https://doi.org/10.2352/AIT.2026.1.1.8

Numerical and experimental investigation of airflows around inkjet printheads

Luc Nussbaumer, Johannes Renner and Gioele Balestra
iPrint, HEIA-FR, HES-SO University of Applied Sciences and Arts Western Switzerland, CH-1700 Fribourg, Switzerland

Abstract

Inkjet printing at elevated print gaps suffers from a number of quality issues caused by unstable airflows between the printhead and the substrate. A comprehensive understanding of the mechanisms underlying these defects is imperative. Computational Fluid Dynamics (CFD) facilitates rapid scanning of various configurations, enabling efficient execution of parametric studies. Yet, the validity of the CFD simulation must be confirmed with experimental measurements of the flow and printed patterns. In this work, a numerical simulation framework was developed and experimentally validated. Good quantitative and qualitative agreement was obtained for the investigated configurations. The numerical simulations were also used to successfully reproduce typical print quality issues induced by unstable airflows such as wood grain effect. The ability to capture wood grain effects opens the door to airflow stabilization strategy studies. The integration of numerical airflow studies in the development of next generation printheads, printers, and printing processes could enable upfront study of instabilities and thus remediate them before costly experimental trials.

Introduction

Driven by the digital revolution and the demand for personalized products, inkjet printing has evolved from being mainly used for graphical applications to a digital fabrication toolkit. As inkjet technology evolves, the push for higher resolutions and smaller droplet sizes—combined with the need for larger throw distances—introduces new aerodynamic complexities that can jeopardize print quality. Complex and unstable airflows emerge, disrupting droplet trajectories and leading to severe drop misplacement, often referred to as wood grain patterns (see Figure 2) [1].

The rationale for these undesired patterns (see Figure 2) is the complex and unstable airflow that affects the drop trajectory for large gaps and high-productivity configurations. The situation is even more dramatic in high resolution configurations, when the smaller droplets become more sensitive to unstable airflows given to their lower momentum [4]. Furthermore, at large print-head-to-substrate throw distances, the lateral airflows within the printer become more significant, which can induce additional droplet misplacement errors.

The airflow is induced by the interaction of the Couette flow (resulting from the relative motion between the printhead and the substrate) and the impinging air jet due to the droplets (see Figure 1) [2].

To prevent these quality defects as efficiently as possible, it is necessary to understand the root cause of the airflow instabilities. Visualization of the airflow is therefore necessary. Such visualization can be achieved experimentally using Particle Image Velocimetry (PIV), but the visualization is often limited to one laser plane and is hard to perform in a low-light environment (the distance between the substrate and the printhead is a few millimeters). Furthermore, testing different configurations can be time-consuming or not even possible. Computational Fluid Dynamics (CFD) enables faster screening of various printing configurations and provides access to flow information throughout the whole airflow field. Clearly, CFD would be a formidable tool to implement into inkjet printing system development. Yet, the CFD approach must be carefully validated beforehand.

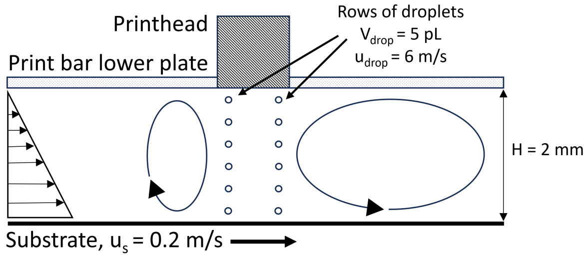


Figure 1: Sketch of the considered configuration. The airflow induced by the flying droplets, combined with the airflow resulting from the relative motion of the substrate and the printhead, creates large eddies front, after and between the rows of droplets.

Figure 2: Example of wood grain effect. The printed image should be of uniform color.

The aim of this project is to study airflows through CFD and to validate the results obtained using experimental measurements.

Materials and methods

A simulation setup has been developed using ANSYS FLUENT. A simple rectangular parallelepiped is used as geometry (see Figure 3). The geometries are based around EPSON S800 and S3200 printheads. An hybrid structured/polygonal mesh has been employed to reduce the computational cost. This allows a fine resolution in the surroundings of the droplets rays and limits greatly the amount of elements further away (see Figure 3). Special care has been given to ensure smooth transition between the two zones. Boundary layer elements have been employed to properly resolve the flow close to the walls.

A Couette flow is used as initial condition for the simulation; its presence being attested experimentally as well as analytically. The printing parameters employed in this investigation are detailed in Table 1. Key parameters include the substrate velocity, denoted as u_s, and the printhead-to-substrate gap, H. The drops are characterized by a volume V_{drop} and an initial ejection speed u_{drop}. The printing frequency is defined as F. A relatively small drop volume of 5 pL has been specifically selected to ensure high sensitivity to aerodynamic effects.

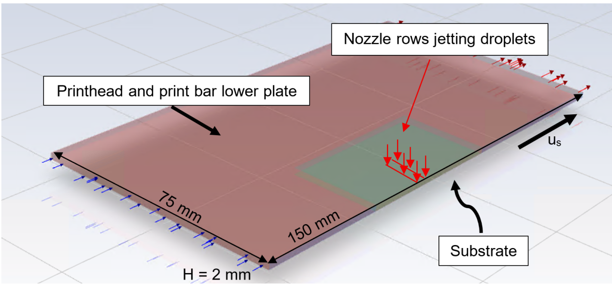


Figure 3: Geometry used in the CFD simulation. A symmetry condition is used to only simulate half of the printhead. Polyhedral mesh is shown in green and the structured part in red.

A Discrete Phase Model (DPM) is used to simulate the droplets and their trajectory. This model enables the coupling of the continuous (air) and discrete phase (ink droplets). It must however be noted that this model is only valid when the discrete phase occupies a small fraction of space, the discrete phase influences the continuous phase by points of force and not fluid displacement.

A method has also been developed to quantify printing quality using the landing positions of each droplet, which is a great way to quantify airflow effects on droplet trajectories and can be precisely monitored during printing. The landing position error is defined as the difference between expected landing position and simulated landing position and is calculated for each droplet. The landing position error of ink droplets can be seen as the sum of all aerodynamic perturbation encountered by the droplets and thus be used to compare different printing setups. Airflow instabilities are clearly visible using this numerical method.

Table 1: Printing parameters.

Physical property	Value	Unit
Printing frequency	4724	Hz
Substrate velocity	0.2	m/s
Ink density	1080	kg/m^3
Drop volume	5	pL
Drop diameter	21.2	μm
Initial drop velocity	6	m/s
Printhead-substrate gap	2	mm

A versatile and automatized printing platform (see Figure 4) is used to generate and visualize the airflow. The platform includes a dual camera system to record two consecutive images, a laser sheet generation system and a fog generation system necessary for the Particle Image Velocimetry. Different printing speeds and printhead-substrate gaps can be configured.

Particle Image Velocimetry (PIV) analysis has been performed by using PIVlab GUI [7]. PIV requires good luminosity and works best with round particles, which is not trivial given the small distance between the printhead and the substrate (2 mm). Often, a choice must be made between a long exposure time, which results in good luminosity but elongated particles, and a shorter exposure time, which results in round particles but some dark areas. In the first case, Particle Tracking Velocimetry (PTV) can provide the velocity of elongated particles and makes it possible to obtain velocity information in areas that would otherwise be poorly illuminated or have significant velocities. With the help of PIV and PTV, the entire experimental velocity field can be measured.

Using a paper substrate fixed on the lower plexiglas sheet, one can also directly observe the printed patterns. The landing drop positions can be measured using a microscope and thus the printing misplacement error can be calculated. The experimental landing positions can be easily compared one to one with the numerical landing positions.

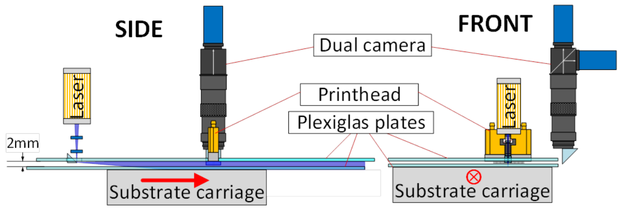


Figure 4: Experimental setup enabling PIV and PTV measurements.

Results

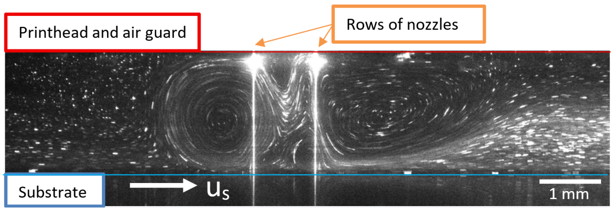
The experimentally observed airflow is visible in Figure 5a, where one recognizes the upstream and downstream vortices, as well as the regions with high and low velocities. The PIV quantitative analysis of the flow field is shown in Figure 5b. Four secondary vortices are also found between the two droplets rows, but they cannot be precisely captured by PIV due to their small size.

The CFD simulation clearly shows similar flow patterns (see Figure 6). To compare them with the experimental data, only the plane in the 3D simulations corresponding to the laser sheet plane in the experiments will be considered.

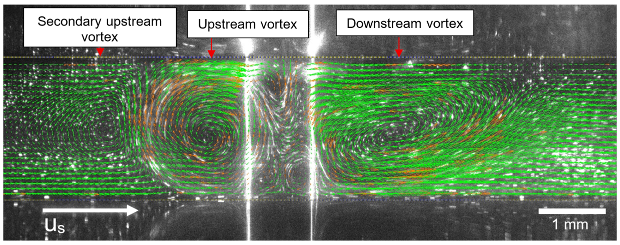
Qualitative comparison shows striking similarities (see Figure 7). The vortices are very similar in structure, size and position. Even the smaller vortices between the two rows of drops appear to be simulated accurately. The numerical upstream vortex has a slightly smaller dimension compared to experimental one. The reason for this discrepancy most likely lies in the uncertainty regarding the volume and velocity of the drops, which were not experimentally verified.

To assess the accuracy of the numerical model, the horizontal velocity profile between printhead and substrate at the eyes of upstream and downstream vortices are used (see Figure 8). One can observe that the velocity profiles are very similar. It must be noted that PIV is not suitable to extract speeds close to walls, where the velocity are dictated by the physical conditions (0 m/s at the printhead and 0.2 m/s at the substrate), as it only provides average speeds for partial areas. It can therefore be concluded that the developed CFD model gives a realistic representation of inkjet printing induced airflow.

The numerical model being experimentally validated, this



(a) Experimental visualization of the airflow in the printing gap around the flying droplets.



(b) PIV analysis of the airflow between the printhead and the substrate.

Figure 5: Experimental airflow measurement. $u_s = 0.2\ m/s$, $F = 4.724\ kHz$, $V_{drop} = 5\ pL$.

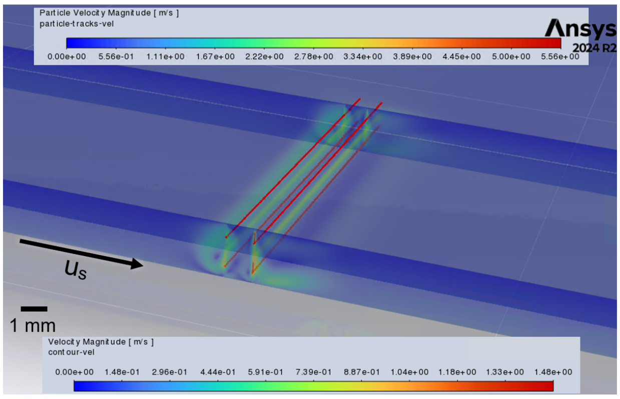


Figure 6: Isometric view of 3D CFD simulation. $u_s = 0.2\ m/s$, $F = 4.724\ kHz$, $V_{drop} = 5\ pL$.

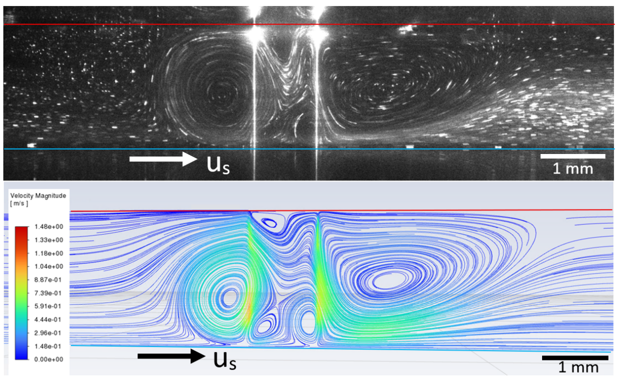
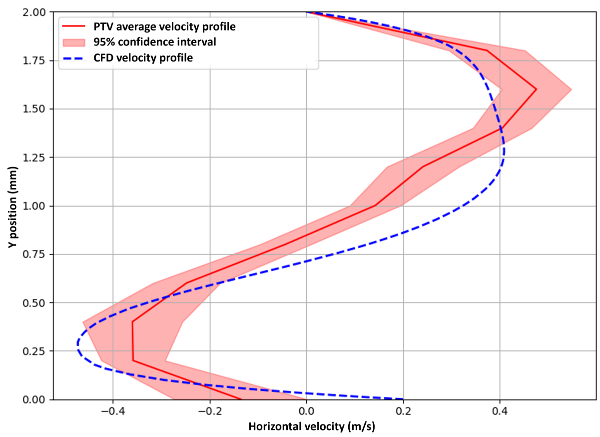
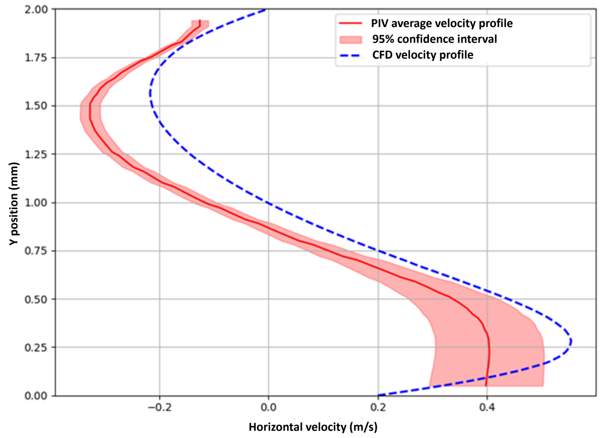


Figure 7: Qualitative comparison between experimental and numerical airflow fields. $u_s = 0.2\ m/s$, $F = 4.724\ kHz$, $V_{drop} = 5\ pL$.

opens the way to deeper studies. One particularly interesting configuration is the effect of the stitching zone (i.e. the area



(a) Comparison of velocity profile in the upstream vortex



0

(b) Comparison of velocity profile in the downstream vortex

Figure 8: Comparison between numerical and experimental horizontal velocity profiles. The profiles are taken on a vertical line between the substrate and the printhead passing respectively by the upstream and downstream vortex eyes. $u_s = 0.2\ m/s$, $F = 4.724\ kHz$, $V_{drop} = 5\ pL$.

where one printhead ends and the second begins), often in a staggered configuration. The EPSON S3200 is used to investigate this case, as it is composed by two staggered double nozzle rows (see Figure 9). This simulation is made with higher substrate speeds ($1\ m/s$) and printing frequency ($24\ kHz$). The drop size is kept to $5\ pL$.

Presence of the second printhead introduces a new constraint to the airflow: the lateral confinement. Air can no longer escape on the sides of the printhead. This has a drastic effect on the upstream vortex, which shrinks dramatically. Furthermore, it has been found that this vortex becomes unstable, leading to spatial oscillations (see Figure 10). This is described as standing wave regime and lead to a corduroy effect [6]. Further eddy destabilization will lead to an unsteady wave and is the root cause of wood grain. The ability to capture this complex phenomenon opens the door to future studies about airflow stabilizations.

Note however that inkjet printing in not synonym with upstream eddy presence, higher substrate speeds with unchanged printing frequency have shown no sign of eddy due to strong Couette flow.

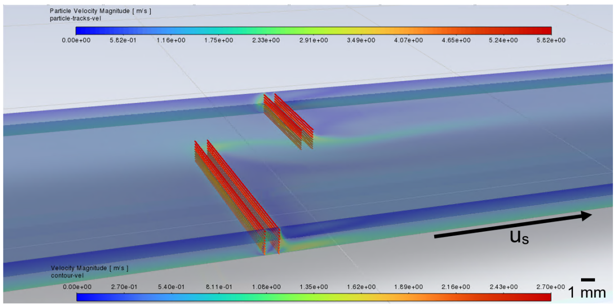


Figure 9: Isometric view of S3200 printhead configuration. $u_s = 1\ m/s$, $F = 24\ kHz$, $V_{drop} = 5\ pL$.

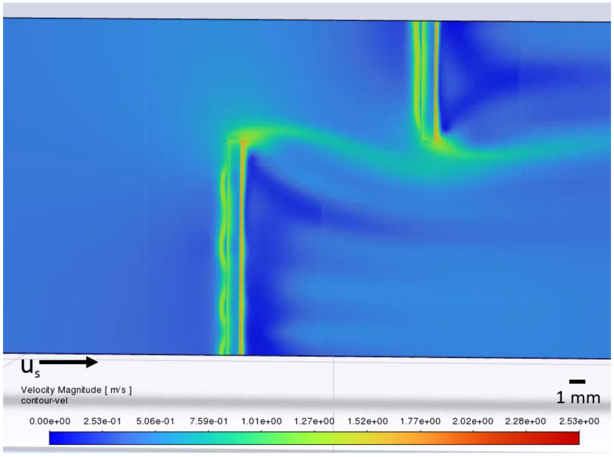


Figure 10: Top view of S3200 printhead configuration. $u_s = 1\ m/s$, $F = 24\ kHz$, $V_{drop} = 5\ pL$.

Conclusions

To mitigate the printing defects inherent to inkjet printing at large gaps, a comprehensive understanding of airflow must be achieved. Numerical simulations can be very helpful in this regard, but their accuracy must first be validated experimentally. For this purpose, a Particle Image Velocimetry system was developed on a research printer. With the aid of a powerful laser, the flow field in the narrow gap between the printhead and the moving substrate could be made visible. Particle Tracking Velocimetry (PTV) has also been used to complete the results. Additionally, the printed patterns could be compared with the numerically obtained droplet landing positions.

The main flow features, namely the upstream and downstream vortices, were clearly visible and well defined. Small discrepancies could be found between the numerical and experimental measurements, notably in the upstream eddy size which is a bit smaller on the numerical simulation. This can be explained by slightly larger or faster experimental droplets, these two printing parameters not having been experimentally verified yet. Smaller structures, such as the four smaller vortices between the two rows of droplets, were also well captured.

Qualitative and quantitative comparison with Computational Fluid Dynamics (CFD) validates therefore the numerical model based on a Discrete Phase Model (DPM). DPM is therefore a powerful tool to use in inkjet simulations, enabling precise parametrizing of the printing parameter and designing new printhead or printer architectures. For instance, the study of the stitching zone has shown that lateral confinement leads to the destabilization of the upstream eddy. This destabilization has been linked to wood grain effect [3, 2, 6]. It is clear that the ability to simulate the underlying cause of wood grain also leads directly to more in-depth investigations of techniques for reducing defects. Stabilizing the upstream vortex should have a direct reducing effect on wood grain and thus on quality defects.

Numerical simulations enable rapid screening of multiple configurations to determine the optimal one. Once the numerical model has been validated, it should be used not only to reduce errors in existing printing configurations, but also in the design of printing systems to maximize printing stability from the outset. A printing system developed with the help of CFD analysis will result in fewer printing errors and thus save time and money. Unlike experimental conditions, numerical simulations are not limited to simple geometries, but also allow the investigation of complex, non-planar geometries, and provide the flow information in the whole fluid volume, leading to a better understanding of the causes of printing errors.

The simulation framework developed is now being used to further investigate airflow problems in inkjet printing.

References

[1] Rodriguez-Rivero, C., & Daly, R. (2019). Aerodynamics of direct-to-shape printing. The Inkjet Conference. Düsseldorf.

[2] Rodriguez-Rivero, C., Castrejón-Pita, J. R., & Hutchings, I. M. (2015, January). Aerodynamic effects in industrial inkjet printing. In NIP & Digital Fabrication Conference (Vol. 2015, No. 1, pp. 29-38). Society for Imaging Science and Technology.

[3] Dan Barnett & Marlene McDonald, Fujifilm Dimatix. (2014). Evaluation and Reduction of Elevated Height Printing Defects. Digital Fabrication and Digital Printing: NIP30 Technical Program and Proceedings

[4] Castrejón-Pita, A. A., Betton, E. S., Campbell, N., Jackson, N., Morgan, J., Tuladhar, T. R., . . . Castrejon-Pita, J. R. (2021). FORMULATION, QUALITY, CLEANING, AND OTHER ADVANCES IN INKJET PRINTING. Atomization and Sprays, 31((4)).

[5] Aquino, A., Mallinson, S., McBain, G. D., Horrocks, G. & Barber, T. (2020). Two-dimensional numerical simulation of inkjet print-zone flows. 22nd Australasian Fluid Mechanics Conference.

[6] Aquino, A., Mallinson, S., McBain, G. D., Horrocks, G., Barber, T. & de Silva, C. (2024). The evolution of print defects in inkjet printers at elevated print gap height

[7] Thielicke, William; J. Stamhuis, Eize (2019). PIVlab - Time-Resolved Digital Particle Image Velocimetry Tool for MATLAB.

Author Biography

Luc Nussbaumer studied mechanical engineering at the University of Applied Sciences and Arts Western Switzerland (HES-SO) in Fribourg. He joined the iPrint Institute in 2025 to continue his research on airflow simulation in inkjet printing, following his work on the subject conducted during his Bachelor's thesis.

Johannes Renner studied mechatronics with specialization in automation technology at the Secondary Federal College of Engineering in Vienna, and mechanical engineering as well as mechatronics at the Bern University of Applied Sciences, where he worked after graduation in 2007 as scientific assistant and later on as scientific officer for the Institute of Print Technology. In 2013 he joined the iPrint institute. Johannes has gained over 15 years of experience in applied research with the focus on inkjet technology and is contributing to many of iPrint's inkjet-related projects.

Gioele Balestra holds a Master's degree in Mechanical Engineering from ETH Zurich and a PhD from EPFL. He joined iPrint in 2019 to *develop new digital deposition technologies and processes and since July 2020 he is in charge of the applied research and educational activities of iPrint.*