic status at 3.5 million cells, which was chosen for the rest simulations.

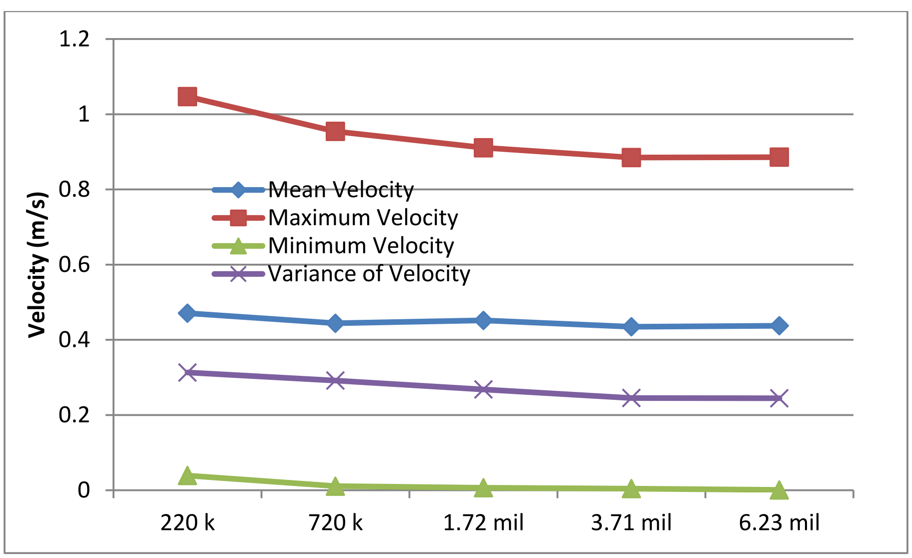
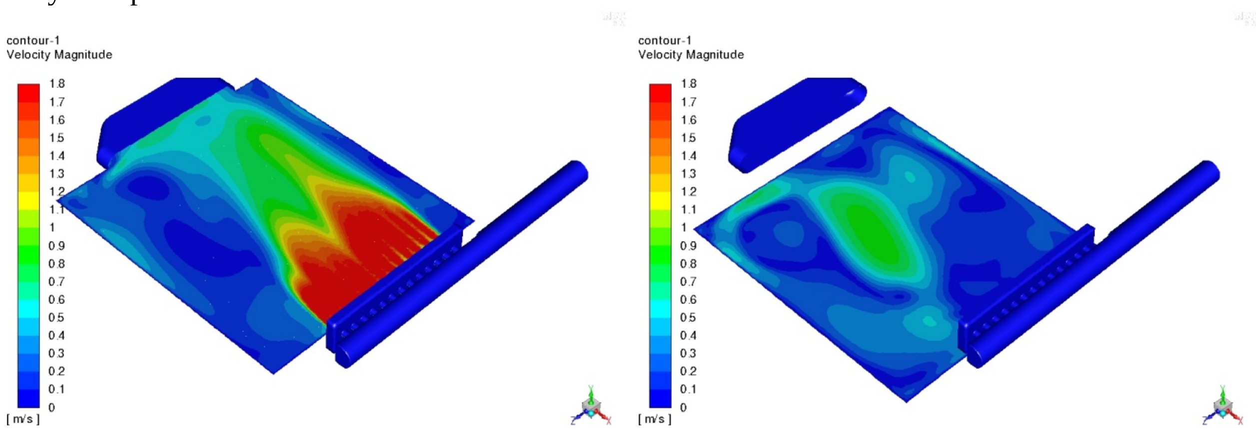


Figure 2, Grid independence study for velocity

Results and Discussion

Original Design

Before the possible optimizations of gas flow, the gas flow inside the chamber with the original design parameter shall be examined first. Velocity contours on two planes over the bottom wall are presented in Figure 3. It can be seen from the velocity contour on the plane through the inlet nozzles' center that the jet flow is uniform among the array of nozzles, especially the jet sweep down at the far end of the inlet rail. So the gas flow is strongly non-uniform on the plane 3mm above the chamber's bottom with a high speed region, as the jet partially sweeps down to the bottom at the far end of inlet rail.



a) b)

Figure 3, Velocity contour on the plane through the inlet nozzle's center a) and the plane 3mm above the bottom of chamber b) for the original design

To check the influence of gas flow on selective laser melting carefully, it is worth to examine the velocity contour over the build plate closely with other parts but off as shown in Figure 4. The high speed spot region can be clearly found located over the left bottom corner of the build plate, leaving other regions without coverage of gas flow. To reveal the underlying mechanism behind the non-uniform velocity of gas flow, a side view of gas flow inside the chamber is demonstrated in Figure 5. It is found that the nozzle is too high compared to the chamber's bottom, so that jet is too far away from the build plate and becomes unstable and partially sweeps down to form high speed spot over the build plate. So reducing the height of nozzles in the original design is supposed to increase the gas flow quality over the quality, which is investigated in the next part.

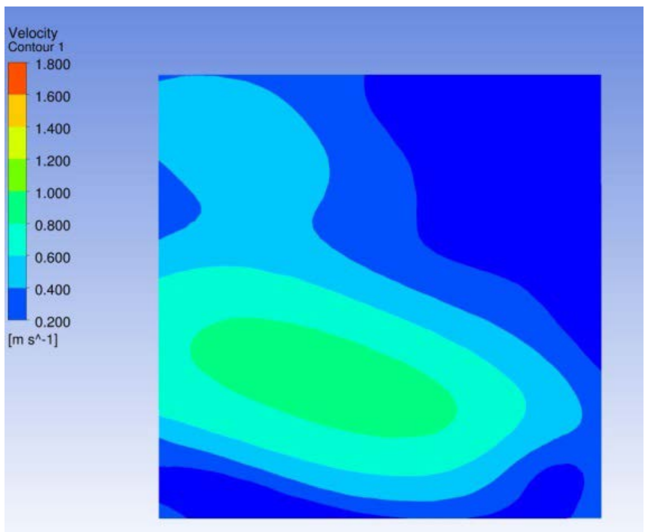


Figure 4, Velocity contour on the plane 3mm above the build plate for the original design

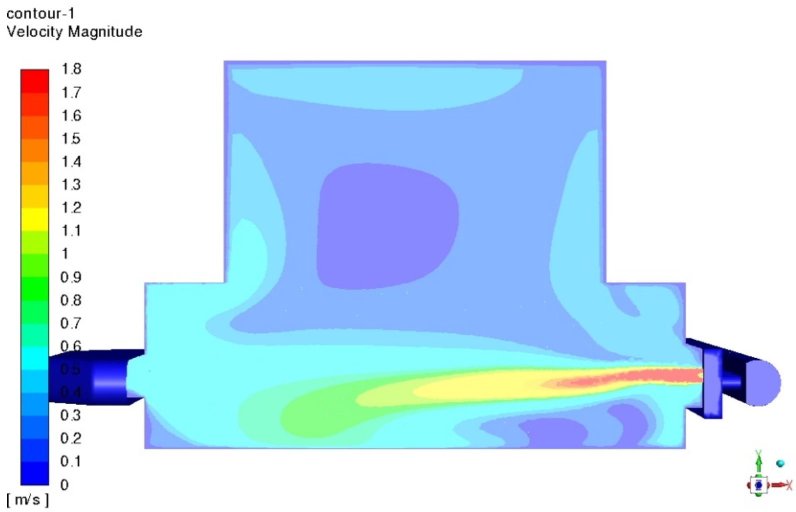


Figure 5, Side view of velocity contour of jets for the original design

Reducing Nozzle height and elongated nozzle length

It can be found from Figure 6 that after reducing the height of nozzles, the gas flow velocity over the build plate becomes much more uniform. However, there is still non-uniformity in the region near the nozzles since the nozzle is not long enough for gas to turn direction completely after going through the inlet rail. So, longer nozzles are believed to obtain jet with more unique directions, which is confirmed by the visualization of velocity contour presented in Figure 7.

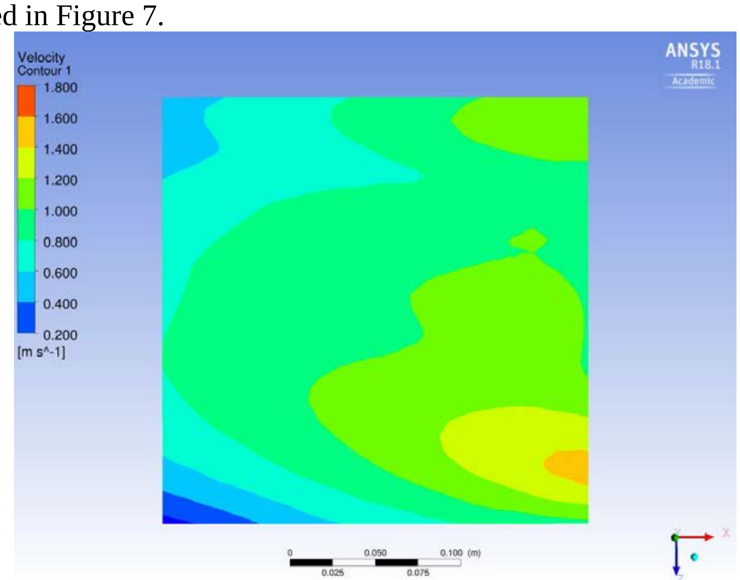


Figure 6, Velocity contour on the plane 3mm above the build plate for the design with reduced nozzle height

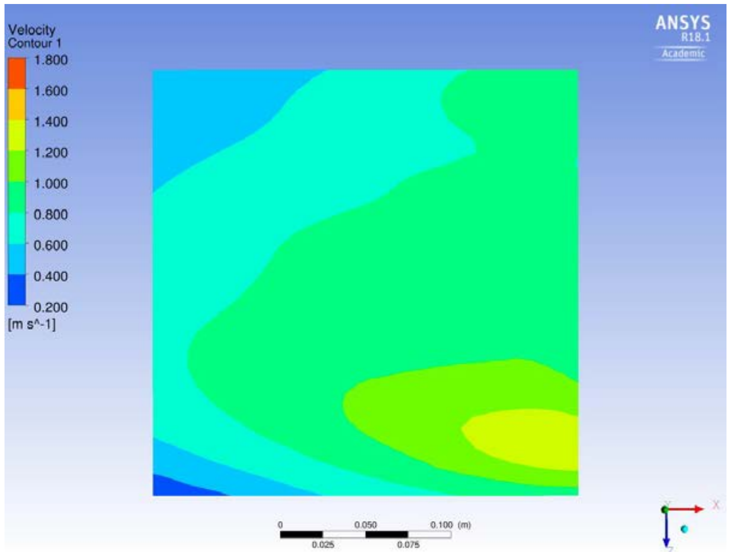


Figure 6, Velocity contour on the plane 3mm above the build plate for the design with reduced nozzle height and elongated length

However, there is still a high speed spot at the right bottom corner of the build plate. Through the examination of the full picture of the velocity contour on the plane 3mm over the chamber's wall in Figure 7, it is demonstrated that although the nozzle is close to the chamber bottom the jet still sweeps down to the bottom especially at the two ends of the nozzle array. So expanding the width of nozzle array shall be effective to extend the sweeping down jet outside of the build plate, resulting in more uniform gas flow velocity over the build plate.

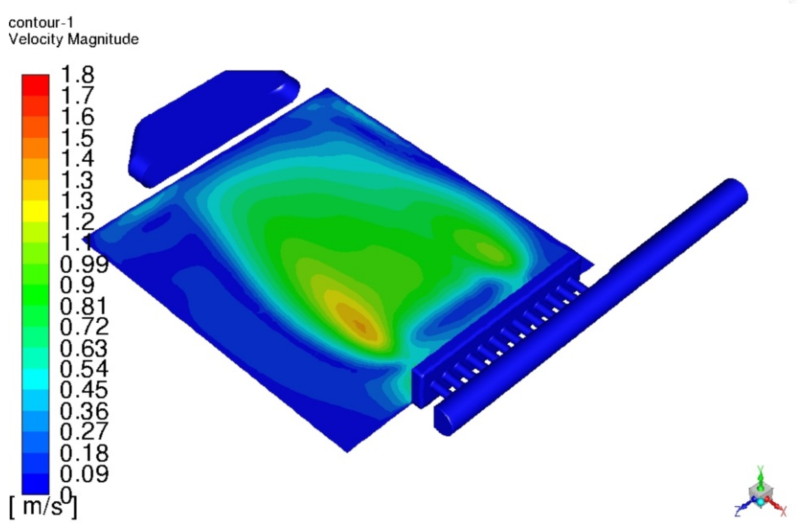


Figure 7, Velocity contour on the plane 3mm above the bottom and sweeping down jet at the ends of nozzle array

Extension of nozzle array width

After extending the width of nozzle array and increasing the number of nozzles from 13 up to 15, the sweeping down jet on the plane 3mm above the chamber's bottom close to the end nozzles is shifted outwards. So, the velocity of gas flow over the build plated shown in Figure 9 becomes much more uniform.

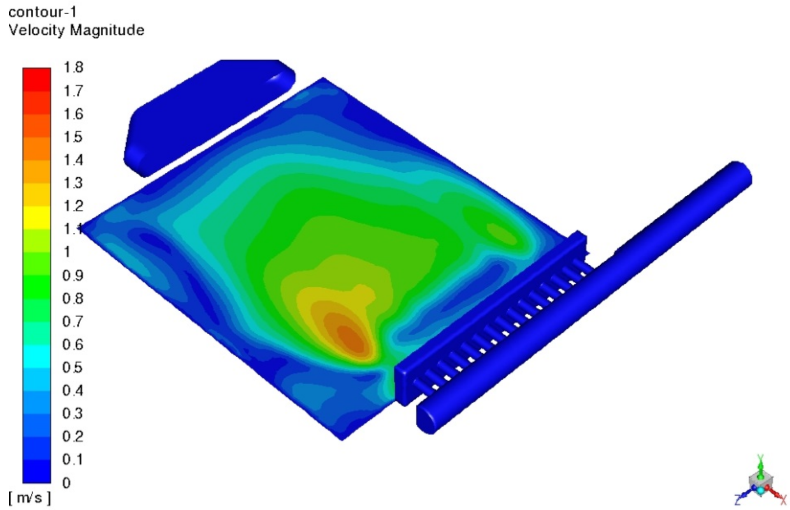


Figure 8, Velocity contour on the plane 3mm above the bottom for the extended nozzle array design

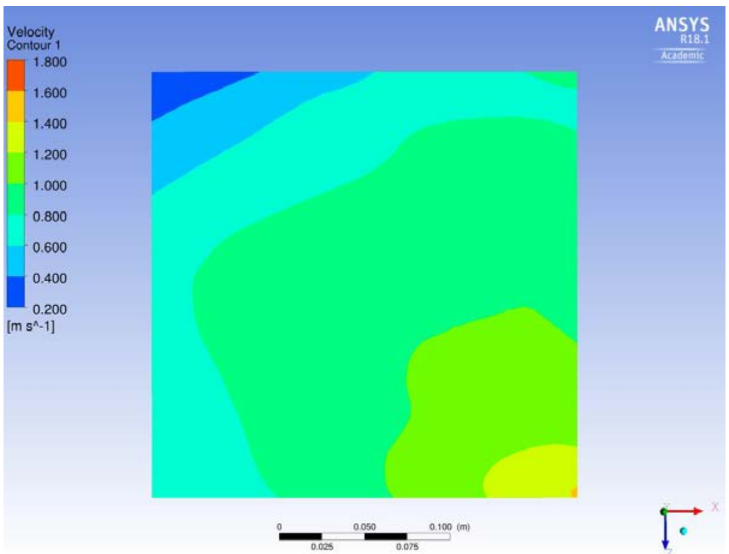


Figure 9, Velocity contour on the plane 3mm above the build plate for the design for the design with extended nozzle array

Secondary flow effect

Although the flow over the build plate has been optimized to very uniform pattern, the recirculation also needs to be investigated to avoid emission to stay and stick on the chamber walls and lens surface. As seen from the side view of the velocity in Figure 11, the recirculation is strongly suppressed by the downward secondary gas from the top wall which pumps in as 2/3 of the mas flow rate of the main inlet through the nozzles. Figure 12 demonstrating the path lines of gas flow from main and secondary inlets shows that the secondary flow comes down and join together with the horizontal main jet stream, thus suppressing the recirculation of main jet inside the chamber effectively.

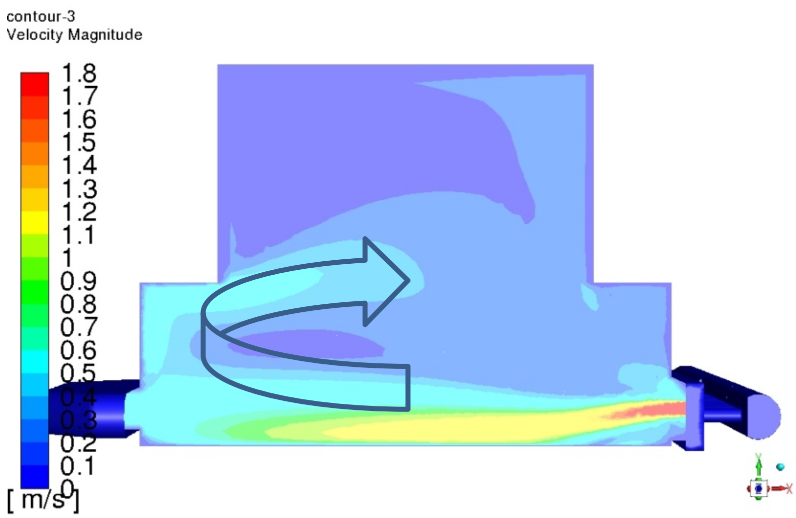


Figure 10, Side view of velocity contour and recirculation for the optimized design

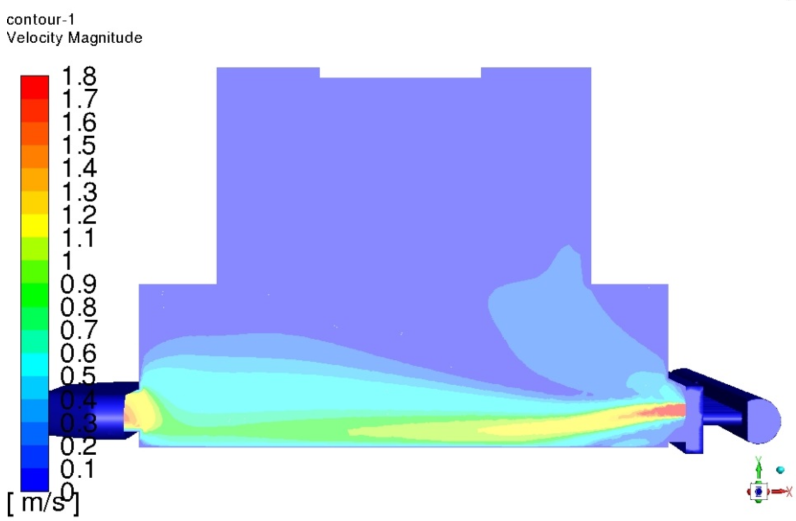


Figure 11, Side view of velocity contour for the optimized design with downward secondary flow

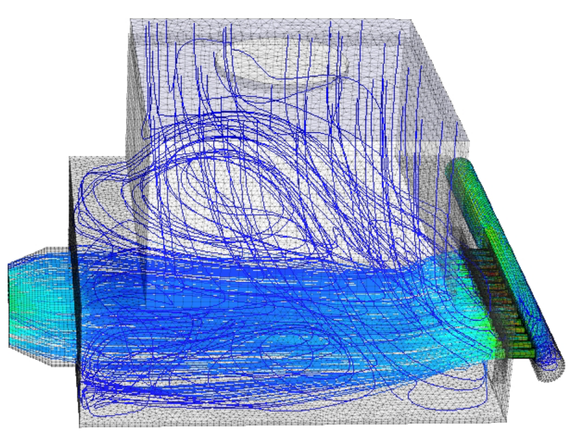


Figure 12, Path lines of gas flow for the optimized design with downward secondary flow

Summary

Through the discussion and visualization of gas flow in the above sections, it can be concluded that the flow becomes more uniform with the optimizations step by step. However, it is still worthwhile to evaluate the uniformity quantitatively based on mean, maximum and minimum velocity and variance of velocity. It can be found that through the optimization process (lower the nozzles, elongate nozzles and extend nozzle array/number), the variance of velocity is reduces, and the minimum velocity increases and gets closer to the maximum velocity, resulting in gas flow with higher quality. Furthermore, involvement of secondary downward flow avoids the disturbance of the uniformity while reducing the recirculation.

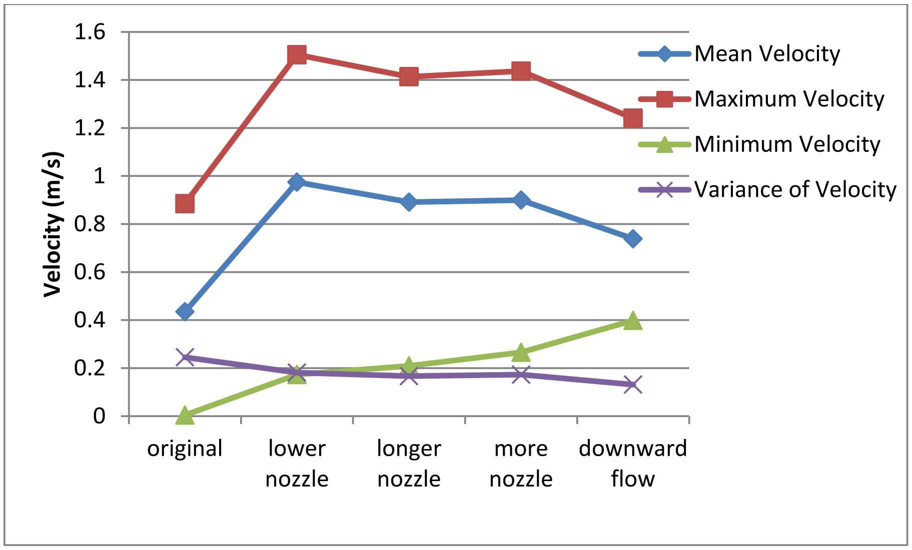


Figure 12, Evaluation of uniformity of flow velocity for design optimization step by step

Conclusions

In this study, CFD tool was implemented to optimize the gas flow inside RENISHAW AM250 chamber, which is believed to play very important role in the quality of final products. Through the step by step optimization, the uniformity of gas flow velocity was enhanced. It was also shown by the analysis that the instability of jets from nozzles affects the uniformity of gas flow velocity significantly. Several design guidelines were suggested, such as nozzles shall be lower and wider relative to the build plate, and the involvement of downward flow is essential to avoid the recirculation of welding plume inside chamber.

As future work, the proposed guidelines shall be verified for other chamber designs besides RENISHAW AM250, and more parameters can be tested the design inlet rail/manifold.

References

1. Ferrar, B., Mullen, L., Jones, E., Stamp, R., & Sutcliffe, C. J. (2012). Gas flow effects on selective laser melting (SLM) manufacturing performance. Journal of Materials Processing Technology, 212(2), 355-364.
2. Anwar, A. B., & Pham, Q. C. (2016). Effect of inert gas flow velocity and unidirectional scanning on the formation and accumulation of spattered powder during selective laser melting. Proceedings of the 2nd International Conference on Progress in Additive Manufacturing (Pro-AM 2016), 531-536.
3. Ladewig, A., Schlick, G., Fisser, M., Schulze, V., & Glatzel, U. (2016). Influence of the shielding gas flow on the removal of process by-products in the selective laser melting process. Additive Manufacturing, 10, 1-9.
4. Anwar, A. B., & Pham, Q. C. (2017). Selective laser melting of AlSi10Mg: Effects of scan direction, part placement and inert gas flow velocity on tensile strength. Journal of Materials Processing Technology, 240, 388-396.

5. Philo[1], A., Lavery[1], N. P., Brown[1], S. G. R., Cherry[1], J., Sienz[1], J., Joannou, J., & Sutcliffe, C. J. (2015). Comparison and Validation of Gas Flow Models in a Powder Bed Selective Laser Melting Process. Proceedings of the 23rd UK Conference of the Association for Computational Mechanics in Engineering 8 – 10 April 2015, Swansea University, Swansea
6. Philo, A. M., Sutcliffe, C. J., Sillars, S., Sienz, J., Brown, S. G. R., & Lavery, N. P. (2017). A Study into the Effects of Gas Flow Inlet Design of The Renishaw AM250 Laser Powder Bed Fusion Machine Using Computational Modelling. Solid Free. Fabr., Proceedings of the 28th Annual International Solid Freeform Fabrication Symposium, pp. 1203