Ink drop motion in wide-format printers
II. Airflow investigation

N. Link¹, S. Lampert², R. Gurka², A. Liberson², G. Hetsroni², R. Semiat³,*

¹ Wolfson Faculty of Chemical Engineering, Technion-Israel Institute of Technology, 32000 Haifa, Israel
² Faculty of Mechanical Engineering, Technion-Israel Institute of Technology, 32000 Haifa, Israel

Article history:
Received 3 August 2007
Received in revised form 11 February 2008
Accepted 12 February 2008
Available online 23 February 2008

Keywords:
Ink jet
PIV
Airflow
Drops motion

A B S T R A C T
Quality ink jet printing is being used increasingly for many types of applications. This paper investigated the relative motion of the printing head, which induces airflow between the head and the printing media. This airflow may interfere with the ejected ink drops as they fly towards the media, resulting in printing inaccuracy. The airflow was investigated experimentally (particle image velocimetry, PIV). Since the crucial areas in the flow field were difficult to measure experimentally, a CFD simulation program was used to complete the understanding of the flow in order to determine the limitations for quality printing. Results show only a small deviation from a linear shear flow, which can be attributed to experimental error. Furthermore, the results show that it is possible to increase the head–media relative motion considerably without causing significant turbulence followed by flow interference that may deteriorate quality prints.

© 2008 Elsevier B.V. All rights reserved.

1. Introduction
Commercial digital printing applications have expanded in recent years and have branched beyond the classic imaging industry. Industries such as printed circuit boards (PCB) [1,2], pharmaceutical research [3,4], textile [5–8] and even food decorating, are using the digital printing technique as a major working tool.

The improvement in product quality, expressed by the accuracy of the printed image, is in constant demand. One aspect of the printing process is the motion of the printing head in different directions relative to the printing media. Printing machines may use up to hundreds of printing heads that move under acceleration from rest or at a constant velocity relative to the media. This motion causes airflow between the printing head and the printing media—a gap of 2 mm for most [Fig. 1], which may interfere with the drop motion towards the target and lead to inaccuracy in the printed image due to flow disturbances.

The aim of this paper is to examine the flow created between the printing head and the printing media and then implicate the results on the ink drops. This article follows and complements the authors’ previous article dealing with the motion of drops towards the printing media [9]. The two articles deal with wide-format printing, where usually the target or the printing heads are moving at a constant relative speed.

The relative motion of printing heads along the printing media can be interpreted as a set of obstacles in the flow. Therefore, the expected laminar Couette-type of flow may create a vortex next to the printing media followed by flow fluctuations. The printing media itself may also contain some obstacles, for example, in PCB uses. No information can be found in the literature regarding this special case, so it is important to start by comparing this type of flow to similar flow solutions.

The current case involves a velocity field above a plate created by the plate’s own movement (the printing media), hence a boundary layer is built. The relative velocities involved in the printing machine are in the order of 1–2 m/s. In this paper, higher velocities (up to 20 m/s) were considered as well, in order to start a discussion on the possibility of increasing printing speed. The calculated Reynolds numbers and boundary layer thickness based on [10] are detailed in Table 1.

Since the gap between the printing head and the printing media is less than $2 \times 10^{-3}$ m, the boundary layer encounters a sudden contraction. Vortices are expected close to the plates and hence may influence the ink drops as they are ejected from a nozzle and reach the printing media.

The issue of a vortex created by a moving wall was studied by Allen and Auvity [11] and Allen and Naitoh [12] for a different application—a piston/cylinder geometry. In this geometry, the fluid moving above the cylinder encounters a sudden disturbance...
created by the stationary wall. The experiments showed the immediate creation of a vortex that increased with time.

2. Theory

The velocity field’s behavior and development and the vorticity gradient along the area considered are investigated. The turbulence intensity generated should give an estimation of possible disturbances in the flow field that could lead to an incoherent pattern of the ejected drops. Each of the above properties was checked by an experimental method (particle image velocimetry, PIV) and a simulation (CFD) program, both of which will be detailed below.

Laminar parallel flow is a known exact solution of the Navier–Stokes equation. A simplification of this equation was made for two-dimensional flow through a straight channel [10]:

\[
\frac{dp}{dx} = \frac{\mu}{h^2} \frac{d^2u}{dy^2}
\]

(1)

where \( p \) is the pressure, \( \mu \) is the fluid viscosity and \( u \) is the \( x \)-direction velocity.

When one of the walls is at rest while the other moves at velocity \( U \), the solution obtained is

\[
u = \frac{y}{h} U - \frac{h^2}{2\mu} \frac{dp}{dy} \left( 1 - \frac{y}{h} \right)
\]

(2)

where \( h \) is the channel height. This velocity profile should be a simple linear profile when the pressure gradient is zero (known as Couette flow). If the pressure gradient is not zero, the solution gives a deviation from linearity.

The span-wise vorticity \( \omega_z \) is defined [12] as the measure of rotation of a fluid element as it moves in the flow field:

\[
\omega_z = \frac{1}{2} \left( \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right)
\]

(3)

where \( v \) is the velocity in the wall normal direction.

Vorticity cannot be created or destroyed in the interior of a homogeneous fluid under normal conditions, and is produced only at the boundaries [13,14]. Hence, it is more useful to describe flows in terms of vorticity distribution. The net viscous force on an element of fluid is determined by local vorticity gradients. When the viscosity is small, the net viscous force is significant only in places where the vorticity gradients are large; if the vorticity is zero over the flow region, viscous stresses make no contribution to the net force on elements of fluid and may be ignored.

Table 1

<table>
<thead>
<tr>
<th>Relative velocity (m/s)</th>
<th>Re number</th>
<th>Flow regime</th>
<th>Boundary layer thickness, ( \delta ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>33,425</td>
<td>Laminar</td>
<td>6.84</td>
</tr>
<tr>
<td>10</td>
<td>334,250</td>
<td>Transition</td>
<td>7.27*</td>
</tr>
<tr>
<td>20</td>
<td>668,500</td>
<td>Turbulent</td>
<td>6.33</td>
</tr>
</tbody>
</table>

* Calculated by turbulent correlation.

Turbulence intensity (TI) is an index of the extent of turbulence in the flow and is calculated by

\[
TI = \sqrt{u'\text{e}^2 + v'\text{e}^2} \quad \text{(4)}
\]

where \( u' \) and \( v' \) are the instantaneous velocity fluctuations and \( \overline{U} \) is the mean velocity in the flow field.

In order to determine whether the vortices created in the flow field due to the movement of the printing media influence the ink drops and hence the printed image quality, the phenomenon was investigated by an experimental method and also modeled by a CFD program.

3. Experimental

Due to various limitations, the experiments could not be conducted on real printing systems. The experimental setup included a custom-made conveyer that was built in smaller dimensions (520 mm × 25 mm) than a real printing machine, as shown in Fig. 2. A paper belt representing the printing media was rotated by the conveyer at a given velocity. The velocity was controlled by a geared engine (1–2 m/s). The printing head (dimensions: 185 mm × 39 mm × 37 mm) was positioned above the printing media at a chosen distance to the media (2–4 mm). The velocity field was measured between the printing head and the printing media, approximately at the center of the printing head length.

A PIV system was used containing a double-pulsed Nd:YAG laser emitting 170 mJ/pulse at 15 Hz, and a 1k × 1k 30 fps CCD camera, where a single pixel area was 9 \( \mu \text{m} \times 9 \mu \text{m} \) producing an eight-bit digital signal to the video output. The camera was mounted with a suitable lens, therefore, the image size recorded at a distance of 94 mm from the test section was 10 mm × 10 mm. The frame rate of the CCD camera was adjusted to the pulse rate of the lasers (15 pairs of images per second). Incense smoke was used for the flow seeding. The calibration procedure and PIV cross-correlation analysis were performed using the Insight 5 computer program.

The measurement procedure was controlled using the software that enabled operating the lasers and synchronizing them with the camera. The time between pulses was determined for each experiment in the software (25–60 \( \mu \text{s} \)), and a batch of 400 images was acquired for each set of experiments. After acquiring the images, a post-process analysis was done, which included calculating the velocity field for every pair of images and validating the calculated velocity field using several filters in order to remove false vectors. The velocity calculation is based on a spatial correlation function that operates on a chosen area. The cross-correlation function is computed using fast Fourier transform (FFT). The cross-correlation operation was done in an interrogation area of 32 × 32 pixels (for...
4. Experimental PIV results and discussion

As detailed previously, the printing medium moves at a constant velocity. The velocity used in commercial printing machines is usually between 1 and 2 m/s; therefore, these velocities were examined.

The gap between the printing media and the printing head in printing machines must be as small as possible—usually not exceeding 2 mm. It was very difficult to use a gap less than 2 mm in the experiment due to the physical constraints of the laser light sheet penetration.

An example of the images acquired by the PIV system is shown in Fig. 3. Since the gap between the printing head and the printing media is very small, the smoke particles used for seeding the flow did not penetrate the gap entirely. The flow close to the printing head was immeasurable due to a poor seeding of particles along the printing head walls and to the lack of laser light penetration, as can be seen in the picture. At the bottom of the flow volume, the laser light reflection from the printing media was too intense to facilitate a good measurement. Hence, the measurement area did not occupy the entire flow area but was large enough for a good understanding of the entire flow region.

Fig. 4 shows the mean velocity profile and mean velocity magnitude map within the experimental area when the velocity of the media is 1 m/s. Since the experimental results were feasible only in the centered flow, a linear approximation was performed for the theoretical velocity profile. The experimental velocity profile obtained was found to be very close to linear at the center, as for a standard Couette flow (Fig. 4(a)). The experimental velocity profile shown is the average profile calculated from about 350 vector maps.

A similar picture was obtained for a conveyor velocity of 2 m/s, as shown in Fig. 5. Here, some fluctuations of the average velocity can be seen along a theoretical straight line, implying that the pressure gradient within the gap was not zero. This may be caused either because the paper belt stitch created interference, or because the printing head and the printing media were not perfectly parallel to each other, even though immense efforts were made to avoid this. It is also important to note that the accuracy of media motion in the experimental system is lower than in the actual industrial system.

Turbulence intensity (TI) and velocity field for 1 and 2 m/s average flows of the printing media in the flow direction are shown in Fig. 6. The velocity vectors shown in Fig. 4(b) increase in intensity, as expected, towards the media. Since the vertical velocity fluctuations are much smaller than the horizontal ones, they may be neglected in the calculation, and Eq. (4) becomes:

\[ \text{T.I.} = \frac{\sqrt{\langle u'\rangle^2}}{U} \]  

(5)

For the 1 m/s media velocity (Fig. 6(a)), turbulence intensity gradually increases towards the media. For the 2 m/s velocity (Fig. 6(b)), turbulence intensity is less ordered; high values exist in the upper end (close to the printing head) and again very close to the printing media. The values of TI for both cases are in the same range (0–33%). The reason for the high values of the TI close to the printing media is due to the intermittent disturbance caused by the belt stitch that could not be eliminated.

Fig. 7 exhibits the average span-wise vorticity in the field at a velocity of 2 m/s. Within the measurable area at both velocities (1 and 2 m/s), the span-wise vorticity magnitude gradient is low and close to zero. This is expected because vorticity gradients should appear near the walls where measurements are not feasible.

Since the crucial areas in the flow field were difficult to measure experimentally, a CFD simulation program was used to complete the understanding and is detailed in the following section.

4.1. Numeric simulation of the airflow

This section describes a numeric simulation of the airflow in the system described earlier. The commercial CFD package FLUENT 6.1 was used for this purpose. FLUENT uses a control volume-based technique to convert the governing equations (Navier–Stokes and continuity) to algebraic equations, which can be solved numerically. This control volume technique consists of integrating the governing equations...
equations, yielding discrete equations that conserve each quantity on a control volume basis.

A two-dimensional geometry of the problem was outlined, as illustrated in Fig. 8. The printing head is shown at the center of the control area used by the program. The lowest horizontal wall represents the moving media. The numbers identify the different boundaries. The boundary condition definitions are given below.

The two-dimensional mesh chosen for the numeric calculations is illustrated in Fig. 9, which also shows the differences between the local mesh densities created in order to improve calculation accuracy in these specified areas.

The goal of the simulation was to evaluate the turbulence level within the gap and the area next to the printing media and to determine whether it influences drop trajectory. Since the flow is laminar, a prediction of the vortices is difficult hence an estimation of the maximum turbulence was taken.

The Reynolds stress model (RSM) for turbulence was applied here with enhanced wall treatment, which adds five transport equations to the model calculation for a more accurate solution of the flow.

The boundary conditions for each boundary are detailed below, based on Fig. 8.
1—Pressure inlet: The constant pressure inlet boundary condition was used to define the fluid pressure at the flow inlets. This is suitable for both incompressible and compressible flow calculations. The pressure inlet boundary condition can also be used to define a “free” boundary in an external or unconfined flow.

The definition of pressure in FLUENT includes the hydrostatic head. This definition allows the hydrostatic head to be taken into the body force term and excluded from the pressure calculation when the density is uniform.

The treatment of pressure inlet boundary conditions can be described as a loss-free transition from stagnation conditions to inlet conditions. For incompressible flows, this is accomplished by applying the Bernoulli equation at the inlet boundary. When flow enters through a pressure inlet boundary, it is considered as the total pressure of the fluid at the inlet plane, \( p_0 \). In incompressible flow, the total inlet pressure and the static pressure, \( p_s \), are related to the inlet velocity via Bernoulli’s equation:

\[
p_0 = p_s + \frac{1}{2} \rho v^2
\]

With the resulting velocity magnitude and the flow direction vector at the inlet, the velocity components can be computed. The inlet mass flow rate and fluxes of momentum and energy can then be computed.

2 & 3—Pressure outlet: Pressure far-field conditions are used to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. A set of “back flow” conditions is also specified to be used if the flow reverses direction at the pressure outlet boundary during the solution process. The relevant values are similar to those entered for pressure inlet.

When the flow enters the domain at an inlet, outlet or far-field boundary, FLUENT requires specification of transported turbulence quantities—turbulent intensity and turbulence length scale. Turbulence intensity of 1% or less is generally considered low; turbulence intensities greater than 10% are considered high.

The turbulence intensity at the core of a fully developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows:

\[
TI = 0.16(Re_{Dh})^{-1/8} = 0.0868 = 8.68\%
\]

where \( D_h \) is the hydraulic diameter. Therefore, the turbulence intensity input was 10%.

The turbulence length scale, \( L \), is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows. In fully developed duct flows, \( L \) is restricted by duct size since the turbulent eddies cannot be larger than the duct. An approximate relationship between \( L \) and the physical size of the duct is

\[
l = 0.07L
\]

where \( L \) is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully developed turbulent pipe flow, where \( L \) is the pipe diameter. In a channel of non-circular cross-section, \( L \) can be based on the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on duct size. Hence, \( L \) was taken as 2 mm—the gap between the printing head and the printing media. As a result, \( L \) equals \( 1.4 \times 10^{-4} \) m.

4 & 5—Wall boundary conditions: Wall boundary conditions are used to limit fluid and solid regions. In viscous flows, the non-slip boundary condition is enforced at walls by default.

Boundary #4 is a moving wall simulating the printing media’s movement. Hence, momentum data are required. A relative translational speed is entered at various velocities. A non-slip condition is set and the wall roughness height and constant are fixed as a smooth wall. Wall roughness affects drag (resistance), as well as heat and mass transfer on the walls.

Boundary #5 is a stationary wall representing the printing head. A non-slip condition is applied.

5. Numeric simulation results and discussion

This section presents the main results of the numeric simulations of flow in the gap between the printing head and the printing medium. As mentioned earlier, the printing medium moves at a constant velocity. The velocity used in commercial printing machines ranges from 1 to 2 m/s. The CFD simulation accounted for these velocities, as well as for greater velocities, between 5 and 20 m/s for comparison. The reason for the higher velocities is for estimating the flow parameters in the cases of faster printing machines.

The expected result of the calculations is the velocity field. Fig. 10 shows the velocity magnitude within the simulation area at a media velocity of 1 m/s. The velocities near the printing media and the printing head are shown to be equal to the condition speed (1 and 0 m/s); a linear profile was shown to exist, as was also seen in the experimental results. The velocity reduces to a non-slip condition on both sides. Compared to the experimental results, Figs. 4 and 5 exhibit a very similar behavior, except for the explained experimental fluctuations around the straight line. This is in good agreement since the model does not include the flow disturbance on the moving media. The flow inside the gap looks like a flow profile between a stationary and a moving wall. Outside this area, the flow field...
changes significantly—the flow direction moves up and backwards. This can be seen more clearly in the vector map in Fig. 11, yet was not measured in the experiments. The disturbances outside the region of the printing head disappear inside the gap between the media and the printing head, so their possible influence on drop motion is negligible.

Fig. 12 represent the velocity field for the extreme situation when the relative motion between the media and the printing head is 20 m/s. This high velocity was chosen in order to investigate the possibility of significantly reducing printing time, neglecting at this stage the drop velocities and their generation frequencies. Fig. 12(a) presents the calculated velocity field in front of the printing head. Even at this high velocity, the flow seems to be calm, especially in the gap between the media and the printing head. Fig. 12(b) shows the calm development of the boundary layer behind the printing head that may be fed into the successive printing head located down the flow.

Turbulence intensity (TI) and vorticity levels were calculated within the gap for the various velocities and are presented in Figs. 13 and 14. Fig. 13 shows that the turbulence intensity at a printing media velocity of 1 m/s is in the 3–13% range and varies throughout the gap. The TI calculated for 1 m/s is lower than the experimental result (0–33%) for the same velocity. This was explained earlier due to the existence of a stitch in the media used in the experiments that caused a jump and changed the ideal field considered in the simulation. Nevertheless, this TI is considered to be strong and causes vortices. When the printing media velocity increases, the TI increases as well, as can be seen in Fig. 14. The TI in the gap at 10 m/s is around 110%, which is considered very high.

Vorticity magnitude was also checked, as shown in Fig. 15. The vorticity is another parameter calculated for the existence of vortices that can influence drop trajectory. The vorticity at 1 m/s is very low in the simulation area except for the gap where the vorticity equals around 500 Hz. Furthermore, the vorticity near the walls (printing head and printing media) is higher. The difference is around 200 Hz. At 20 m/s, the vorticity is much higher—about 3500 Hz. It is very high near the walls, around 19,000 Hz. The gradient is around 15,000 Hz. It is difficult to compare these values with the experimental results since high vorticity is expected close to the walls where no measurements could be taken. The experimental calculated vorticity at the center was lower than the calculated one.

5.1. Trajectory calculations

In order to examine the effect of these vortices on the ink drops ejected from the printing head, a particle tracking mode was used in the simulation. FLUENT enables simulating a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets) dispersed in the continuous phase. The trajectories of these discrete phase entities are calculated. The coupling between phases and its
impact on both the discrete phase trajectories and the continuous phase flow can be included. The particles were defined as drops, 20 μm in diameter with a density of 1000 kg/m³, which were injected as in the printing head.

5.2. Equations of motion for particles

Particle force balance is calculated by integrating the force on the particle, which is written in a Lagrangian reference frame. This force balance equates the particle inertia with the forces acting on the particle and can be written (for the x direction in Cartesian coordinates) as

\[
\frac{du_p}{dt} = F_D(u - u_p) + \frac{g(p_p - \rho)}{\rho_p} + F_x
\]  

(10)

where \( F_D(u - u_p) \) is the drag force per unit particle mass and

\[
F_D = \frac{18\mu}{\rho_D D_p^2} \frac{C_D Re'}{24}
\]

(11)

Here, \( u \) is the fluid phase velocity, \( u_p \) is the particle velocity, \( \mu \) is the molecular viscosity of the fluid, \( \rho \) is the fluid density, \( \rho_p \) is the density of the particle, and \( D_p \) is the particle diameter. The drag coefficient, \( C_D \), is taken from a proper correlation.

\( Re' \) is the Reynolds number, defined as

\[
Re' = \frac{\rho D_p |u_p - u|}{\mu}
\]

(12)

While Eq. (10) includes a force of gravity on the particle, it is important to note that the default gravitational acceleration is zero since it is negligible in these cases.

Eq. (10) incorporates additional forces (\( F_x \)) in the particle force balance that can be important under special circumstances. The first of these is the “virtual mass” force, the force required to accelerate the fluid surrounding the particle and is important when \( \rho > \rho_p \), which is not the case here. An additional force arises due to the pressure gradient in the fluid:

\[
F_x = \left( \frac{\rho}{\rho_p} \right) u_p \frac{\partial u}{\partial x}
\]

(13)

In the stochastic tracking approach, FLUENT predicts the turbulent dispersion of particles by integrating the trajectory equations for individual particles using the instantaneous fluid velocity along the particle path during the integration. By computing the trajectory in this manner for a sufficient number of representative particles (termed the “number of tries”), the random effects of turbulence on the particle dispersion may be accounted for. The discrete random walk (DRW) model is used in which the fluctuating velocity components are discrete, piecewise constant functions of time. Their random value is kept constant over an interval of time given by the characteristic eddies lifetime.

Prediction of particle dispersion makes use of the concept of integral time scale, which describes the time spent in turbulent motion along the particle path. The integral time is proportional to the particle dispersion rate, as larger values indicate more turbulent motion in the flow.

The results are shown in Fig. 16 at two different velocities. At low velocities, the particle tracking shows an invisible, very small shift in the particle’s trajectory. The shift in drop trajectory was also calculated. At a media velocity of 1 m/s, the shift is 13 × 10⁻⁴ m, which is negligible and within the error radius of the particle itself.

As the printing media velocity increases, the shift in drop trajectory increases. The shift in particle trajectory computed for the 5 m/s media velocity is 75 × 10⁻⁶ m, which is still negligible, especially since all of the drops have the same shift. The shift starts to become significant at 10 m/s, where it reaches a value of 163 μm. At a media velocity of 20 m/s, the shift is 380 μm. These results are obtained at the relatively high turbulence reported, and the vorticity at higher velocities.

Since the drop trajectory is only shifted and the vortices do not influence it, the printing process may be performed at high media velocities based on the above analysis. The printed picture will be accurate according to this simulation.

Another way to determine whether the vortices influence drop trajectory or not is an energy calculation. The kinetic energy per unit mass at the breakup from the nozzle can be calculated by

\[
E_p = \frac{1}{2} \frac{v_p^2}{24} \approx 110 \text{ m}^2/\text{s}^2
\]

(14)

This can be compared to the turbulence kinetic energy per unit mass of air that was calculated by the simulation. This quantity equals between 2.38 × 10⁻² m²/s² for the 1 m/s simulation and 2.64 m²/s² for the 20 m/s simulation. Hence, only a minor influence of airflow turbulence on the ink drop trajectory can be expected.

6. Conclusions

A comprehensive investigation was made regarding the influence of the airflow created by printing head–media movement. Both experimental and simulation results show that the effect is minor. A small shift exists in the printed image at the most that does not decrease printed image quality. A good agreement between the experimental and calculated velocities was found. Lower agreement was found regarding turbulence intensity and vorticity. The calculated analysis can complete the understanding where the experiments were difficult to perform.

It was shown that is possible to increase significantly the relative motion up to 20 m/s and more where the flow becomes turbulent without causing significant turbulence, followed by flow interference that may deteriorate printing quality.

References