NUMERICAL AND EXPERIMENTAL STUDY OF GAS FLOW CHARACTERISTICS IN MICROSCALE NOZZLES

ARAKI, Takuto, IWAI, Hiroshi and SUZUKI, Kenjiro
Department of Mechanical Engineering
Kyoto University, Kyoto, 606-8501, Japan

For effective and proper design of micro devices, the establishment of the physical laws governing the fluid flow and heat transfer in micro geometry is required. The authors[1] previously conducted an experimental and analytical study for low Reynolds number flow in relatively long micro channels (d=2μm~,L/d=1000~) to investigate the gaseous flow characteristics in micro geometry, where the effects of fluid compressibility and rarefaction were discussed. Due to the channel geometry and relatively lower velocity, the entrance effect and effect of dissipation[2] were neglected in the previous study. However, both of these parameters will be important for high speed flow in relatively short micro-channels (microscale nozzle). As microscale nozzles have a good potential to be utilized for mixing enhancement, impinging cooling and in micro combustors etc., understanding such effect is essential. This motivated the present study. High speed flow in microscale nozzle and related thermal phenomena are investigated through experimental and numerical approaches.

EXPERIMENTAL AND COMPUTATIONAL PROCEDURES

Experimental Setup and Conditions

The experimental apparatus used in this study is shown in Fig.1. The pressure difference between the inlet and outlet of the test section and mass flow rate were measured.

Table 1 Nozzle geometries

Nozzle type	Shape of cross section	$D(W)_{inlet}$ (μm)	$D(W)_{outlet}$ (μm)	L (mm)	L/D
A	Circular	29.20	30.10	0.030	1.01
B	Circular	29.70	31.00	0.050	1.65
C	Circular	30.90	35.90	0.100	2.99
D	Rectangular	100.4	100.8	0.10	0.99

Computational domain, schemes

The governing equations are two-dimensional time-dependent compressible Navier-Stokes equations,

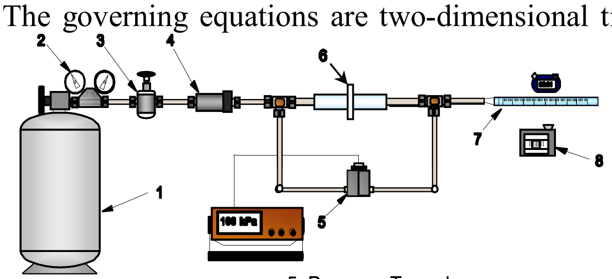

Fig.1 Experimental apparatus

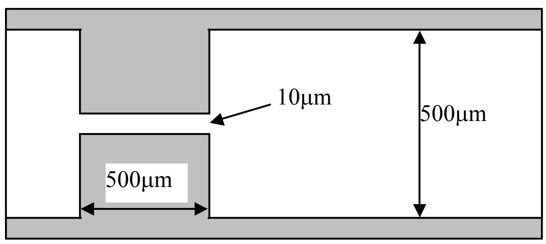

Fig.2 Computational domain

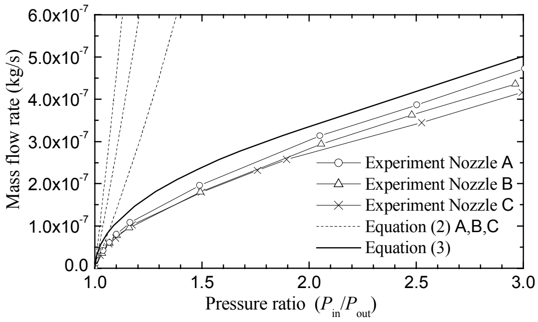

Fig.3 Mass flow rate as a function of Pressure ratio

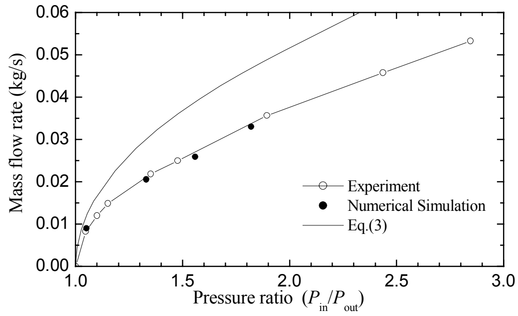

Fig.4 Comparison of Numerical results with Experimental data for nozzle D

energy equation and continuity equation. Viscous dissipation and reversible work terms are considered in the energy equation (refer Eq.(1)) . The equations are discretised using the finite difference method. SIMPLE algorithm is used for the computation of pressure correction. Maxwell's velocity slip boundary condition was adopted on the wall[3].

$$\rho Cp\frac{DT}{Dt}+\left(\frac{\partial \ln \rho}{\partial \ln T}\right)_p\frac{Dp}{Dt}=\mathrm{div}\left(k\ \mathrm{grad}T\right)+\mu\left[2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)^2-\frac{2}{3}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}\right)^2\right] \quad (1)$$

RESULTS AND DISCUSSIONS

Fig.3 shows experimental results of mass flow rate as a function of inlet/outlet pressure ratio. At the very low pressure ratio ($P_{in}/P_{out}<1.05$), the mass flow rate is not far from Eq.(2) which we adopted for long micro-channel analysis[1]. As the pressure ratio increases, however, the present experimental data show better agreement with Eq.(3). Eq(3) is obtained assuming one-dimensional invisid flow with constant entropy, and it represents *Choking* phenomena, which means that the flow cannot be accelerated over sonic speed without a diffuser. Therefore it is concluded that *Choking* dominates the mass flow rate in micro scale laminar flow under such conditions rather than viscosity or skin friction.

$$\dot{m}=\frac{\pi D^4\rho_{out}P_{out}}{256\mu L}\left[\left(\frac{P_{in}}{P_{out}}\right)^2-1+16\frac{2-\sigma}{\sigma}\mathrm{Kn}_{out}\left(1-\frac{P_{in}}{P_{out}}\right)\right] \quad (2)$$

$$\dot{m}=A\sqrt{\frac{2\kappa}{\kappa-1}P_{in}\rho_{in}\left(\frac{P_{out}}{P_{in}}\right)^{2/\kappa}\left[1-\left(\frac{P_{out}}{P_{in}}\right)^{(\kappa-1)/\kappa}\right]} \quad (3)$$

Numerical simulation is an effective approach to study the local phenomena in microscale geometry since it is very difficult to take measurements experimentally. Prior to a systematic analysis, some computations were carried out under the condition equivalent to nozzle D to verify the numerical procedure used in this study. Numerical results are plotted in Fig.4 with experimental data, in which very good agreement can be seen.

A systematic numerical analysis is conducted to study the effect of the thermal boundary condition of the wall. For this purpose, a longer and narrower nozzle compared to the nozzles in Table 1 is selected. The computational domain is shown in Fig.2. Pressure and velocity distributions along the nozzle center line are plotted in Fig.5 for the case of Re=100. The maximum velocity exceeds 200m/s (Ma≈0.6) near the nozzle outlet as seen in Fig.5. It is expected that a transonic laminar flow is established at such locations. Such flow condition cannot be seen in conventional scale. Fig.6 shows temperature contours and velocity vectors near the nozzle inlet. There is a large pressure drop at the nozzle inlet due to the acceleration and the separation of flow. The degree of pressure drop is more than 10 times larger than what would be expected in conventional scale[4]. Similarly, temperature drop is also

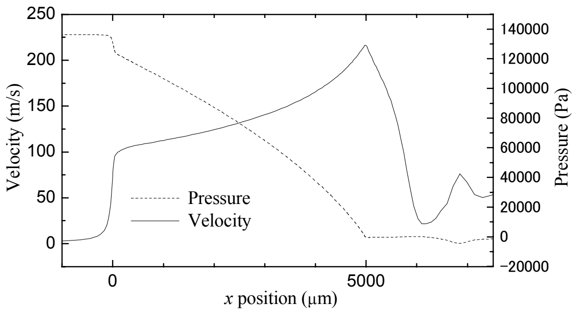

Fig.5 Velocity and Pressure distribution along the centerline

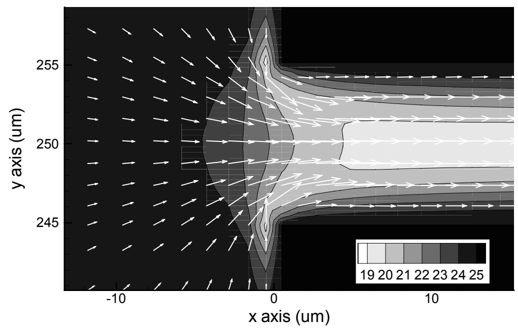

Fig.6 Temperature contours and velocity vectors (Isothermal wall with dissipation)

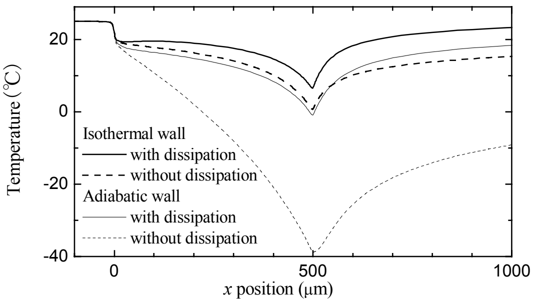

Fig. 7 Temperature distribution along the centerline

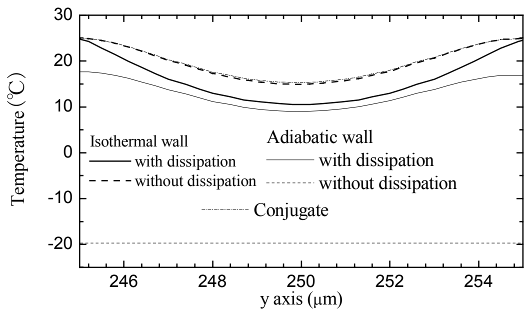

Fig.8 Temperature profile at x=400μm

observed at nozzle inlet where the flow is drastically accelerated. This is because of the reversible work term. These results indicate that the inclusion of nozzle entrance effects to the computation is quite important for simulation of flow and thermal fields in a microscale nozzle. The temperature drop is actually observed not only at the nozzle inlet but also along the whole length of the nozzle as shown in Fig.7. Energy conversion from thermal energy to kinetic energy is clearly observed. The temperature drop is reduced if the dissipation term is included. It follows from this that both dissipation and reversible work strongly affect the temperature field indeed and should not be ignored. Additionally, this conclusion is made clearer in Fig.8.

A further computation, which considers the thermal conduction inside the solid parts, was also performed, and the temperature profile appears in Fig.8. The results appear identical to that of isothermal wall case although slight a deviation between the two cases is observed in some regions such as in the neighborhood of the outlet. This suggests that even when the flow is transonic, it is acceptable to adopt an isothermal wall boundary without considering conduction through the solid part in microscale flow. This conclusion is in contrast to the practice in the conventional scale where an adiabatic wall boundary is normally used.

REFERENCES

1. Araki, T., Kim, M.S., Iwai, H. and Suzuki, K., An Experimental Investigation of Gaseous Flow Characteristics in Micro-channels, *MTE J.*, 2001, in printing
2. Z. Y. Guo, Size Effect on Flow and Heat Transfer Characteristics in MEMS, *Proc. Int. Conf. on HTTPM*, 2000, pp24-31
3. Beskok, A. and Karniadakis, G. E., A Model for Flows in Channels, Pipes, and Ducts at Micro and Nano Scales, *MTE J.*, 3, pp. 43-77, 1999.
4. General Electric, Fluid Flow data book, 405-3, General Electric Co.1970