# Heat Transfer Characteristics of Additively Manufactured Surfaces: An Experimental and Computational Study

**Kuldeep Mandloi[1], Chris Evans[1,2], Jason Fox[2], Harish Cherukuri[1], Jimmie Miller[1], Angela Allen[1], Jeff Raquet[1], and Brian Dutterer[1]**
**[1]University of North Carolina at Charlotte, USA**
**[2]National Institute of Standards and Technology, Gaithersburg, USA**

## INTRODUCTION

For adoption of additive manufacturing (AM) for parts requiring complex cooling channels, an understanding of the relationship between the as-built surface finish and heat transfer must be developed. In laser powder bed fusion (LPBF), there are numerous build parameters (e.g., part orientation during the build) that affect the final part surface topography and hence heat transfer [1].

To understand the relationship between topographies of real AM surfaces and heat transfer, surfaces with different orientations were fabricated using LPBF and analyzed [3]. Here we focus on the orientation of weld tracks relative to flow direction.This orientation can be realized in the AM parts built through adjustments to part orientation within the build chamber.

Simplified surfaces were developed for use in computational fluid dynamics (CFD) simulations and were based on measured surface topographies from prior work by the authors [1]. The simplifications were necessary to reduce the computational overhead and meshing issues associated with the modeling of real surfaces (i.e., surfaces with a wide range of spatial wavelengths present). Also, in order to better model AM surfaces, a unique approach to treating particle/spatter (e.g., ejecta from the melt pool, partially melted powder particles, etc.) deposits and their distribution has been adopted in this study. This framework was used to analyze heat transfer in terms of Nusselt number (Nu), pressure drop (ΔP) and performance factor (PF) while isolating the effects of track orientation and attached particles/spatter. The performance factor is defined as:

$$PF = 0.01 \cdot \frac{Nu}{f}$$

where

$$f = \frac{2 \, . \, \Delta P \, . \, D}{\rho \, . \, L \, . \, V^2}$$

Nu = Nusselt number.
$f$ = Friction factor.
$\Delta$P = Pressure drop.
D = Hydraulic diameter.
L = Length of the channel.
$V$ = Mean velocity.
ρ = Density of fluid.

CFD simulations for different flow conditions (e.g., laminar and turbulent) and quantities of heat supplied were performed using the commercial CFD software, STAR-CCM+. The CFD simulations also informed the design of the experimental set-up for the validation of computational results. The experimental set-up is designed to analyze the effects of changing just one AM surface in a cooling channel. This allows variation in weld track orientation and the effect of the wide range and complexity of topographies, commonly seen on as-built AM surfaces, to be explored.

## METHODOLOGY TO MODEL AM SURFACES

To analyze the effect of surface roughness on heat transfer,[eg 2] the "sand-grain" roughness model is commonly used in CFD modeling. In this approach, the profile Root Mean Square (RMS) value of roughness ($R_q$) is used to estimate the roughness heights ($K_s$, see Fig. 4 in [2]) of spherical grain and is used as an input that simplifies the heat transfer calculations. This model assumes an isotropic, statistically stationary surface topography, which does not describe LPBF AM surfaces well.

Our models to date suggest that the finest scales of roughness have a negligible effect compared to "weld-track" dimensions and orientation. The biggest limitation of the "sand grain" approach is that it can not model the complexity of real AM surfaces (e.g., build orientations, distribution of particles with different sizes and heights). To address this, a more realistic approach to model these AM surfaces have been developed, as described in [1], and CFD simulations were performed.

**Methodology to Model Spatter Deposits**

Hemispherical shapes appear to best capture the spatter particle shapes on real AM surfaces. However, from a CFD meshing perspective, these shapes are not practical since even a small number of hemispherical particles require a large number of cells, which in turn lead to impractical computational run times.

Our analyses indicate the particle density is in the range of 6.25 to 18.75 particles/$mm^2$. To accommodate these densities and reduce computational load, we replaced hemispherical particles with various other shapes (e.g. hexagonal prisms and cuboids) while keeping the effective surface area the same. The particle densities for cuboid particles are taken from the measured values from fabricated surfaces measured in prior work by one of the authors are used [3,4]. CFD analyses were then conducted for both laminar (Re = 1000) and turbulent (Re = 4000) flows. Comparisons of performance show that cuboids are a good replacement for hemispheres as they allow meshing up to ten times the number of particles than when using hemispherical particles.

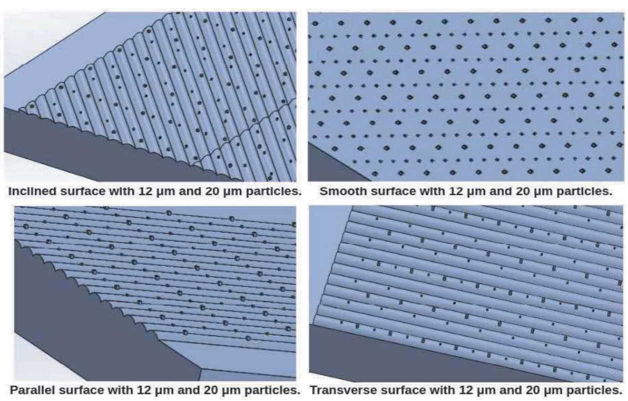


*FIGURE 1. Modeled AM surfaces with cuboid particles.*

As mentioned above, the cuboid particles have approximately the same surface area as the hemispherical particles. This requirement is imposed to ensure that the same surface area is available for heat-transfer irrespective of the shape of the particles. With this approach, the side of a cuboid particle was found to be 30 μm to get approximately the same surface area at the fluid-solid interface as a hemispherical of radius 30 μm. The same number of particles (both hemispheres and cuboids) were modeled and used in the CFD simulations to better understand the differences. Results shown in Fig. 2 show good agreement in Nu for the two particle geometries.

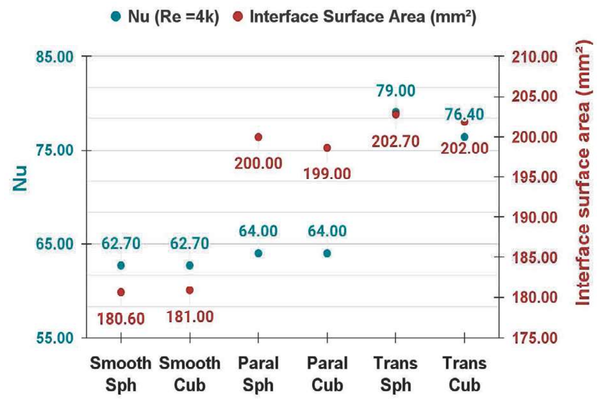


*FIGURE 2. Comparisons of results between spherical and cuboid particles (turbulent flow).*

**THE EXPERIMENTAL SET-UP**

For the smooth entry and exit of the fluid, converging-diverging sections were provided at both inlet and outlet. The experimental set-up (Fig.3) has been designed such that the maximum temperature at the metal-plastic interface should not exceed the deflection temperature of plastic and heat loss to the environment should also be less than 10 percent of total heat supplied. This design is also able to accommodate exchangeable AM parts.

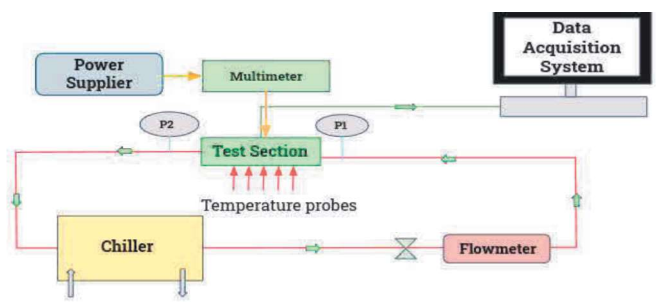


*FIGURE 3. Flow loop for experimental study*

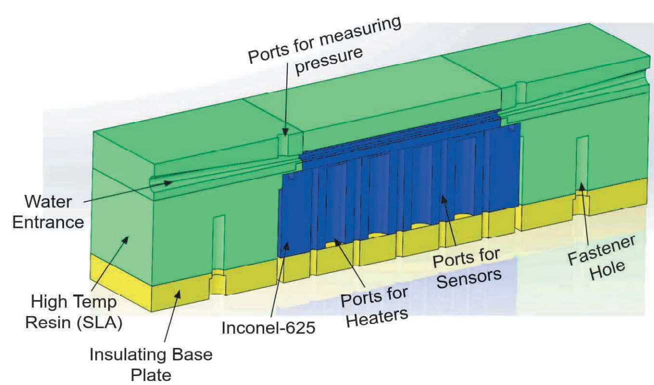


*FIGURE 4. The Experimental Set-up.*

## RESULTS AND DISCUSSION

The effects of AM surface roughness on heat transfer and fluid flow characteristics have been investigated in terms of the Nusselt number (Nu), the pressure drop ($\Delta P$) across the channel and the performance factor (PF). CFD results in terms of percentage change in Nu for various channels are shown in Fig. 5. Here, the smooth surface has been considered as a reference surface, therefore it has been assigned zero value for percentage change throughout the paper. Fig. 5 has been divided into two segments (i.e., light green and light pink) in order to explain the differences in the results in terms of which feature (e.g., weld track orientations relative to the flow or spatter deposits) is more dominant for heat transfer in terms of Nu. Light green highlights results from smooth and parallel tracks (with and without particles), while light pink highlights results from traverse and 45° inclined tracks (with and without particles). In the light green segment of Fig.5, it can be observed that when particles are placed on the top of smooth and parallel tracks, it increases Nu significantly. But while observing the results of the pink segment in the same figure, it can be seen that build angle is the more dominant factor. Placing particles on the top of transverse and 45° inclined tracks decrease the Nu by a small amount due to the stagnation of fluid behind the particles and in the valleys of the AM tracks (shown in Fig.6).

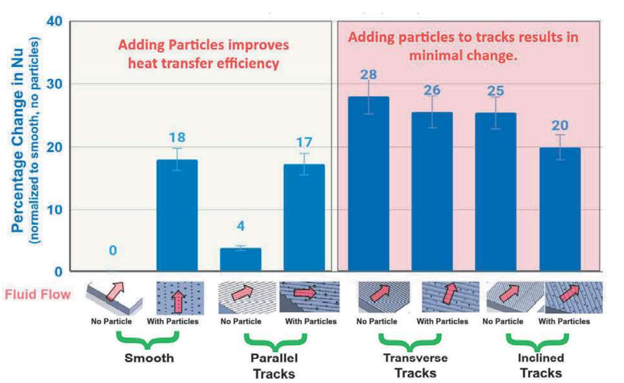


*FIGURE 5. CFD results in terms of Nu for turbulent flow (Re =4000).*

*FIGURE 6. Turbulent flow over transverse AM tracks containing particles.*

In much of the classical literature [5], surface area has been shown to be a critical feature to enhance heat transfer. Hence we plotted our results in terms of percentage change in Nu against the percentage change in interface surface area of different AM surfaces, as shown in Fig. 7.

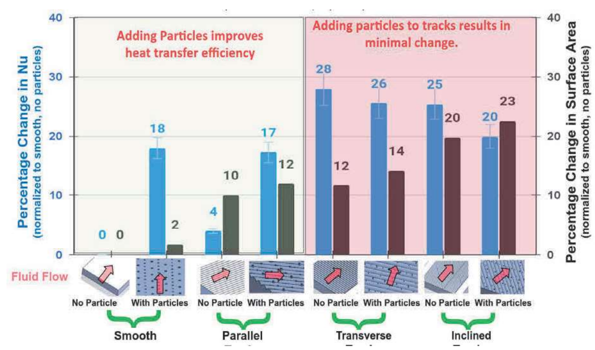


FIGURE 7. Percentage change in Nu vs. interface surface area for turbulent flow.

From Fig. 7, it can be seen that the impact of surface area on heat transfer is minimal in all cases and particles and build angles are more dominant features. The reason for this is an increase in mixing (or diffusion) within adjacent layers of fluid near the wall. This occurs due to the presence of particles and AM tracks rather than the amount of surface area exposed for heat transfer. Inspecting Figs. 5 and 7, adding particles improves heat transfer efficiency for smooth and parallel track surfaces. By contrast, transverse and inclined tracks improve heat efficiency more than particle addition. Transverse track alignment also shows highest heat transfer efficiency when the flow is turbulent.

Results for laminar flow (i.e., Re = 1000) for the modeled AM surfaces are shown in Fig. 8. When the mass flow rate decreased significantly (i.e., approximately by four times) and flow became laminar, AM surfaces were performing inversely and data indicate that smooth surface without particles has been performing well and even better than the smooth surface containing particles. While investigating the behavior of these AM tracks in more detail, we found that at a very low Reynolds number the fluid flowed in the form of layers. Therefore, it stagnates easily in the valleys of AM tracks and fills them with fluid layers and makes the surface flat. This creates a significant convection loss (determined by calculating temperature gradient at different locations in the valleys and at peaks of AM tracks in the CFD simulations) and AM tracks exhibited poor thermal performance.

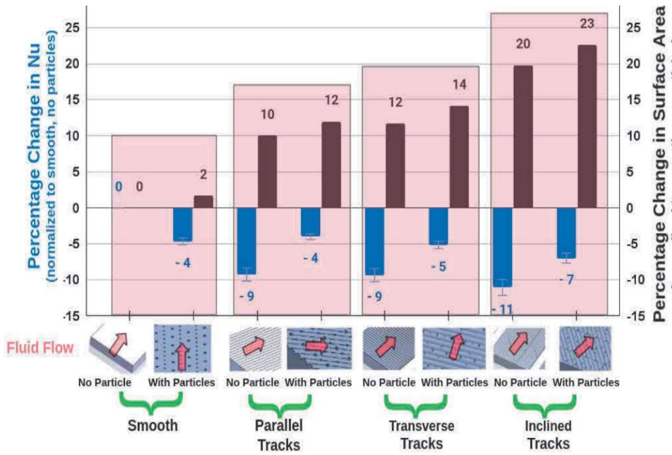


*FIGURE 8. Percentage change in Nu Vs. interface surface area for laminar flow.*

PF is an important parameter of this study because it tells us how much convective heat transfer can be increased at the expense of pressure drop, which is directly related to the pumping power. Use of PF depends on the application of these types of channels and where the pressure drop or pumping power requirements play a critical role in terms of performance. From Figs. 9-10, it can be observed that the difference in PF for different channels is minimal because the channel with the higher value of Nu also has the higher value of pressure drop and vice-versa. However, for long channels where the pressure drop may be much larger, the type of AM surface can have a bigger influence on the performance factor. Therefore, it is important to consider this when evaluating performance of AM surfaces in various heat transfer applications.

Note: Preliminary results from on-going experiments will be discussed during the conference.

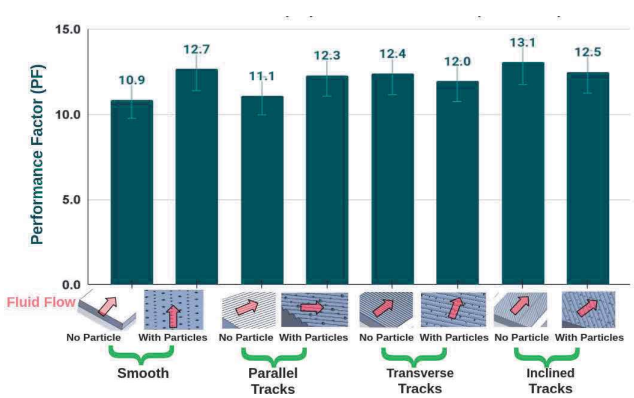


*FIGURE 9. Performance Factor (PF) for turbulent flow.*

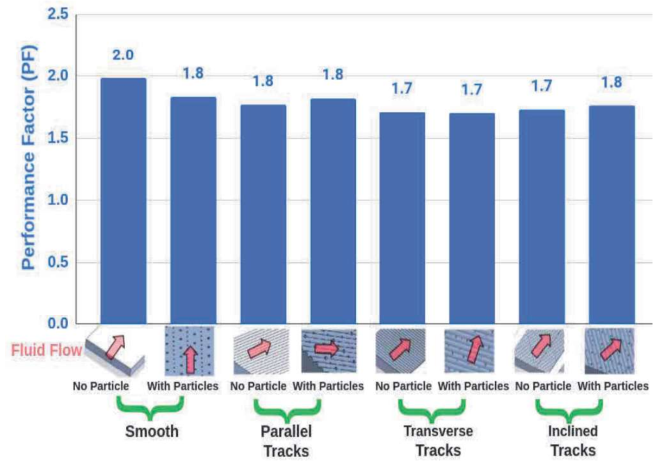


*FIGURE 10. Performance Factor (PF) for laminar flow.*

## CONCLUSION

The fluid-flow and heat-transfer characteristics of various modeled AM surfaces have been investigated for both laminar and turbulent flow conditions. CFD results show that the orientation of weld tracks relative to the flow and spatter/particles attached to the surfaces are the dominant features of AM surfaces and the surface area impact on heat transfer is minimal in all the cases. Under turbulent flow conditions, transverse track alignment shows highest efficiency and adding particles improves heat transfer efficiency for smooth and parallel track surfaces. However, when the flow becomes laminar, the opposite behavior was observed in CFD simulations.

## DISCLAIMER

Certain commercial entities, equipment, or materials may be identified in this document in order to describe an experimental procedure or concept adequately. Such identification is not intended to imply recommendation or endorsement by the National Institute of Standards and Technology, nor is it intended to imply that the entities, materials, or equipment are necessarily the best available for the purpose.

## ACKNOWLEDGEMENT

This work was funded in part by the University of North Carolina at Charlotte's Center for Precision Metrology Affiliates program and the National Institute of Standards and Technology. J. Tarr (NIST) supported sample fabrication. We would also like to thank Alkan Donmez and David Deisenroth for their help with the design of the experimental setup.

## REFERENCES


[1] Mandloi K, Evans C, Fox JC, Cherukuri H, Miller J, Allen A, Deisenroth DC, Donmez A. Toward specification of complex additive manufactured metal surfaces for optimum heat transfer. Advancing Precision in Additive Manufacturing. 2021; AM21136.

[2] Brezgin, D.V., Aronson, K.E., Mazzelli, F. et al. The surface roughness effect on the performance of supersonic ejectors. Thermophys.Aeromech 2017; 24, 553–561.

[3] J.C. Fox, Variation of Surface Topography in Laser Powder Bed Fusion Additive Manufacturing of Nickel Super Alloy 625, J. Res. NIST. 124 (2019).

[4] Fox JC, Evans C, Mandloi K. Characterization of laser powder bed fusion surfaces for heat transfer applications. CIRP Annals. 2021 Jan 1;70(1):467-70.

[5] Yeh Chien-Chih et al. Heat transfer analysis of a loop heat pipe with biporous wicks. International journal of heat and mass transfer. 2009, 4426-4434.