Novel Microfluidic Jet Deflection:
Significant Modeling Challenge with Great Application Potential

Christopher N. Delametter, David P. Trauernicht, James M. Chwalek

Electronics Imaging Products, Research & Development,
Eastman Kodak Company, Rochester, New York 14650-2121
Email: christopher.delametter@kodak.com

ABSTRACT

A novel method is described for deflecting a liquid jet emanating from an orifice having a diameter of approximately ten microns. While awkward mechanical valves or unwieldy electrodes can be used to deflect a jet, the new method relies on asymmetrically heating a portion of the orifice producing the jet. Embedded in the orifice structure, the orifice and heater are fabricated employing precise techniques used in the manufacture of silicon-based microelectronic components. The jet deflection phenomenon is probed experimentally and investigated theoretically using a computational fluid dynamics approach. It is shown that the jet deflection phenomena stem from three highly coupled competing effects. Two of these are related to the variation of liquid surface tension with temperature and the third to that of viscosity variation [1]. The former manifest themselves as stresses that act normal as well as being tangent to the fluid interface, the tangential component being Marongoni stress that often plays a role in small scale thermocapillary flows [2].

Keywords: jet deflection, thermocapillary, inkjet, printhead, computational fluid dynamics (CFD), volume of fluid (VOF), and surface tension.

1 INTRODUCTION

In this paper, we examine the phenomena of thermal steering of liquid microjets. This novel steering mechanism has been employed to develop a new approach to continuous inkjet printing, replacing the more traditional electrostatic deflection technique. The printhead, described in this paper, is produced using CMOS compatible MEMS fabrication technology, and requires only very low operational energy expenditure, avoiding many of the limitations of the current technology, while retaining the benefits of high productivity printing with superior image quality.

Figure 1(a) shows a top view of a single nozzle of this new printhead device. It consists of an orifice in a thin nozzle plate with two semicircular heaters on either side. The region behind the nozzle plate is a relatively large chamber that holds pressurized fluid. Shown in Figure 1(b) is a section of an array of nozzles. When heat is applied to one of the heaters, the microjet (which would be flowing out of the page), deflects away from the heated side, thus out of the plane formed by the array of jets.

To gain a better understanding of the underlying jet deflection mechanisms, the phenomenon is studied using a computational fluid dynamics (CFD) software package that utilizes a finite difference volume of fluid (VOF) approach for tracking the free surface interface. These results are compared to a variety of experimental conditions, fluids, and geometry's.

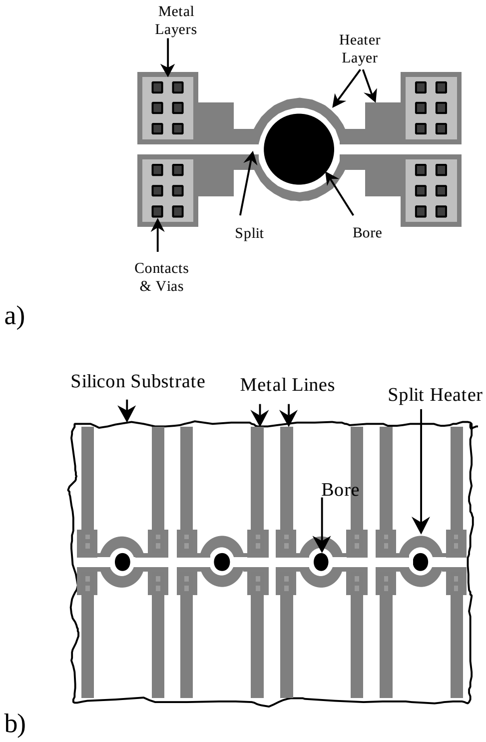

Figure 1: (a) Top view of a single nozzle split heater configuration. (b) A section of an array of nozzles showing the heater orientation to the plane of jets.

2 PRINCIPLE OF OPERATION

The cross-sectional view in Figure 2 is a diagram of a single nozzle in operation. An ink delivery channel and nozzle bore are etched in a substrate, which is single-crystal silicon for the initial devices. The ink delivery channel is pressurized, and liquid streams are formed. At a given distance above nozzle bore, the stream breaks into droplets, primarily because of the modulation of surface tension because of heat generated by the heater element. The energy supplied to the heater is in the form of an electrical pulse train. To achieve synchronous drop breakup, only a very small amount of energy is needed (<1 nJ). The drop size may be precisely controlled by the delay time between the heater pulses, (in conjunction with the orifice diameter and ink flow rate, which is controlled by the applied pressure). This thermally stimulated Raleigh breakup of a liquid jet has been studied in detail and is described by Furlani, et al. [3].

The heater is isolated from the substrate by a thermal and electrical insulating layer, which minimize heat loss to the substrate. The nozzle bore is defined by etching this insulating layer. Further details of operation and fabrication can be found in several references [4–6].

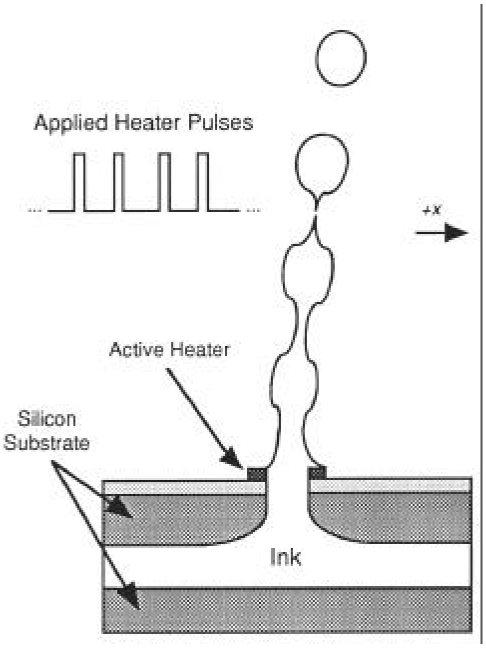

Figure 2: Schematic cross-sectional view of a nozzle

3 SIMULATION DETAILS

While the experimental work offers invaluable insight into the phenomenon, there are limitations in the experimentation because of the small size of such devices. Quantitative measurements of the velocity and temperature gradients in the region of the orifice are difficult if not impossible. Therefore, to extend our understanding of the jet deflection phenomenon, we performed a series of simulations using a CFD code. The FLOW-3D software is a commercially available finite difference code for solving the complete Navier-Stokes equations using the VOF method [7, 8]. The capabilities of the code include three-dimensional, time dependent, coupled fluid/structure heat transfer and free surface tracking of complex nonlinear flows. A brief summary of the mathematical formulation for describing the motion of a laminar incompressible fluid flow is given below. The governing equations solved are the equation of mass continuity,

$$\nabla \cdot \mathbf{v} = 0 \tag{1}$$

the Navier-Stokes equation,

$$\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla)\mathbf{v} = \mathbf{g} - \frac{1}{r}\nabla p + \frac{m}{r}\nabla^2 \mathbf{v} + \frac{s}{r}\left(\frac{1}{R_1} + \frac{1}{R_2}\right)\nabla F \tag{2}$$

and the continuity equation for the VOF function *F*,

$$\frac{\partial F}{\partial t} + (\mathbf{v} \cdot \nabla)F = 0 \tag{3}$$

where **v** is the velocity vector, *t* is the time, **g** is the gravitational acceleration (which can be neglected at these scales), *p* is the scalar pressure, *m* is the viscosity, σ is the surface tension, and R_1 and R_2 are the principal radii of curvature of the free surface. The VOF function *F* is defined to be unity in any computational cell fully occupied by fluid, zero in any empty cell, and between zero and unity in any cell containing a free surface. Whereas *F* moves with the fluid, this function satisfies the continuity equation. In equation (3), the gravitational, viscous, and surface tension effects are explicitly included in the model. This implementation of the VOF method is described by Hirt and Nichols [7]. When supplied with a geometry definition, boundary and initial conditions, equations (1)-(3) are solved by the code utilizing a finite difference approach.

Because the jet deflection problem is cylindrical in nature (yet inherently nonsymmetrical with the application of heat to one side), a comprehensive model would require a full three-dimensional numerical solution. The computational cost of such a simulation is extremely high. therefore, a two-dimensional planar geometry is adopted and has been shown, through experimental verification, to capture the essential features of the jet deflection phenomenon.

An example of the computational cell definition used, to study a single orifice, consists of 150 cells in the cross-jet (sheet) direction (*x*), and 300 cells in the jet axial direction (*z*). It extends from $-15.0 < x < 15.0$ *m*m and $0 < z < 50.0$ *m*m. Steady state power is uniformly apply to the polysilicon heater structure in the model. Figure 3 shows a typical output from the simulations. Energy diffuses through the surrounding oxide/nitride layers to the fluid structure boundary where it diffuses into and is advected away in the fluid as it exits the orifice and forms the jet. The physical properties of the fluid include viscosity and surface tension, which both vary with temperature, as well as density, thermal conductivity, specific heat, and contact angle.

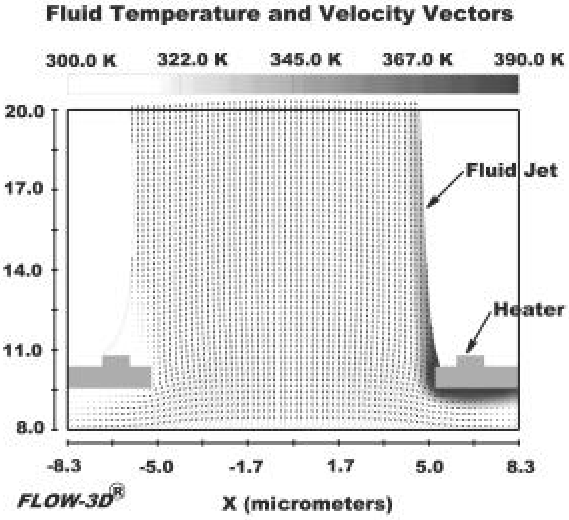

Figure 3: Cross-sectional view of 2-D model geometry.

4 SIMULATION RESULTS

Figure 4 shows a comparison of some initial modeling results with the experimental data for water and 2-propanol (IPA). The correspondence is very good for water, and quite reasonable for IPA in that the general behavior and trend with velocity is captured. In fact, a variety of fluids, operational conditions, and geometries have been modeled and experimentally evaluated and the two-dimensional modeling approach described above provides data that trend very well with the experimental data. Some slight discrepancy can be expected because the model is two-dimensional where the experimental device is three-dimensional with some inherently three-dimensional behavior.

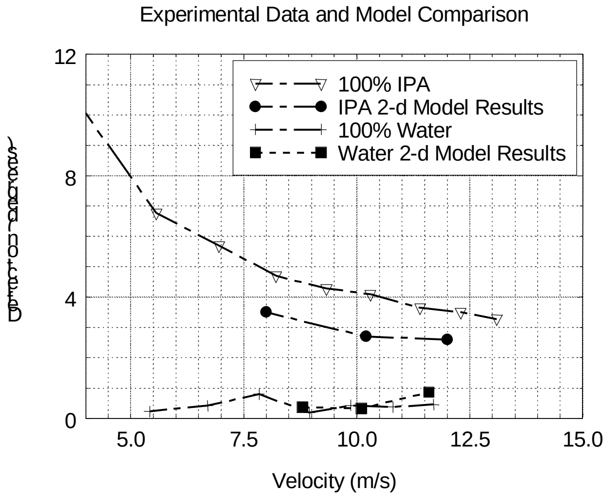

Figure 4: 2-D model and experimental results.

A complete understanding of the jet deflection phenomenon is complex because of its highly coupled nature, the large number of parameters involved, and its sensitivity to geometry and geometric features. The simulations have shown that jet deflection for a given geometry is primarily controlled by the competition between three governing effects. Viscosity variation with temperature causes a momentum imbalance across the exiting jet resulting in deflection away from the heated side. Similarly, variation in surface tension with temperature and the resulting gradients cause changes in the local mean curvature as well as generating surface flow as a result of Marongoni stresses (flows tangent to the free surface). We have shown that both of these effects play a role in jet deflection as well. In addition, the surface tension and viscous effects described above cause the fluid meniscus (on the heated side of the orifice (see figure 3)) to pull in towards the jetting fluid which further enhancement deflection.

Additionally, the simulations have shown that the viscous effect plays a more significant role in low surface tension fluids. The simulations have also shown that the surface tension effect(s) play a somewhat larger role with the higher surface tension fluids (such as water and water mixtures).

5 EXPERIMENTAL DETAILS

We image the streams of fluid in a time-resolved manner using a strobe light and camera system. The video signal output of the CCD camera is captured using frame grabber in a personal computer. The fluids are filtered through a 0.45-µm pore filter in the supply line just before entering the printhead. Compressed nitrogen gas is used to pressurize the fluid supply container under the control of a pressure regulator. The pressurized fluid forms a cylindrical jet exiting the nozzle bore. We vary the velocity of the jets by adjusting the pressure in the fluid supply container.

The heaters are driven with voltage pulses from an arbitrary waveform generator that is subsequently amplified to provide the necessary current. For these experiments, the repetition rate for the application of a series of heat pulses was about 10 Hz (the frame rate chosen for our camera system). Shown in Figure 5(a) are images of a jet of IPA with and without the application of heat pulses. The angle θ is the deflection angle of interest. The heater on the left side of the nozzle is used and the deflection is to the right in these images. The camera is tilted at 20^{o} from the plane of the printhead. A portion of the heater pulse sequence used is shown in Figure 5(b). A pulse width of 2 µs with a period of 8 µs was used for these experiments. Notice the formation of a uniform sequence of deflected drops within a short distance from the nozzle when the heat pulses are applied. Without the application of the heat pulses, the jet does not deflect and breaks up somewhat randomly at a longer distance from the nozzle.

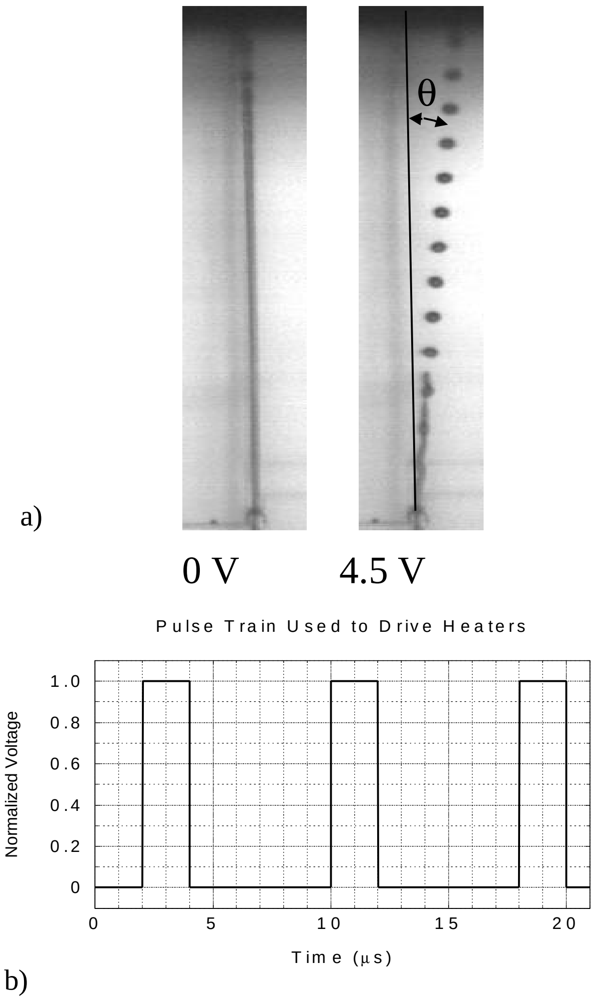

Figure 5: (a) Images showing a jet of fluid being deflected. (b) A portion of the waveform used to drive the heater.

We measure the deflection angle from such images using an image analysis program. For each image of a deflected jet, we acquire a corresponding image of the same jet without the application of heat pulses. We determine the velocity of the jet by measuring the distance between drops (knowing that the time spacing between drops is 8 µs).

To compare the performance of various fluids, we use the same voltage pulses for each fluid. This assures that the same energy per pulse is being supplied to the heater, but does not mean that the same heater temperature is being achieved. The peak heater temperature will be dependent on the thermal properties of the fluids and the velocity of the fluid as it flows past the heated portions of the structure.

We measure heater temperature by monitoring the resistance of the polysilicon heaters. The resistance of polysilicon increases linearly with increasing temperature. By placing a small resistor (10 Ω) in series with the heater between the lower voltage contact of the heater and ground, we can monitor the current flow through the heater during the heat pulse using a digital oscilloscope. As the heater resistance increases, the current decreases because the voltage is held constant during the heat pulse. We determine the temperature coefficient of resistance (TCR), $\Delta R/R$ in $^{o}C^{-1}$, by measuring the heater resistance at room temperature and one elevated temperature, typically 130°C.

6 CONCLUSION

In this paper, we have described a novel microjet deflection phenomenon, which has been applied to a continuous inkjet printing application. CMOS/MEMS devices have been constructed, which make use of this deflection approach. The device performance has been evaluated for a variety of fluids and various device configurations. CFD models have been constructed to evaluate these devices and study the deflection phenomena. Much has been learned about the underlying mechanism of this novel microjet deflection phenomenon and subsequent devices have been optimized through the utilization of this knowledge. However, the deflection phenomena exhibit a highly coupled nature and therefore much more work is needed to gain the intuitive understanding, which is truly desired.

7 REFERENCES

[1] J. M. Chwalek, D. P. Trauernicht, C. N. Delametter, R. Sharma, D. L. Jeanmaire, C. Anagnostopoulos, G. A. Hawkins, B. Ambravaneswaran, J. C. Panditaratne, O. A. Basaran, "A New Method for Deflecting Liquid Microjets," submitted to Physics of Fluids (December 2001).

[2] C. N. Delametter (Interview) "Micro Resolution" Mechanical Engineering Magazine (July 2001).

[3] E. P. Furlani, C. N. Delametter, J. M. Chwalek, D. P. Trauernicht, "Surface Tension Induced Instability of Viscous Liquid Jets" Technical Proceeding of the Fourth International Conference on Modeling and Simulation of Microsystems (April 2001).

[4] J. M. Chwalek, D. P. Trauernicht, C. N. Delametter, D. L. Jeanmaire, C. Anagnostopoulos, Novel Silicon-Based Continuous Inkjet Printhead Employing Asymmetric Heating Deflection Means" Technical Proceeding of the IS&T Non-Impact Printing (NIP17) Technologies Conference (October 2001).

[5] J.M. Chwalek et al., U.S. Patent 6,079,821 (2000).

[6] D. P. Trauernicht, C. N. Delametter, J. M. Chwalek, D. L. Jeanmaire, C. Anagnostopoulos, "Performance of Fluids in a Silicon-Based Continuous Inkjet Printhead Using Asymmetric Heating," Technical Proceeding of the IS&T Non-Impact Printing (NIP17) Technologies Conference (October 2001).

[7] FLOW-3D Theory and User's Manual, Version 7.7, Flow Science, Inc., 2001.

[8] C. W. Hirt and B. D. Nichols, J. Computational Phys. 39, 201 (1981).