Joint Special Interest Group meeting between euspen and ASPE
Advancing Precision in Additive Manufacturing
Inspire AG, St. Gallen, Switzerland 2021
www.euspen.eu

Toward specification of complex additive manufactured metal surfaces for optimum heat transfer.

Kuldeep Mandloi[1], Chris Evans[2], Jason C. Fox[2], Harish Cherukuri[1], Jimmie Miller[1], Angela Allen[1], David C. Deisenroth[2], Alkan Donmez[2].

[1]*University of North Carolina at Charlotte, USA*
[2]*National Institute of Standards and Technology, Gaithersburg, USA*

kmandloi@uncc.edu

Abstract
Metal additive manufacturing (AM) offers the possibility of incorporating cooling channel geometry into components in high-temperature applications. Additionally, it has the prospect of optimizing cooling channel geometry unconstrained by geometric limitations of conventional machining processes. Such channels will necessarily have surfaces manufactured at various orientations resulting in different topographies that may influence heat transfer. Numerous studies have shown that conventional (amplitude) roughness parameters do not discriminate between topographies produced under different build conditions – but such descriptions have been used in studies of micro-channel, heat exchanger performance. The motivation behind this study is to explore the correlation between AM roughness characteristics (weld tracks amplitudes/wavelengths and their orientation, spatter, etc) and the resulting effect on heat transfer and pressure drop in fabricated microchannels. Computational Fluid Dynamics (CFD) models for mini-channels using StarCCM+ (a CFD code) were developed by acquiring the roughness data from the real AM surfaces with various roughness parameters such as different wavy patterns and the part orientation during the build. Simplified versions of measured surface topographies reduce the computational overhead. The pressure drops across the mini-channels and Nusselt (Nu) numbers were computed and analyzed for these cases under both laminar- and turbulent-flow conditions. Significant differences in the Nu numbers and pressure drops were observed across the different AM surfaces considered. Further CFD modeling of mini-channels with different wavy surfaces helped in exploring the suitable dimensions for the mini-channel experimental set-up and also enabled exploration of the Reynolds number range to consider experimentally. Heat-balance considerations have been used to validate the current findings. An experimental set-up is under development to compare models in an idealized set-up. Initial results from the experiments are also described.

Cooling, Experimentation, Fluid, Modeling, Surface Roughness

1. Overview

The aerospace community has a great interest in additive manufacturing (AM), particularly laser powder bed fusion (LPDF), since it can be used to manufacture parts from high-temperature nickel alloys. However, the uncertainty in material properties makes LPBF questionable for use in high stress turbine parts, though interest for high temperature parts with complex cooling channels is growing [1]. The complexity of these cooling designs is limited in conventional manufacturing methods. Using AM, designers have the opportunity to create cooling geometries that would be impossible to achieve with other methods. While roughness of external surfaces may be reduced through conventional machining, most surface treatment processes cannot be applied to internal channels especially when the dimensions are a millimeter or sub-millimeter scale [6].

For adoption of AM for parts requiring complex cooling channels to continue, an understanding of the relationship between the as-built surface finish and heat transfer must be developed. In LPBF, there are numerous build parameters, such as part orientation during the build, that affect the final part-surface topography and hence heat transfer. There is already a substantial literature in the measurement of surfaces produced by metal additive manufacturing and by powder bed fusion techniques, in particular [2]. It is generally recognized that complex, textured surfaces are poorly characterized or specified by conventional statistical parameters (see for example [3]). The classic literature on the impact of surface roughness on heat transfer (Moody's diagram) uses a simplified treatment of surface roughness, while powder bed fusion processes generate complex surfaces with a mixture of strongly anisotropic, periodic components and a population of individual features (pores and particles, for example), all of which may affect heat transfer and fluid flow. Simonelli et al. [4] have investigated the effect of build direction on mechanical properties of Selective Laser Melting (SLM) and Direct Metal Laser Sintering (DMLS) parts. However, no build-direction studies have examined the effects on internal channel geometry with spatter and their effects on heat transfer at different Reynolds number conditions.

The focus of this study is the dependence of fluid-flow and heat transfer characteristics on the surfaces based on build-orientation and spatter. Our goal is to relate appropriate surface specification to heat transfer. Various Computational Fluid Dynamics (CFD) models for mini channels using Star CCM+ were developed by acquiring the roughness data from the real AM surfaces with varying surface characteristics such as different wavy patterns and build direction variation. Simplified geometric models of AM surfaces based on the measured surface topographies were developed to reduce the computational overhead (see Figs 3 and 4). The pressure drops across the mini-channels and Nusselt numbers (Nu) are computed and analyzed for a variety of surface topographies under both laminar and turbulent flow conditions. The CFD results show a significant difference in Nu numbers and pressure drop with AM surfaces and different part orientation during the builds. Further CFD modeling of mini-channels with different wavy surfaces facilitated exploration of suitable dimensions for the mini-channel experimental set-up and also enabled identification of the Reynolds number range for the experiments. Heat balance considerations and results from equipment experimental setup have been used to validate the current findings.

2. Methodology to Model AM surfaces

In this work, simplified AM surfaces were modeled. Prior work on characterization of topography with changes in part orientation during the build is published in CIRP Annals [5]. In that work, several characterization techniques analyzed surface data captured using a focus-variation microscope to better understand the effect of part orientation during the build on surface features. Investigations of the area scale (Figure 1), amplitude-wavelength content, and positions of partially-melted powder particles on the surface showed the large dynamic range of surface-structure wavelengths and amplitudes. Initial attempts at modeling heat transfer with real AM surfaces were unreasonable due to long computational time as a result of the required mesh sizes.

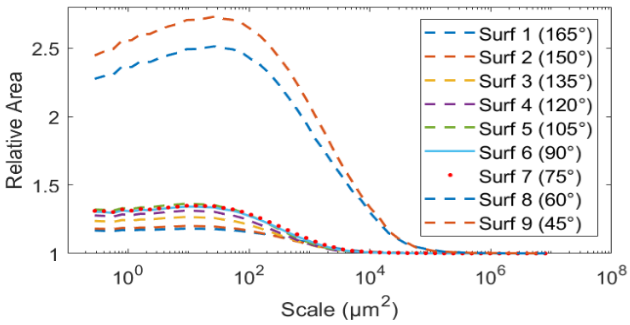

Figure 1. Relative Area analysis of AM different surfaces [5].

The methodology to model AM surfaces is shown in Fig. 2. To model the wavy surfaces, the root mean square (RMS) dimensions of actual surfaces (e.g., wavelength of 150 µm and amplitude of 30 µm) have been used. The wavy surface is idealized by using the absolute value sine function with a period and amplitude consistent with these RMS values.

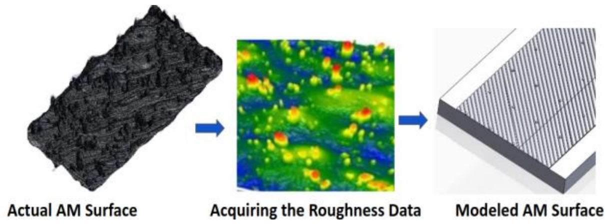

Figure 2. Methodology to Model AM Surfaces.

To test variation due to part orientation during the build, three surface patterns have been modeled. These surfaces include bead orientations (track/scan directions) parallel to the flow, transverse to the flow, and at a 45° angle to the flow. For the comparison of CFD results, a smooth surface has also been modeled as a reference. According to Figures 3 and 5 in [5], the relative area of AM surfaces is significantly affected by protrusions (spatter deposits). To identify the effect of those spatter deposits on the heat transfer, hemispherical protrusions on the top of wavy surfaces were added. The size of the protrusions (i.e., spatter) is the mean radius (29 µm) of the powder particles used. Modeled AM surfaces with different part orientations during the builds, with and without spatter, are shown in Fig.4.

3. Methodology for Numerical Investigation

3.1. CFD Models

The CFD models and planned test channels have a thermoplastic resin (I.e, Polyoxymethylene, POM) side and upper walls while the bottom part of the channel was made of nickel superalloy 625 (IN625) (Fig 3). The POM thermoplastic resin was chosen as the material for side and upper walls as preparations for experimental analysis showed this material could be fabricated to the required designs while isolating the effects of AM surfaces and reducing the conduction loss from side and upper walls.

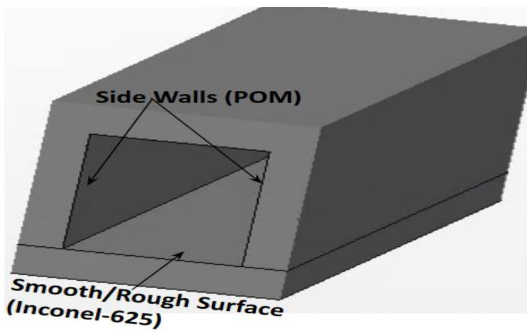

Figure 3. CFD Model with artificial AM surfaces

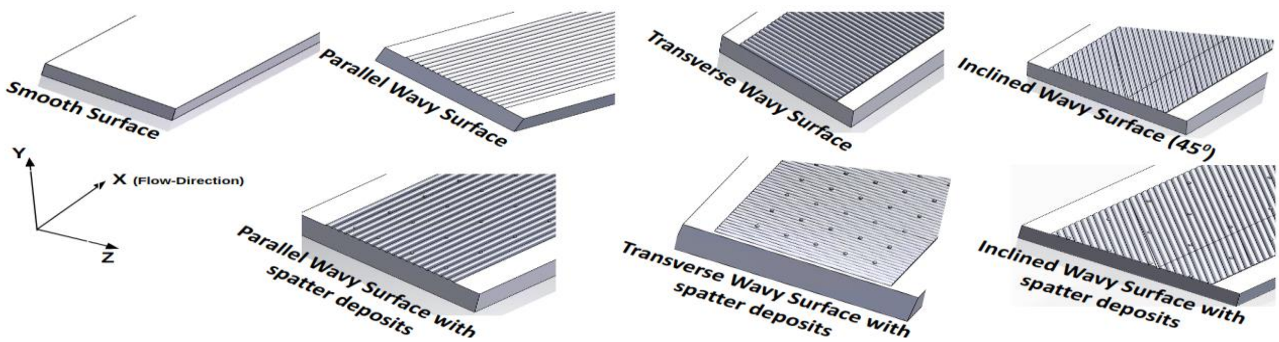

Figure 4. Modeled AM Surfaces

3.2. Model Assumptions

Steady-state conditions were assumed for the fluid flow and heat transfer through conduction and convection. Mass flow inlet and pressure outlet boundary conditions were applied at the entry and exit of the channel, respectively. All sidewalls of the channel were assumed to be insulated and constant heat flux was provided at the bottom of the channel. In-built laminar and turbulent flow models (k-ϵ model) in StarCCM+ were used during the simulations.

4. Grid Independence Study

For the grid independence analysis, the average surface-output temperature was plotted against the number of cells (shown in Fig.5) and various trades were run to optimize the mesh size, ranging from 85000 to 5 million cells. The optimum number of cells was approximately 3 million. For the meshing, the prismatic, boundary-layer cells were used to resolve the boundary layers at fluid-solid interfaces. In order to capture the circulation near the wavy surfaces, the growth size of prismatic layers was kept at 15 percent of the base size.

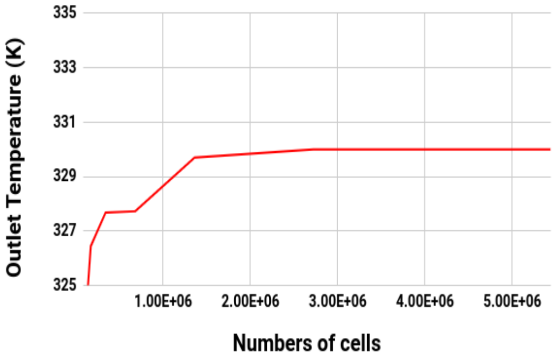

Figure 5. Grid Independence Analysis.

5. CFD Results and Discussion

The effects of surface roughness on heat transfer were analysed in terms of the Nusselt Number (*Nu*), the Pressure Drop (ΔP) across the channel, and a Performance Factor (*PF*). Here, the performance factor has been defined as the ratio of Nu to (ΔP).

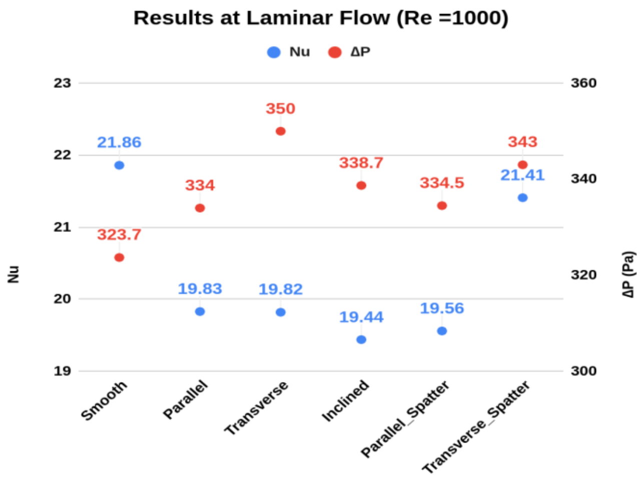

Figure 6. Nusselt Number (*Nu*) and Pressure Drop (Δ P) for various channels for the laminar flow case.

The laminar flow (*Re* = 1000) results for Nusselt Number (*Nu)* and Pressure drop (ΔP) for various channels containing different artificial rough surfaces are shown in Fig 6. The corresponding results for turbulent flow with *Re* = 4000 are shown in Fig 7. As the figures indicate, smooth surfaces perform (thermally) the best (higher *Nu* and lower ΔP values) under laminar flow conditions. However, under turbulent flow conditions, the *Nu* number for smooth surface case decreases significantly. Another interesting observation is that the channel with transverse AM surface and spatter deposits performs as well as the smooth surface case under laminar flow conditions. However, when turbulent flow conditions are present, it performs better than the smooth surface case. This is also confirmed by the performance factor calculations shown in Fig 8. From Fig 7, it is clear that, for the turbulent flow case, the thermal performance of the channels with transverse AM surfaces is much higher than the channel with smooth surfaces.

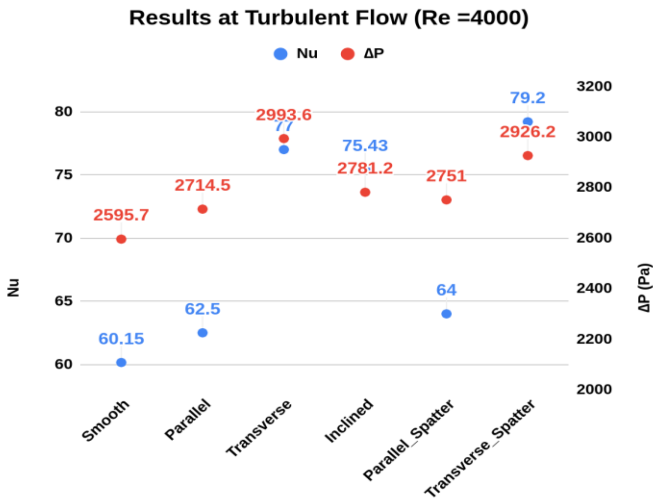

Figure 7. Nusselt Number (*Nu*) and Pressure Drop (ΔP) for various channels for the turbulent flow case.

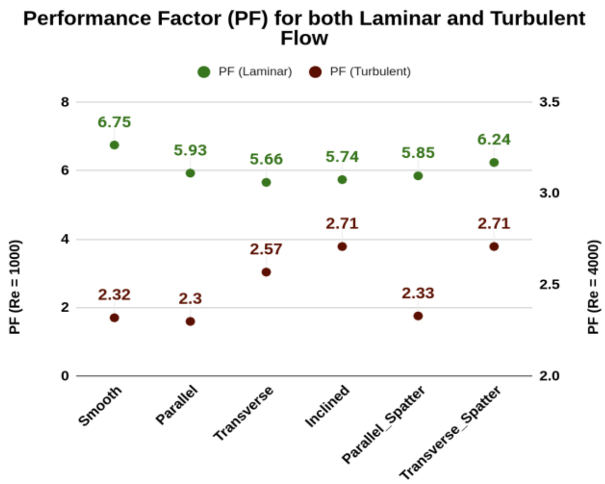

Figure 8. Performance Factor (PF) x 100 for various channels for the laminar and turbulent flow cases considered.

6. Validation of CFD Modeling

To validate the CFD modeling and to test the experimental equipment, an initial experimental setup was designed with aluminium alloy 6061(having smooth internal surface) and tested with different mass flow rates and amount of heat supply. The test setup contains four holes (on the upper side of the channel) with different sizes to accommodate two cylindrical heaters and temperature probes. Figures 9 and 10 show the CFD model and the setup for equipment testing. The experimental results from this test setup were used to validate the steady-state CFD simulation. Preliminary results from the experiment are shown in Fig.11 and the CFD and experimental results are shown in Table 1 for comparison.

From Table 1, it can be observed that the CFD and the experimental results are quite similar. The slight difference in the outlet temperature is due to the varying environmental conditions and distance from the actual outlet of the channel to the outlet water temperature probe (as shown in Fig. 9). Similarity between the results, however, validates the CFD models.

Table 1: **Comparison of CFD and Experimental results (Laminar-Flow)**

Parameters	CFD Results	Experimental Results (mean)	Uncertainty in Experimental Results
$\dot{m}$	0.0053 kg/s	0.0053 kg/s	-
Q_{in}	28 W	28 W	-
T_{in}	26.85°C	26.85°C	-
T_{room}	22.20°C	22.20°C	-
T_{out}	29.10°C	28.15°C	0.01°C
T_{Probe1}	35.30°C	33.12°C	0.03°C
T_{Probe2}	37.20°C	35.35°C	0.06°C
T_{Probe5}	35.40°C	33.23°C	0.05°C
Time Constant	32.01 s	32.6 s	

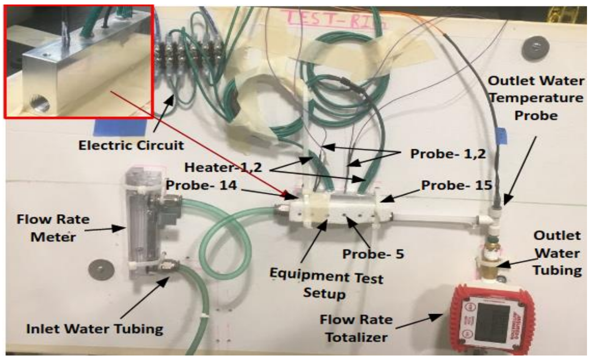

Figure 9. Experimental Setup.

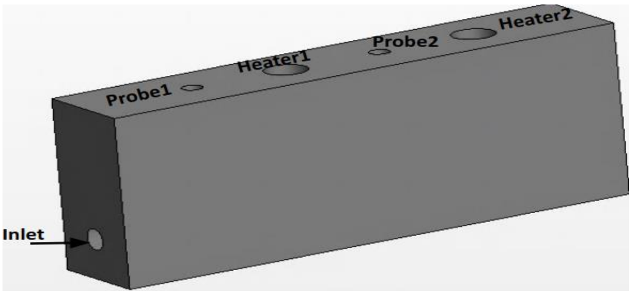

Figure 10. CFD Setup.

7. Conclusion

In this paper, the fluid-flow and heat-transfer characteristics of various mini-channels with different AM surfaces were studied computationally. Both laminar- and turbulent-flow conditions were considered in the CFD simulations. The computational studies show that smooth surfaces provide the best thermal performance under laminar-flow conditions. However, under turbulent-flow conditions, smooth surfaces along with parallel wavy surfaces appear to perform the poorest. Interestingly, the transverse AM surface with spatter deposits performs almost as well as the smooth surface case in laminar-flow studies and significantly better in turbulent-flow studies. This observation is further confirmed by the performance factor calculations. Preliminary results on validating CFD results with experiments have been encouraging. Further experimental work to validate the CFD results for various AM surfaces is currently under development.

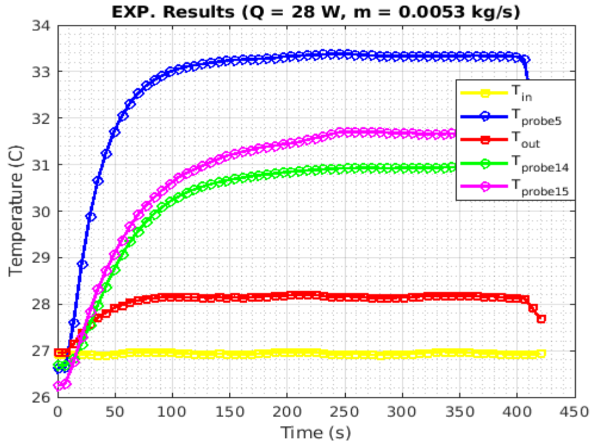

Figure 11. Preliminary Results from Experiments.

Acknowledgements

We would like to extend gratitude to the Affiliates of Center for Precision and Metrology (CPM) at UNC Charlotte for providing the financial support to this project. We also extend our heartiest thanks to the CPM and the MEES Department at UNC Charlotte for providing access to their research facilities such as high-performance computing (HPC) clusters for CFD simulations and CPM laboratories for conducting experiments.

References

[1] Jacob C. Snyder, Curtis K. Stimpson, Karen A. Thole and Dominic Mongillo Y 2016 Build Direction Effects on Additively Manufactured Channels J. Turbomach. **TURBO-15**-1262.

[2] R.K. Leach, D. Bourell, S. Carmignato, A. Donmez, N. Senin, W. Dewulf, Geometrical metrology or metal additive manufacturing, CIRP Ann. 68 (2019) 677–700.

[3] X.Jiang, N. Senin, P.J. Scott, Francois Blateyron Feature based characterization of surface topography and its application, CIRP Ann 70,2(2021) in press

[4] M. Simonelli, Y.Y. Tse and C. Tuck Y *2014 J. Materials Science and Engineering.* **616** 1-11.

[5] Jason Fox, Chris Evans and Kuldeep Mandloi Y 2021 Characterization of laser powder bed fusion surfaces for heat transfer applications CIRP Annals Manufacturing Technology.

[6] Pyka G., Burakowski A., Kerckhofs G., Moesen M., Van Bael S., Schrooten J. and Wevers M. Y 2012 Surface modification of Ti6Al4V open porous structures produced by additive manufacturing. Adv. Eng. Mater. 14(6) pp. 363–370.

Disclaimer