Periodica Polytechnica Mechanical Engineering, 69(4), pp. 392–396, 2025

Development of Part Cooling in 3D Printer through the Design of a Custom Cooling Duct Using Generative Design

Bálint Leon Seregi[1]*, Péter Ficzere[1]

[1] Department of Railway Vehicles and Vehicle System Analysis, Faculty of Transportation Engineering and Vehicle Engineering, Budapest University of Technology and Economics, Műegyetem rkp. 3., H-1111 Budapest, Hungary
* Corresponding author, e-mail: seregibalint@edu.bme.hu

Received: 25 October 2025, Accepted: 30 October 2025, Published online: 28 November 2025

Abstract

This study addresses the improvement of part cooling in FFF 3D printing, using the Creality Ender 3 V2 as a case study. The stock cooling system directs airflow from a single side, which leads to uneven cooling and reduced surface quality. To overcome this limitation, both a reference redesign and a generative design approach were investigated. The development involved flow domain optimization with Autodesk Fusion's Generative Design module, followed by CFD simulations in Ansys CFX. Results indicate that generative design can enhance cooling performance and enable more effective two-sided airflow. At the same time, challenges remain in achieving uniform outlet velocity distribution, as flow separation effects may occur. Overall, the study demonstrates the potential of generative design as an innovative tool for accelerating fluid dynamics design processes, while highlighting the need for further refinement and experimental validation.

Keywords

additive manufacturing, generative design, MEX, CFD

1 Introduction

In recent years, generative design has transformed the development of components. Initially, its application was primarily focused on mechanical optimization problems, such as lightweighting, stiffness maximization, and compliant mechanism design in additive manufacturing [1]. The integration of topology optimization and machine learning methods has made it possible to explore complex design spaces that are often inaccessible to traditional, intuition-driven engineering approaches [2, 3]. In recent years, generative methods have been extended beyond structural mechanics into the domain of fluid dynamics. Here, they have been employed to minimize pressure drop in flow channels, maximize convective heat transfer, and optimize aerodynamic performance [4, 5]. CFD frameworks now enable direct gradient-based optimization of flow fields, while reinforcement learning and surrogate modelling approaches have expanded the applicability of generative design to industrially relevant problems such as heat exchanger channel optimization, atomizer design and duct flow uniformity [6–9]. This development directly motivates the use of generative CFD in extrusion-based additive manufacturing (MEx). In this process, part cooling plays a decisive role: the airflow distribution around the printed part influences filament solidification, interlayer adhesion, crystallinity, residual stress formation, and dimensional accuracy [9–11]. Insufficient or non-uniform cooling can lead to surface defects, warping, poor bridging quality, or weakened interlayer bonds [12]. Conversely, properly optimized cooling geometries can stabilize overhangs, improve dimensional fidelity, and enhance mechanical properties [13].

The design of part-cooling ducts in FFF printers is particularly well-suited for generative CFD, as it embodies the same trade-offs typical in fluid design optimization: balancing cooling uniformity, targeted local airflow, pressure loss, and noise generation. By coupling CFD-driven generative design with in-situ thermal monitoring techniques such as infrared thermography, it is possible to close the loop between simulation and experiment, achieving data-driven optimization of cooling strategies.

In this work, generative CFD was applied to the design of a part-cooling duct, with the aim of improving convective cooling uniformity while maintaining acceptable pressure drop and maximized outlet air velocity. The approach

Cite this article as: Seregi, B. L., Ficzere P. "Development of Part Cooling in 3D Printer through the Design of a Custom Cooling Duct Using Generative Design", Periodica Polytechnica Mechanical Engineering, 69(4), pp. 392–396, 2025. https://doi.org/10.3311/PPme.42624

builds on the recent advances in differentiable CFD and inverse fluid design, applying them to a practical additive manufacturing challenge where thermal history directly governs final part quality.

The cooling system of the Ender 3 V2 printer cools the part only from one direction by default. As a result, the undercut surfaces opposite the cooling outlet cool at a different rate, which prevents proper dimensional accuracy and significantly degrades the surface quality in case of certain overhangs (see Fig. 1). To address this issue, we intend to build a new cooling system, initially using only the stock fans, in order to attempt the development without relying on any newly purchased components. For the development, the Generative Design module available in Autodesk Fusion 360 [14] was utilized, which also provides the possibility to optimize flow domains. To ensure comparability of the results, a simulation was also carried out using the flow domain extracted from the model of the original cooling channel. The aim of the study is to investigate and enhance the fluid dynamic and thermal phenomena occurring in the cooling channel of the Creality Ender 3 V2 [15] using numerical methods. The objectives include mapping the flow within the channel, accelerating the flow at the channel outlet through geometric modification, and applying Generative Design in the optimization process as a novel and innovative design approach.

Fig. 1 The surface error caused by insufficient cooling on the overhangs

2 Methodology

Before the CFD analysis, the optimization of the flow domain was carried out. For this, it was necessary to define the preserved, obstacle, and starting volumes. The preserved geometry represents the volume that the generative algorithm cannot remove or modify. In this case, these were the inlet and outlet geometries, more precisely the fixed flow domains where the boundary conditions for the subsequent CFD analysis would be applied. The obstacle geometry acts as a dead zone within the optimization space, since the program is not allowed to place flow volumes there. In this case, these were the printer head's carriage, hotend, and the fans, whose positions had to be specified.

Fig. 2 shows the complete head assembly CAD model, which made it easier to define both the obstacle geometries and the initial geometry. The initial geometry was required to provide a reference shape for the flow domain, similar to the one we intended to achieve. In this case the double outlet section was the goal which had to be defined in advance in the form of the starting shape. The geometry set can be seen on Fig. 3. For the generation parameters, the velocity of the medium passing through the inlet was specified based on the manufacturer's fan data. The optimization objective was to minimize pressure drop and to reach a target flow volume.

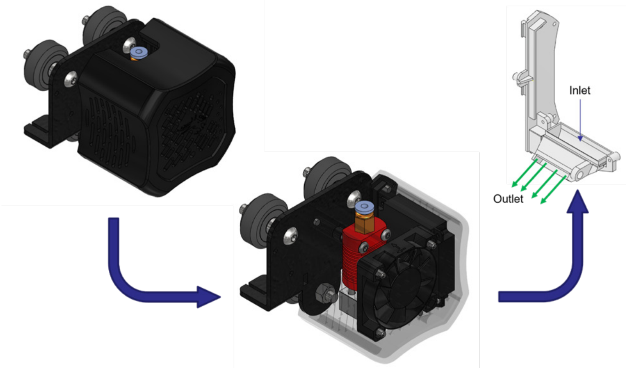

Fig. 2 The CAD model of the original printer head and it's fan duct

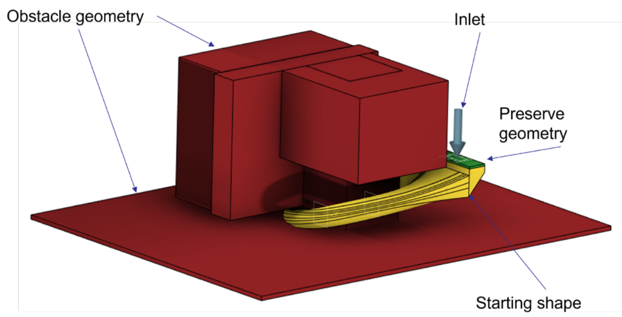

Fig. 3 The design geometries in the generative design space

After generating the design result, CFD simulations were carried out in Ansys CFX [16]. Following the proper definition of inlet and outlet boundaries and the generation of the numerical mesh, boundary conditions and material properties were specified (Figs. 4 and 5). The inlet mass flow was set to 0.00188 kg/s at a static temperature of 25 °C, assuming subsonic flow. At the outlet boundary, subsonic flow was also defined with a static pressure of 0 MPa. The walls of the flow domain were set as no-slip walls with adiabatic heat transfer and low surface roughness. The SST turbulence model was applied. In the simulation results, the maximum and cross-sectional average velocities at the outlet were monitored.

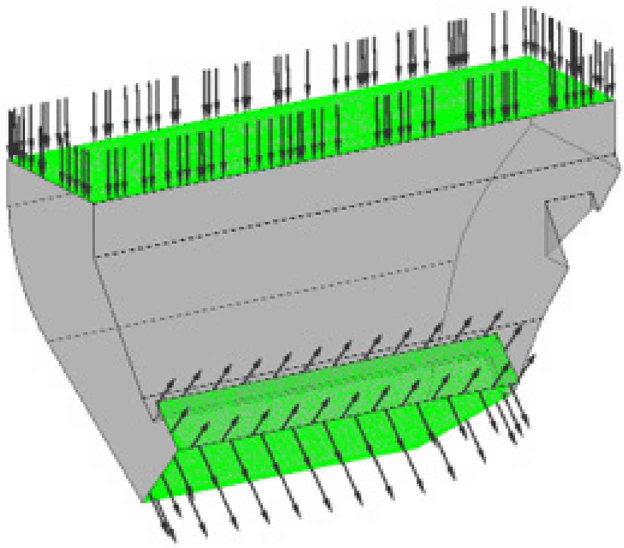

Fig. 4 Flow setup of the original duct in CFX

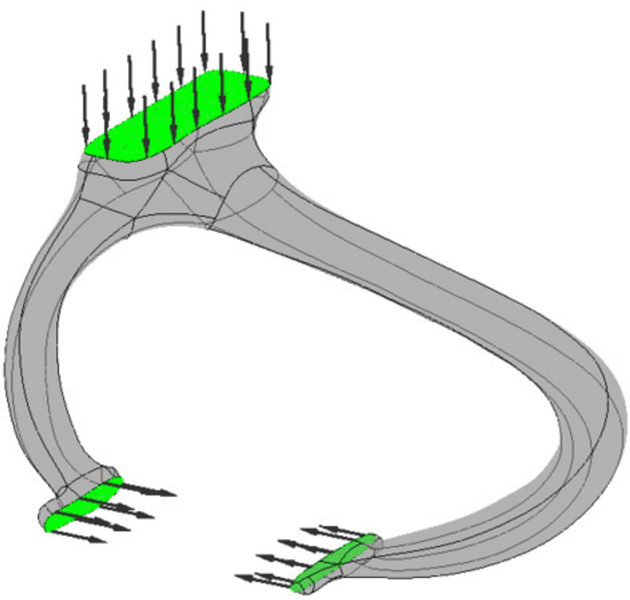

Fig. 5 The generative designed flow volume in CFX

3 Experimental results

Among the simulation results, the analysis of the original channel shows that due to its confuser-like design, the flow velocity continuously increases. However, because of the asymmetric geometry, the inclined channel wall causes an uneven velocity distribution at the outlet cross-section, resulting in a local maximum on one side (see Fig. 6).

The outlet maximum and average velocity is 17.13 m/s and 32.62 m/s respectively. In the case of the generatively designed flow domain, the flow velocities developed significantly differently. The global maximum did not occur at the outlet cross-section, and despite the overall uniform flow, the curved geometry caused spiral-like vortical flows to form within the channels. At the outlet, the maximum flow velocity reached 58 m/s, while the average velocity was 35.26 m/s. The average value alone exceeds the maximum flow velocity of the original factory configuration.

A more detailed examination of the outlet cross-sections reveals that although the flow velocity is higher, its distribution shows significant variations and is not uniform (this is most clearly visible in Figs. 7 and 8). Near the center of the cross-section the velocity is nearly uniform, while at the edges only minimal flow can be observed, lower by orders of magnitude. This can be explained by the fact that the flow domain expands too abruptly, leading to a flow pattern similar to the Carnot–Borda effect (see Fig. 9). Since the algorithm is not aware that a uniform velocity distribution across the entire cross-section

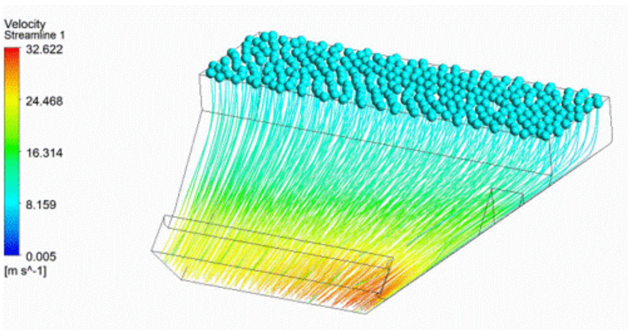

Fig. 6 Velocity [mm/s] and streamlines of the original duct

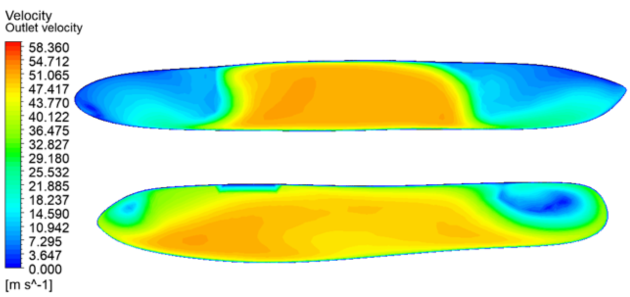

Fig. 7 Average velocity [mm/s] on outlet of the generative result

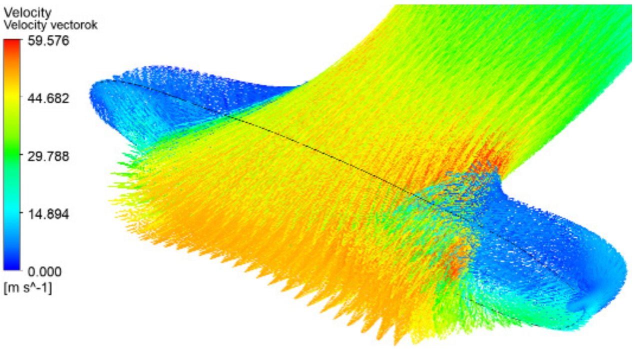

Fig. 8 Outlet velocity [mm/s] vectors of the generative result

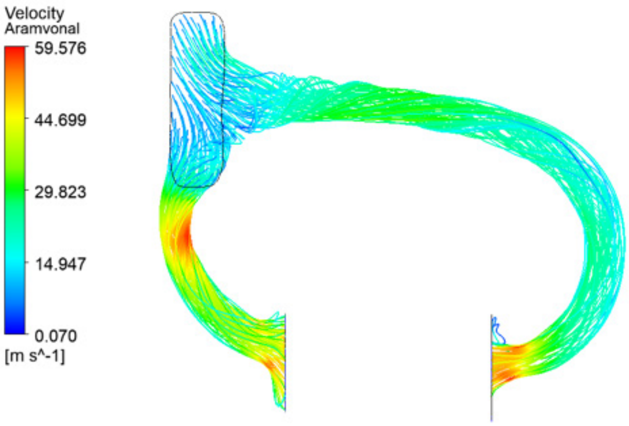

Fig. 9 Velocity [mm/s] streamlines of the generative result

is required, its objective function focuses solely on minimizing pressure drop. Consequently, the functionality of the outlet becomes questionable. Repeated runs of the generative algorithm resulted in similar outcomes.

4 Conclusion

With the generative algorithm, it was possible to achieve a higher outlet velocity while ensuring cooling from both sides of the part. However, in the case of the generative model, due to the constraint of the mandatory outlet geometry defined by the user, the velocity distribution along the cross-section is not uniform so it cannot be considered final.

In the next steps, the generative result needs to be remodeled into a channel with mounting points and validated through real manufacturing, which is part of the future plans. Nevertheless, it can be stated that generative design has the potential to accelerate fluid dynamics design processes once the definition of algorithm parameters and objective functions becomes more accessible. In its current state, however, only limited configuration options are available, and a high level of expertise is required to set the parameters and to generate usable results.

References

[1] Gaynor, A. T., Guest, J. K. "Topology optimization considering overhang constraints: Eliminating sacrificial support material in additive manufacturing through design", Structural and Multidisciplinary Optimization, 54(5), pp. 1157–1172, 2016. https://doi.org/10.1007/s00158-016-1551-x

[2] Schumacher, C., Bickel, B., Rys, J., Marschner, S., Daraio, C., Gross, M. "Microstructures to control elasticity in 3D printing", ACM Transactions on Graphics (TOG), 34(4), 136, 2015. https://doi.org/10.1145/2766926

[3] Hu, S., Jin, Q., Gao, C., Zhang, X., Lu, M., He, Y., Chu, D., Bai, W. "The new paradigm of computational fluid dynamics: Empowering computational fluid dynamics with machine learning", Physics of Fluids, 37(8), 081302, 2025. https://doi.org/10.1063/5.0280743

[4] Yonekura, K., Hattori, H., Nishizu, T. "Fluid topology optimization and additive manufacturing of a liquid atomizer using an extensive number of grid points", The International Journal of Advanced Manufacturing Technology, 126(3), pp. 1799–1806, 2023. https://doi.org/10.1007/s00170-023-11084-y

[5] Fawaz, A., Hua, Y., Le Corre, S., Fan, Y., Luo, L. (2022) "Topology optimization of heat exchangers: A review", Energy, 252, 124053. https://doi.org/10.1016/j.energy.2022.124053

[6] Bezgin, D. A., Buhendwa, A. B., Adams, N. A. "JAX-Fluids: A fully-differentiable high-order computational fluid dynamics solver for compressible two-phase flows", Computer Physics Communications, 282, 108527, 2023. https://doi.org/10.1016/j.cpc.2022.108527

[7] Bezgin, D. A., Buhendwa, A. B., Adams, N. A. "JAX-Fluids 2.0: Towards HPC for differentiable CFD of compressible two-phase flows", Computer Physics Communications, 308, 109433, 2025. https://doi.org/10.1016/j.cpc.2024.109433

[8] Kim, I., Jeon, Y., Chae, J., You, D. "Deep Reinforcement Learning for Fluid Mechanics: Control, Optimization, and Automation", Fluids, 9(9), 216, 2024. https://doi.org/10.3390/fluids9090216

[9] Vanaei, H. R., Khelladi, S., Deligant, M., Shirinbayan, M., Tcharkhtchi, A. "Numerical Prediction for Temperature Profile of Parts Manufactured using Fusws Filament Fabrication", Journal of Manufacturing Processes, 76, pp. 548–558, 2022. https://doi.org/10.1016/j.jmapro.2022.02.042

[10] Zhang, J., Neeckx, J., Troukens, J., Ferraris, E. "A reheating temperature criterion for adaptive strategy in fused filament fabrication", CIRP Annals, 71(1), pp. 197–200, 2022. https://doi.org/10.1016/j.cirp.2022.03.046

[11] Pálfi, T., Ficzere, P. "Aerodynamic Study of Different Types of Wingtip Devices", Periodica Polytechnica Transportation Engineering, 53(1), pp. 77–86, 2025. https://doi.org/10.3311/PPtr.37240

[12] Gabštur, P., Kočiško, M., Török, J., Kaščak, J. "Enhancement of Primary Profile Surface Quality in FFF Printing Through Vibration Monitoring and Active Control", Applied Sciences, 15(21), 11346, 2025. https://doi.org/10.3390/app152111346

[13] Choudhary, M., Mukherjee, S., Kumar, P. "Analysis and optimization of geometry of 3D printer part-cooling fan duct", Materials Today: Proceedings, 50, pp. 2482–2487, 2022. https://doi.org/10.1016/j.matpr.2021.10.444

[14] Autodesk Inc. "Autodesk Fusion 360, (v2.0.11514)", [computer program] Available at: https://www.autodesk.com/products/fusion-360 [Accessed: 25 October 2025]

[15] Creality Co. Ltd. "Creality Ender-3 V2 3D printer", [computer program] Available at: https://www.creality.com/products/ender-3-v2-3d-printer-csco [Accessed: 25 October 2025]

[16] ANSYS Inc. "ANSYS CFX, (v21.2)", [computer program] Available at: https://www.ansys.com/academic/terms-and-conditions [Accessed: 25 October 2025]