A Simulation Model of Multi Drops Operation in Inkjet Head

Ryutaro Kusunoki, Takashi Norigoe, Toshio Miyazawa, and Tkao Ito
Toshiba TEC Corporation
Mishima, Japan

Abstract

A new method of analyzing an operation of an inkjet printhead by treating flow of ink in a nozzle and flow of ink in a chamber as separated models and connecting the boundary conditions of the two models was developed. This method was applied to simulate ink ejection by the multiple droplets, which had been difficult to analyze, and the simulation results successfully matched to the real experimental results.

Since the gray scale printing by the multiple droplets requires ejection of small ink droplets at a high frequency of over 100kHz, the problem of this simulation was how to treat flow of the ink which vibrates at high frequency in a chamber. The ink in a nozzle was modeled as two-dimensional incompressive flow having a free surface and the ink in a chamber was modeled as three-dimensional compressive flow, then connected the boundary conditions of these two models. Both practical accuracy and practical computing speed were obtained by this method.

This paper describes these models, compares the simulation with the experiments, and demonstrates the appropriateness of the simulation.

Introduction

In simulating an operation of ink ejection of a piezo type inkjet head, a combined model of CFD (Computational Fluid Dynamics) and an equivalent circuit have often been used.[1] In the combined model, the CFD model is applied to flow of ink in a nozzle, and the equivalent circuit model is applied to flow of ink in a channel and motion of an actuator.

Figure 1 shows an example of the typical equivalent circuit model. The coils 'L', the condensers 'C', and the resistances 'R' in the circuit represent inertia, compliance, and flow resistance of ink in a channel, respectively. It is relatively easy to add this kind of equivalent circuit model to commercial CFD software, and this is a merit of the equivalent circuit model.

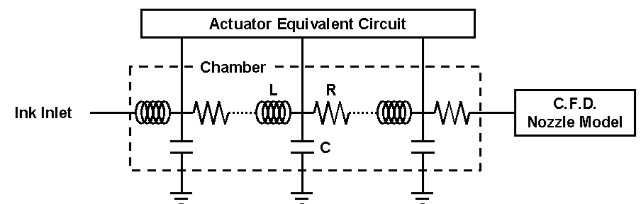

Figure 1. Equivalent circuit model

Since, in the equivalent circuit model, the flow in the channel is simplified to one-dimensional flow, velocity profile of the flow in the channel is assumed to be a parabola shape to determine the flow resistance 'R'. But the assumption is not appropriate when the flow vibrates at a high frequency. Therefore, the flow resistance of the equivalent circuit becomes inaccurate as the frequency increases.

One case of the high-frequency flow is ink flow in a channel of a printhead driven by the multi drops operation, which is a method of gray scale printing. Since the multi drops operation requires ejection of small ink droplets at a high frequency of over 100kHz, the flow frequency in the channel is also high.

To simulate the multi drops operation, CFD models were used not only for flow in the nozzle but also for flow in the channel. By using CFD models, the flow resistance at the high frequency could be evaluated accurately, which was inaccurate in the equivalent circuit model.

Simulated Object

Figure 2 shows a structure of the inkjet printhead to be simulated. The printhead is a shear mode and shared wall type. The actuator deforms according to a drive signal and presses ink in the chamber, so that a small droplet of 6pl is ejected from the nozzle.

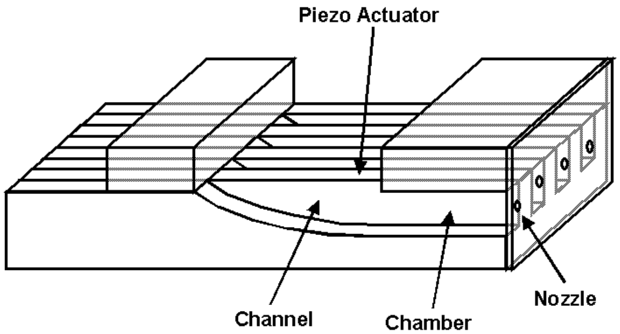

Figure 2. The structure of the inkjet printhead to be simulated

Figure 3 shows the principle of the multi drops operation. The gray scale printing in the multi drops operation is performed by controlling the number of the small droplets to be ejected. The ejected droplets are merged on a surface of printing medium into one pixel, and the dot size is changed. The droplets are ejected at a frequency of over 100kHz.

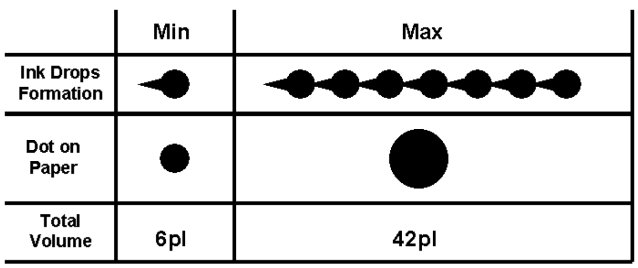

Figure 3. The principle of the multi drops operation

Problems of Simulation

In simulating the multi-drops operation, the flow frequency in the channel is too high to apply the equivalent circuit model for the flow in the channel.

Figure 4 shows velocity profiles of flow in the channel at a frequency of 1kHz, 10kHz, and 100kHz, where the channel width is 80μm, the ink density is 850kg/m^3, and the ink viscosity is 10mPa s. In the normalized position in the figure, 0 indicates one side of the channel wall, and 1 indicates the other side of the channel wall. The velocity scale is also normalized so that the flow rate of each velocity profile is equal to 1.

Figure 5 shows the relation between flow frequency and flow resistance, where the conditions are same as above. The scales of flow resistance is normalized that the flow resistance of constant flow indicates 1.

The velocity profile at 1kHz depicts a parabolic curve because there is enough time for the flow to develop. The flow resistance is constant in low frequency range. But the velocity profile at 100kHz depicts a trapezoid curve because there is not enough time for the flow to develop. This increases the flow resistance as the flow frequency increases.[2] The flow resistances at frequency of over 100kHz become to more than 3 times the flow resistance at the lowest frequency.

The conventional equivalent circuit model is not desirable as a model of the multi drops operation since the flow resistance in the model is constant regardless of the flow frequency. On the other hand, if the flow in the channel is directly analyzed by CFD, the frequency dependency of the flow resistance automatically considered, and we can skip the step of evaluating L, C, and R in the equivalent circuit.

But the approach by using CFD also involves problems. One of the problems is that the scheme of treating a free surface flow in the nozzle and a compressive flow in the channel simultaneously has not established yet. The VOF (Volume Of Fluid) method,[3] which is the general scheme of analyzing a free surface flow, cannot be applied to a compressive flow. Another problem is the number of unknown variables in the model. Even if we could treat the free surface flow in the nozzle and the compressive flow in the channel as one model, the model would have too large number of variables to analyze in a practical time.

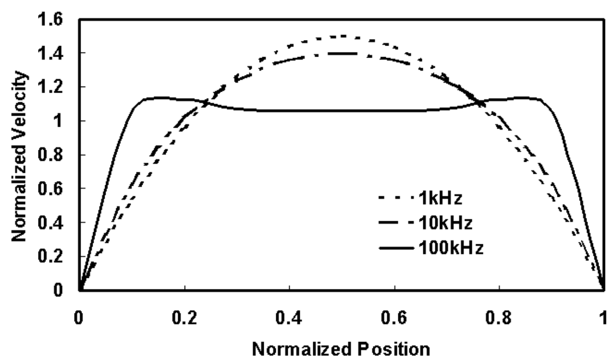

Figure 4. Velocity profiles in a channel

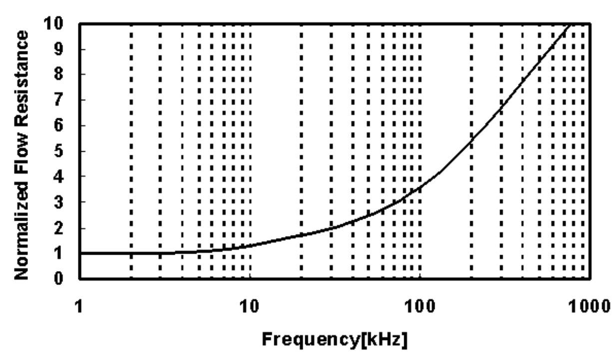

Figure 5. Flow frequency VS Viscous resistance

The Simulation Model

To avoid the problems in analyzing the flow in the channel with CFD, we developed a method of combinational analysis by modeling the flow in the nozzle and the flow in the channel separately, and coupling the boundary condition of one model with the boundary condition of the other model.

Figure 6 shows the simulation model of the inkjet printhead. The simulation model is comprised of two sub models. One sub model is the channel model to analyze flow of ink in the channel and the manifold, and motion of the piezo actuator. The other sub model is the nozzle model to analyze flow of ink in the nozzle and ejected from the nozzle. Each of sub models is solved by numerical analysis of finite difference method.

Table 1 shows the respective properties of the two sub models. The channel model is applied a 3D plane coordinate system. The channel model includes an ink part and an piezo actuator part. The ink part is governed by the Navier-Stokes equation, and the mass conservation equation considering the slight compressibility of ink. The actuator part is governed by the momentum equation considering the piezoelectricity. The mesh division in the channel model is relatively coarser than that in the nozzle model because the channel model has no free surface in it. On the other hand, the nozzle model is applied a 2D cylindrical coordinate system. The flow in the nozzle model is governed by the Navier-Stokes equation and the continuative equation. Free surface between the ink and the air is analyzed by the VOF method. To represent the complex shape of free surface, a fine mesh is used to divide the nozzle model.

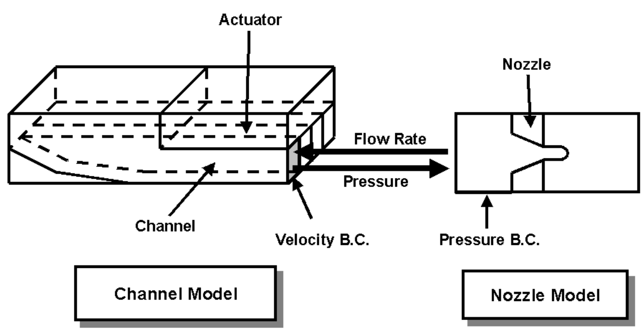

Figure 6. The simulation model for the multi drops operation

Table 1. The respective properties of the channel model and the nozzle model

	Channel Model	Nozzle Model
Coordinate System	3D Cartesian	2D Cylindrical
Considering Free Surface	No	Yes
Considering Compressibility	Yes	No
Typical Mesh Size	10-100 μm	1 μm

The channel model and the nozzle model cooperate by exchanging their boundary conditions each other at each step. In the channel model, interface to the nozzle model is set to be a velocity boundary condition. In the nozzle model, the interface to the channel model is set to be a pressure boundary condition. The velocity of the boundary condition in the channel model is given by Eq. (1).

$$V_{vbc} = \frac{1}{S_{vbc}} \int_{S_{pbc}} \mathbf{v}_{pbc} \cdot \mathbf{n}\, ds \qquad (1)$$

The pressure on the boundary condition in the nozzle model is given by Eq. (2).

$$P_{pbc} = \frac{1}{S_{vbc}} \int_{S_{vbc}} p_{vbc}\, ds \qquad (2)$$

where, V_{vbc} is the velocity applied to the velocity boundary condition; S_{vbc} is the sectional area of the velocity boundary condition; S_{pbc} is the sectional area of the pressure boundary; $\mathbf{v}_{pbc}$ is velocity vector on the pressure boundary face; **n** is normal vector of the pressure boundary face; P_{pbc} is the pressure applied to the pressure boundary condition; and p_{vbc} is pressure on the velocity boundary face.

Since the channel part and the nozzle part were modeled separately instead of modeling the printhead in one model, both free surface of ink in the nozzle model and compressibility of ink in the channel model could be treated, and the number of unknown variables could be reduced by selecting the best coordinate system and mesh size for each of sub models. As a result, an operation of an inkjet printhead could be analyzed with practical accuracy and practical computing time even if the flow in the channel was analyzed as 3D flow.

Case Study

A successive ejection of seven small droplets in the multi drops operation is analyzed by using the simulation model, and the simulated results were compared with the experimental results. Figure 7 shows relations between a velocity of the last droplet and a total volume of seven droplets in the simulation and the experiment. Figure 8 shows sequential droplets formations in the simulation at every 10 μs from the start of ejection. Figure 9 shows the experimental result in the same case as the simulation.

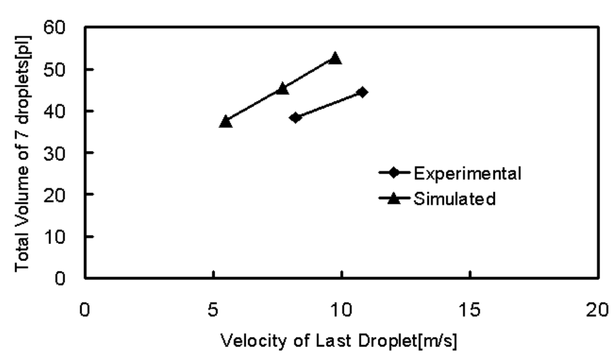

Figure 7. Relation between the velocity of the last droplet and total volume of 7 droplets

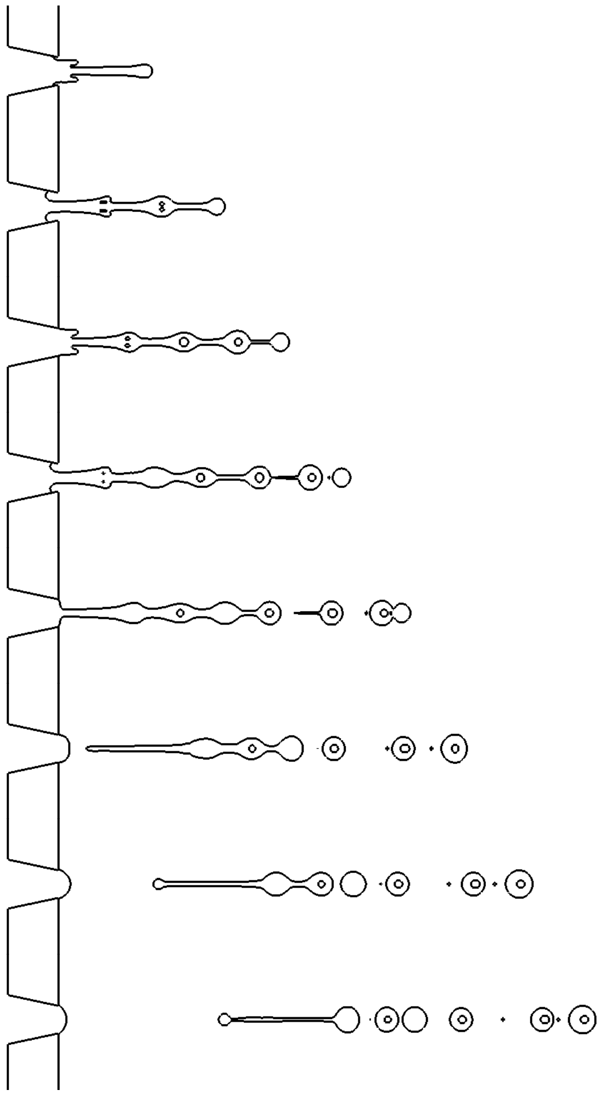

Figure 8. Droplets formation in the simulation

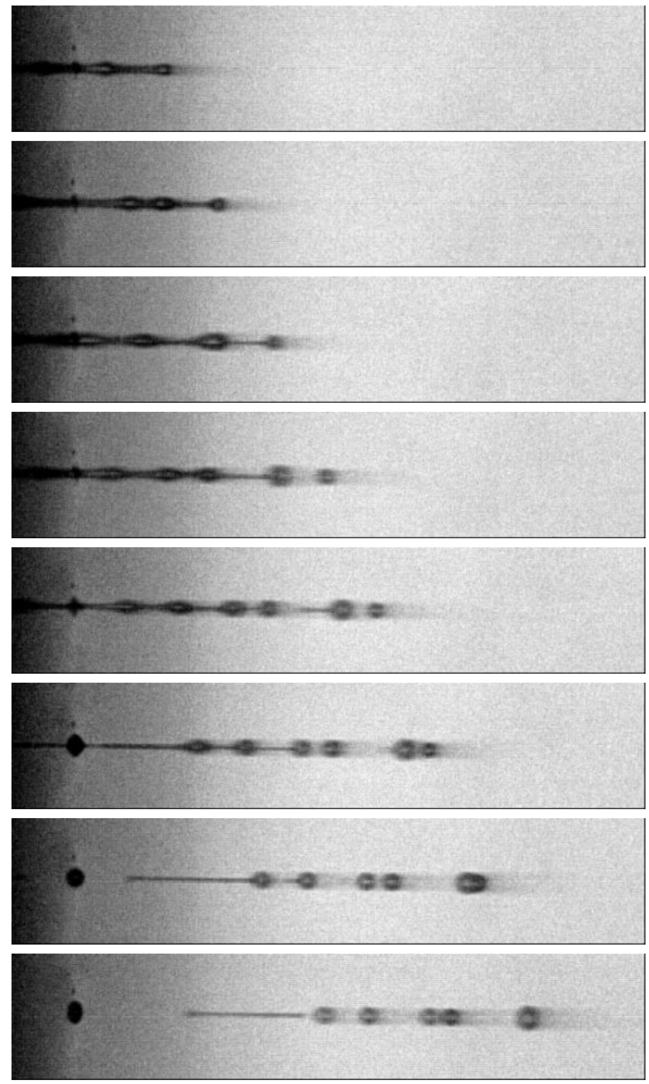

Figure 9. Droplets formation in the experiment

As shown in Figs. 8 and 9, the simulation well represents a form of ink drops similar to the experiment. As shown in Fig. 7, the simulation also nearly corresponds to the experiment in respect to quantity.

Conclusion

A new method of analyzing operation of an inkjet printhead by treating flow of ink in a nozzle and flow of ink in a channel as separated models was developed. This method was applied to simulate ink ejection by the multiple droplets, which had been difficult to analyze, and the simulation results successfully matched to the experimental results.

References

1. Sung-Cheon Jung et al., A New Approach for Analysis of Ink Jet Devices System, *Proc. IS&Ts NIP 15*, pg. 18. (1999).
2. Robert Hasenbein, Ink Jet Modeling for Development of New High Performance Ink Jet, *Proc. IS&Ts NIP 18*, p. 166, (2002).
3. C. W. Hirt and B. D. Nichols, Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, *J. of Comp. Phys.*, **39**, 201 (1981)

Biography

Ryutaro Kusunoki was born in Japan. He received his B.E. in mechanics from Science University of Tokyo, Japan, in 1989. He has engaged in studying of inkjet printheads, especially of shear mode and shared wall type since 1993. He is in charge of analyzing ink ejecting process, designing printheads, and developing drive waveforms.