出國報告（出國類別：參加研討會）

出席第 24 屆國際傳輸現象會議
心得報告

服務機關：國防大學理工學院機電能源及航太工程學系
姓名職稱：曾培元教授
派赴國家：日本
出國期間：102/10/30~102/11/07
報告日期：102/12/6

目次

摘要

在日本山口縣東京理科大學（Tokyo University of Science, Yamaguchi）舉行的第 24 屆國際傳輸現象會議（The 24th International Symposium on Transport Phenomena）是國際工程學界有關傳輸現象學術與應用的重要會議，由太平洋熱流工程中心 (The Pacific Center of Thermal-Fluids Engineering) 發起，自 1985 年起至今已舉辦 24 屆，參與活動的學者遍布亞、美、歐、澳等各洲國家，研究範圍含蓋宏觀到微觀、理論計算與實驗的動量能量與質量傳輸等，在國際上具重要學術地位。本屆議程內容計有邀請專家發表主題講演 6 場，有 151 篇論文分成 14 項議題場次以口頭報告方式發表。本人與研究團隊所提有關微通道中稀薄氣體流動與熱潛變之研究論文，於 11 月 3 日下午由博士班研究生戴中傑上台口頭報告。和在場學者專家們有相當熱烈的問答激盪與經驗交流。參加本次會議除了發表研究計畫的成果外，並聽取多場議題研討及論文報告，了解熱流工程傳輸現象學術領域，目前的研究重點與將來的發展趨勢，吸收新知開闊視野，對提升和精進個人與研究團隊未來之研究績效很有幫助。

目的

1. 發表國科會補助計畫 NSC101-2221-E-606-003 之研究成果。
2. 與熱流工程傳輸現象領域之國際學者專家進行學術交流。

過程

1. 第 24 屆國際傳輸現象會議（The 24th International Symposium on Transport Phenomena, ISTP-24）於 2013 年 11 月 1 日至 5 日在日本山口縣的全日空飯店(ANA Crowne Plaza)和東京理科大學(Tokyo University of Science, Yamaguchi)舉行。國際傳輸現象會議是國際工程學界有關傳輸現象學術與應用的重要會議，由太平洋熱流工程中心 (The Pacific Center of Thermal-Fluids Engineering) 發起，自 1985 年起至本次已舉辦 24 屆，參與活動的學者包含亞、美、歐、澳等各洲國家學者，研究範圍包蓋宏觀到微觀、理論計算與實驗的動量能量與質量傳輸等，在相關學術界具重要地位。議程計有 6 場邀請專家發表主題講演，151 篇論文以口頭報告方式發表。略分為 14 個不同科技議題 (Technical Session)：Heat and Mass Transfer, Combustion and Reacting Flows, Boiling and Multiphase Flows, Experimental and Computational Fluid Dynamics, Sustainable and Renewable Energy, Transport in Porous Media, Noise and Vibration in Fluids, Heat Exchangers, Thermo-fluids Machinary, Micro and Nano Scale Transport, Electronic Packaging and Thermal Management, Manufacturing and MEMS Application 等。
2. 1 日下午在山口縣宇部市的全日空飯店報到並參加歡迎茶會。遇到幾位常在國際學術會議上見面的老朋友，如大會總顧問也是太平洋熱流工程中心負責人 Prof. Mochizuki，韓國高等科技學院 (KAIST) Prof. Lee 都是舊識，而來自委內瑞拉國立塔吉拉大學機械系的 Prof. Velazquez-Araque 是捷克科技大學博士，都參加過 2005 在布拉格和 2010 在高雄的 ISTP，今年又重逢於日本大家倍感歡欣。
3. 2 日上午開幕典禮及三場大會主題演講在全日空飯店舉行。奧地利雷歐本大學環境與能源工程系主任 Prof. Raupenstrauch 給的第一場大會演講，題目是高溫氣固反應爐的數學模型及在資源回收程序之應用。Prof. Raupenstrauch 穿著奧地利里歐本傳統禮服很受矚目，雷歐本是很美的城鎮將接辦 ISTP-26。 韓國高等科技學院 (KAIST) Prof. Lee 給的大會演講，是他長期致力的液滴撞擊具微結構或多孔性質壁面之研究。第三場的講者是東京理科大學 Suwa 校區的校長 Prof. Kawamura，他介紹所帶領的國際太空站日本微重力實驗模組 Kibo 研究團隊，近期有關 Marangoni Convection 實驗的成果，看到三公分直徑大水珠的形成過程，相當精彩。
4. 2 日下午起各議題場次的論文發表移師至山口縣東京理科大學舉行。為了瞭解

Prof. Velazquez-Araque 在委內瑞拉開拓計算流力領域的研究現況，並期許他將來能在南美洲接辦一次 ISTP，特別去聽他有關具內推進系統飛行器翼形數值分析的論文報告。另外也聽取同場次兩位韓國學者 Dr. Ryu 和 Dr. Cha 有關鈍形體或超音速飛機表面熱傳和紅外線幅射計算模擬的研究等。

5. 3 日下午本人與研究團隊所提之論文為 ”滑移流區微通道中稀薄氣體流動與熱潛變之研究 ” (Rarefied Gas Flow and Thermal Creep through Micro-channel in Slip Flow Regime) 於 16 點 20 分至 40 分由博士班研究生戴中傑上台口頭報告。本論文研究重點在於針對滑移流區微通道中稀薄氣體流動與熱潛變，進行氣體分子直接模擬蒙地卡羅法 (Direct Simulation Monte Carlo, DSMC) 分析，探討不同邊界設定模式以及切向溫度梯度的變化，對熱潛變現象與壁面滑移流動的影響。與在場日本京都大學的 Dr. Matsumoto，九州理工學院的女教授 Prof. Nagayama 和主持人 Prof. Kohno 等多位學者，就本研究的應用和滑移速度沿流道增加的原因等問題，有相當熱烈的問答激盪與經驗交流，獲益良多。
6. 3 日晚上在全日空飯店大會晚宴，欣賞日本古樂和太鼓表演。4 日議程與本人研究主題較無相關，參觀東京理科大學圖書館和機械系後，和韓國高等科技學院 Prof. S.Y. Lee 與漢陽大學機械學院院長 Prof. K.S. Lee 等人一起搭巴士賦歸。

心得及建議

1. 參加本次會議除了發表研究計畫的成果外，並聽取多場議題研討及論文報告，了解熱流工程傳輸現象學術領域，目前的研究重點與將來的發展趨勢，吸收新知開闊視野，對提升和精進個人與研究團隊未來之研究績效很有幫助。
2. 會議期間曾和多位地主國日本和韓美英印度等國學者，以及來自國內清華大學潘欽教授和旅日東北大學機械系博士生許書函等多位學者專家們晤談甚歡。但是很多經常參與 ISTP 也來過台灣的國際友人，都對這次會議台灣與會人數驟減感到不解。過去每屆 ISTP 臺灣參加人數平均在二十人左右，今年卻只有個位數，國內傳輸現象相關的研究似乎成果不如以往，令人擔憂。
3. 山口東京理科大學主辦本次會議，主席和秘書長都是機械系熱流工程領域的教師，以相對小規模且不在都會區的校系願意承辦國際學術會議，展現他們小而精美的自信，仔細瞭解該校，確有其辦學之特色，他們集中於發展幾項重點如液晶和先進材料的研究、並設立科技交流中心積極推動產學合作，令人印象深刻，值得學習。
4. 本次會議所需費用較低，特別鼓勵本人的博士班研究生戴中傑一起參加。地主國日本教授們也大多帶領研究生參加，宣讀論文時多由研究生上台，讓研究生在國際性的學術活動中得到啟發與學習的機會。國科會對國內教師和研究生參加國際學術會議雖有補助，但仍顯不足。建議國科會或國防大學增加相關預算經費，給予教師和研究生參加國際學術會議足夠的鼓勵與獎助。

附件：參加該研討會照片及論文資料。

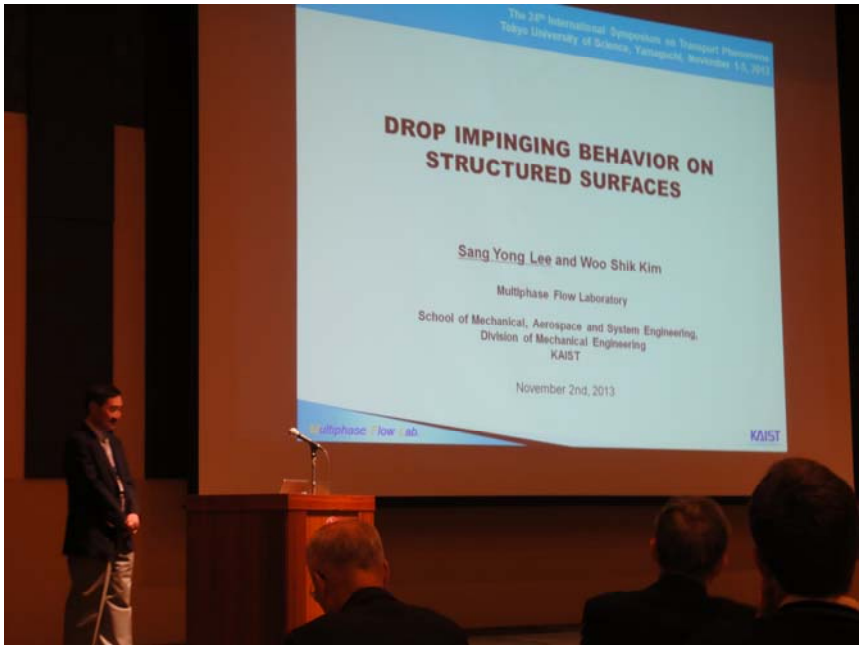
在報到處接受大會秘書長 Professor Yuki 的問候

與參加本會議的學者在歡迎茶會中交談

大會主席 Prof. Koichi Suzuki 開幕致詞

聆聽大會主題演講

奧地利雷歐本大學 Prof. Raupenstrauch 給的大會演講



韓國高等科技學院 (KAIST) Prof. Lee 給的大會演講

在研討會場與博士生戴中傑和大會秘書長 Professor Yuki 合影

在微奈米尺度傳輸場次之論文發表前留影

在大會晚宴中與大會主席 Prof. Koichi Suzuki 合影

大會晚宴宣布下二屆主辦國家是泰國和奧地利

RAREFIED GAS FLOW AND THERMAL CREEP THROUGH MICROCHANNEL IN SLIP FLOW REGIME

C.C. Tai [1], P.Y. Tzeng [2], C.Y. Soong[3] and F.C. Ji[4]

[1] School of Defense Science
[2 4]Department of Mechatronic, Energy and Aerospace Engineering
Chung Cheng Institute of Technology, National Defense University, Taoyuan 33509, Taiwan, R.O.C.
[3]Department of Aerospace and Systems Engineering
Feng Chia University, Taichung 40724, Taiwan, R.O.C.

ABSTRACT

The purpose of this research is to study the rarefied gas flow through micro-channel with tangential temperature gradient along channel walls. The major concern is focused on the effects of tangential gradient of wall temperature on velocity slip and the corresponding interfacial phenomenon called thermal creep. Investigation of effective models applied in DSMC simulation for wall boundary conditions will also be explored with the application on rarefied gas through micro-channel in slip flow regime. The results of this study will be useful in providing proper boundary treatment for simulation of rarefied gas flow problems, and significant insight for the thermal effect on slip boundary.

INTRODUCTION

With the rapid development of micro and nano electromechanical system (MEMS or NEMS) technology, microscale rarefied gas flow become a new and important research field. A better understanding of transport phenomena in the basic components of microdevices such as microchannels, microvalves, and microbearings are very desirable. Gaseous flows in microchannel are encountered in many MEMS devices. The study of microchannel flow is partly in respond to the need for thermal control in the operation of MEMS. In these flows, the characteristic speeds of order of 1m/s are much smaller than the sound speed at normal temperature. The deviation and break down of the microflow state from continuum are measured by Knudsen number (*Kn*) which is defined as the ratio of mean free path of molecules to the characteristic system size. For microchannel of width of 1 micron or less, Kn~0.001-0.1 lies in slip flow to transition regime. In this regime, the fluid can no longer be regarded as a continuum and its motion has to be described from a molecular point of view. Therefore it would lead to incorrect results if the gas flow in microchannel were considered as continuum phenomena and solved by conventional Navier-Stokes equations. Previous researches have shown that there are definite differences between continuum predictions and actual experiments (1-4). Arkilic et al. (5, 6) develop an analytical solution for microchannel flow based on slip boundary conditions. Their results showed that the pressure distribution in microchannels is nonlinear. They also studied the effect of tangential momentum accommodation coefficients for the slip flow conditions. Pong et al.(7) experimentally investigated the pressure distribution along a microchannel. Their results proved experimentally, for the first time nonlinearly in the pressure distribution along a microchannel. As an alternative molecular based direct simulation Monte Carlo (DSMC) approach (8) has been recognized to be best suited for studying dilute gases in the slip to transitional regime.

DSMC method is based on the splitting of the molecular motion and intermolecular collisions by choosing a time step smaller than the mean collision time and tracking the evolution of these molecular processes in space and time. The traditional DSMC method is widely applied in the prediction of hypersonic rarefied gas flows. Recently, this method has been extended to low speed flows in microchannel. Many researchers have studied the flow and heat transfer characteristics in microchannels using DSMC method. Piekos and Breuer(9) confirmed the nonlinearity in pressure distribution and attributed it to compressibility effect with DSMC method. Oh et al.(10) have used DSMC method to investigate high speed flow in microchannels for range of Kn. The results showed that the velocity slip and temperature jump at the channel entrance increased as *Kn* increase. Fang and Liou (11, 12) studied the heat transfer characteristics of supersonic flows in microchannels where possible causes of the increase of wall heat transfer were discussed. Recently, application of the pressure boundary conditions using DSMC method has been extended to low speed flows in microchannel. Fan and Shen (13) proposed more efficient and accurate simulation of low speed microchannel flows, measures that can reduce statistical noise such as information preservation technique. Hassan and Esmail (14) investigated the flow and heat transfer behaviors under different *Kn* for subsonic flows in parallel and series microchannels. Moreover, Kursun and Kapat (15) studied the microscale flow behavior over backward facing steps where flow field, temperature, and wall heat flux at various *Kn* numbers were compared.

Molecular-based rarefied gas investigation is of great importance in micro/nano system design. Although numerical simulation of micro-gas flows has been developed for over forty years, there are still many aspects that can be improved. For low-speed micro-flows, the boundary conditions which can be obtained from the experiments are pressure and temperature. Kinetic theory successfully present models characterizing the molecular momentum and energy transport at ideal gas-solid interfaces. However, non-equilibrium effects such as rarefaction and gas surface interactions need to be taken into account. The physical mechanism of more complex

surface conditions still remains unclear. More researches are desired to setup the database and develop models for various gas/solid interfacial treatments in real engineering conditions. The interactions of the gas molecules with the solid surface affect the aerodynamic forces and thermal effect on the surface. Thus, continued improvement of gas-surface interaction models are certainly required for correct prediction of the aero-thermodynamic characteristics in rarefied gas flows.

Very few previous researches have examined extensively the effects of tangential gradient of wall temperature and gas-surface interaction models on microchannel flows. Therefore, the main purpose of this study is to explore the flow behaviors inside the micro-channel when the effects of tangential gradient of wall temperature and gas-surface interaction models become important.

NUMERICAL METHOD

DSMC is a numerical tool to solve the Boltzmann equation on the direct statistical simulation of the molecular processes described by the kinetic theory, which employs a large number of statistically selected simulated particles of appropriate physical size. The basic outline for the DSMC algorithm can be briefly described as follows. A computational cell system representing physical space is first generated for the requirement of sampling of macroscopic flow properties. The cell dimension should be of the order of the local mean free path. The simulated molecules are distributed into cells and appropriate velocity and internal energies are assigned to the molecules as initial conditions. The physical space is divided into computational cells and each cell is also divided into sub-cells. Sub-cells are used to select possible collision pairs. The computation then proceeds in small time steps over which the motion of the molecules and intermolecular collisions are treated separately. This time step must be smaller than the mean collision time.

Four primary steps constitute the core loop of the DSMC method.

Step 1: Streaming. For the given time interval, the simulated molecules are translated according to their velocities, and interactions with boundaries are computed.

Step 2: Indexing. The simulated molecules are sorted and their cell locations are rearranged.

Step 3: Colliding. Binary collisions in each sub cell are performed probabilistically.

Step 4: Sampling. The microscopic states of molecules in each cell are sampled to calculate the macroscopic properties at the cell center.

In the present study, the Modified No Time Counter (MNTC) scheme(16, 17) is used to determine collision pair selection, with collision cross-section obtained from the Variable Hard Sphere (VHS) mode. More details of the DSMC methodology can be found in Bird(8).

BOUNDARY CONDITION

Stream boundary

Ikegawa and Kobayashi(18) developed a DSMC simulator for pressure specified boundary conditions using particle flux conservation concept. Nance et al. (19) applied characteristic based boundary conditions at the downstream boundary and the number flux to determine the velocity at the upstream. Wu et al.(20) developed the particle flux conservation based method and applied it at both the upstream and downstream pressure boundaries.

At the inlet, the streamwise component, u_i, is determined for each boundary cell m by considering the conservation of particle fluxes across a boundary cell m's boundary surface. For a given mean speed and temperature, the particle flux across a boundary surface with area A in a particular direction can be determined, assuming equilibrium Maxwell-Boltzmann distribution, as

$$\frac{\dot{N}}{A} = \frac{nC_{mp}\left\{\exp\left(-s^2\right)+\sqrt{\pi}s\left[1+erf\left(s\right)\right]\right\}}{2\sqrt{\pi}} \tag{1}$$

where

$$s = \frac{C}{C_{mp}}\cos\theta \tag{2}$$

$$C_{mp} = \sqrt{\frac{2kT}{m}} \tag{3}$$

Considering the interface of boundary cell m we can apply Eq.(1) to determine the rate of particles crossing in either direction. Applying the particle flux conservation at the boundary gives

$$\left(u_i\right)_m = \frac{\left(\dot{N}_+ - \dot{N}_-\right)_m}{n_i\left(A\right)_m} \tag{4}$$

$(\dot{N}_+)_m$ and $(\dot{N}_-)_m$ are computed using the latest updated $(u_i)_m$ and sampled $(u_1)_m$ will be varied during the simulation and eventually attain a nearly constant value as the steady state is reached.

On the other hand, at the exit, all fluid properties except pressure are computed from the simulation. The same way, we apply the concept of particle flux conservation for the exit, rather than applying the theory of characteristics. Then, the similar procedure for inlet conditions is carried out for updating $(u_e)_m$. Note that the exit temperature, $(T_e)_m$, which is not given in advance, is enforced to equal to the temperature of each exit boundary cell m, as

$$\left(T_e\right)_m = \left(T_2\right)_m \tag{5}$$

where $(T_2)_m$ is initially guessed and then evolves during the simulation. Additionally, the exit number density is computed using the equation of state as

$$\left(n_e\right)=\frac{P_e}{k\left(T_2\right)_m} \tag{6}$$

Combining Eqs. (1)-(6) and applying the enforcement of conservation of particle flux Eq.(4), at the outflow pressure boundary, the simulated exit pressure is found to be consistent with the specified exit pressure and the mass conservation holds automatically as well.

Solid boundary

In order that the micro-channel with tangential temperature gradient along channel walls can be properly modeled, two kinds of boundary conditions are specified. One is the adiabatic walls with inlet/exit temperature difference. The other is the heat transfer wall with linear temperature increment (LTI-BC) along the wall using regular diffuse reflection rule for DSMC calculation.

Partial specular reflection boundary condition (PS-BC): Tzeng et al. (21) proposed a relatively novel boundary treatment in which the reflected particle, after colliding with the wall, was also assumed to have its total energy unchanged, $|c^*|=|c|$ .This collision rule was applied every where on the wall with the normal velocity reversal, $v^*= -v$, but with the two velocity components tangential to the wall, v and w, reflecting randomly, viz.,

$$u^*=\cos\left(2\pi R_f\right)\left(|c|^2-v^2\right)^{1/2} \tag{7}$$

$$w^*=\sin\left(2\pi R_f\right)\left(|c|^2-v^2\right)^{1/2} \tag{8}$$

where R_f is a random number. In this collision rule, the random reflection of the molecule velocity components in the two directions tangential to the wall partially simulates the molecule reflection on a rough surface. This treatment is a hybrid one with partial specular and diffuse reflection boundary conditions.

Isotropic scattering boundary condition (IS-BC) : Later Tzeng et al.(22) develop the isotropic scattering gas-surface interaction model. This model using energy conservation concept, keeping the particle velocity magnitude invariant and the velocity direction of the reflected particle set based isotropic scattering boundary conditions in the half space. In the spherical coordinate system(c, ζ , φ). The three velocity components normal and tangential to the wall, u, v, w, reflecting randomly are

$$v^*=-|c|\cos\zeta \tag{9}$$

$$u^*=|c|\sin\zeta\cos\phi \tag{10}$$

$$w^*=|c|\sin\zeta\sin\phi \tag{11}$$

since the sampling of the polar angle ζ is according to the cosine rule of diffuse reflection, cosζ is uniformly distributed in the axial direction, and the azimuth angle φ is uniformly distributed between 0 and 2π, i.e. $\varphi=2\pi R_f$ therefore

$$v^*=-|c|R_{f1}^{\frac{1}{2}} \tag{12}$$

$$u^*=|c|\left(1-R_{f1}\right)^{\frac{1}{2}}\cos\left(2\pi R_{f2}\right) \tag{13}$$

$$w^*=|c|\left(1-R_{f1}\right)^{\frac{1}{2}}\sin\left(2\pi R_{f2}\right) \tag{14}$$

Therefore the preceding two boundary conditions have the adiabatic effect and with random reflection which signified the general characteristics of a real solid wall.

PROBLEM MODELING

The computational domain, schematic of the channel geometry and coordinates for LTI-BC are shown in Figure 1. The argon gas is flowing through the two dimensional microchannel of 10 micron long by 1 micron high.

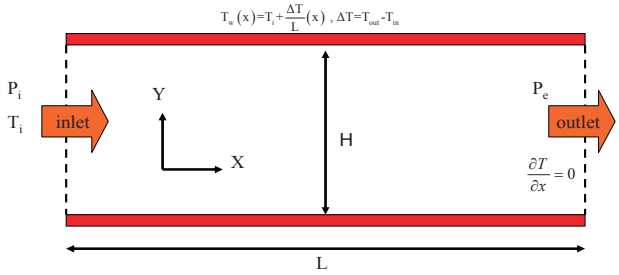


Figure 1. The schematic of channel geometry and coordinates.

In the present research, both the upper and the lower plane are solid boundary. The inlet temperature T_i is kept constant at 300K. The effect of thermal transport is investigated at three linear temperatures increment at ΔT =100K, 200K, and 300K conditions respectively. The interaction between molecules and the solid boundaries are modeled as diffuse reflection for LTI-BC and PS or IS reflection rules for adiabatic wall. The left and right sides of the channel are the stream boundary with pressure P_i and temperature T_i at entrance, and pressure P_e, temperature T_e at exit. We assume that the exit temperature gradient is equal to zero, and the entrance velocity u_i and exit velocity u_e are resolved according to the number flux conservation theory with Wu et al. pressure boundary treatment (20). The initial temperatures inside the flow field are set to be the same with the inlet temperature, and the actual temperature varies with the simulation process.

Table 1. Conditions for Microchannel Simulation

	Case1	Case2	Case3	Case4
Gas	Argon			
AR(L/H)	10			
P_i/P_e	3			
T_i	300K			
Cell	250×50			
$\triangle T_w$	0K	100K	200K	300K

The geometry is fixed. The values of n_i and n_e are changed by setting the value of P_i, P_e and adjusting the variation of Kn_i, Kn_e respectively. The Argon gas approximated with the Variable Hard Sphere (VHS) model is simulated. The number of simulated particles is about 500000, the calculations are implemented on a high-performance computer and the main configurations are Intel

I7 with four processors. Each calculation of case takes about 200 hours of CPU time. More detail of this simulation conditions are listed in Table 1.

RESULTS AND DISCUSSION

The flow regime for Kn<0.001 is known as the continuum regime, where the Navier-Stokes equations with no-slip boundary conditions govern the flow. In the slip flow regime ($0.001 \leqq Kn \leqq 0.1$) the often-assumed no-slip boundary conditions seem fail, and a solid boundary layer on the order of one mean free path, known as the Knudsen layer, starts to become dominant between the bulk of the fluid and the wall surface. The flow, in the Knudsen-layer cannot analyze with the Navier-Stokes equations, and it requires special solutions of the Boltzmann equation. This results in a finite velocity slip value at the wall, and the corresponding flow regime in known as the slip flow regime. In the slip flow regime the flow is governed by the Navier-Stokes equations, and rarefaction effects are modeled through the partial slip at the wall using Maxwell's velocity boundary conditions as

$$u_s = -\frac{(2-\sigma_v)}{\sigma_v} Kn \frac{\partial u}{\partial n} - \frac{3}{2\pi} \frac{(\gamma-1)}{\gamma} \frac{Kn^2 \,\mathrm{Re}}{Ec} \frac{\partial T}{\partial s} \qquad (15)$$

where Re is the Reynolds number, Ec is the Eckert number, γ is the specific heat ratio.

For pressure-driven micro-Poiseuille flow, use of the first order slip boundary condition and assuming isothermal conditions results in the following distribution for the mean velocity (23) as

$$u = \frac{h^2}{2\mu} \frac{dP}{dx} \left[\left(\frac{y}{h} \right)^2 - \frac{y}{h} - \frac{2-\sigma_v}{\sigma_v} Kn \right] \qquad (16)$$

Equation (16) shows that the velocity is a function of the local Knudsen number, the distance to the wall, as well as the pressure gradient. For full diffusely reflecting wall, σ_v=1, using the centerline velocity as the velocity scale the normalized mean velocity can be obtained as

$$\frac{u}{u_c} = \left[-\left(\frac{y}{h} \right)^2 + \frac{y}{h} + Kn \right] \Big/ \left(\frac{1}{4} + Kn \right) \qquad (17)$$

Figure 2 illustrates the comparison of the DSMC velocity profiles with the continuum-based analytical solution of Eq.(15) at four different locations along the channel. The value of Kn changes from 0.04 to 0.07. For the small Kn, there is good agreement in centerline velocity and a little difference between the first order analytical solution in slip velocity. Simultaneously, analytical formula using the Navier-Stokes equations with slip wall conditions of the velocity distribution can give the slip velocity quantitative at y=0. The resulting normalized wall slip-velocity can be written as Eq.(18)

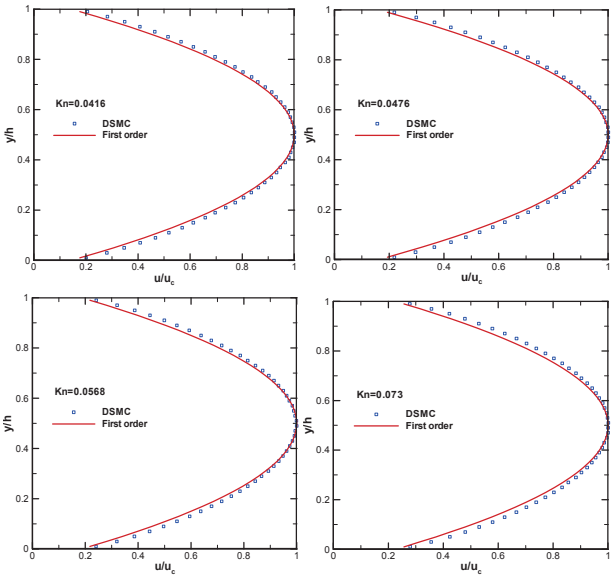


Figure 2. Comparison of nondimensional velocity profiles of micro-Poisuille flow, in different position.

$$\frac{u_s}{u_c} = Kn \Big/ \left(\frac{1}{4} + Kn \right) \qquad (18)$$

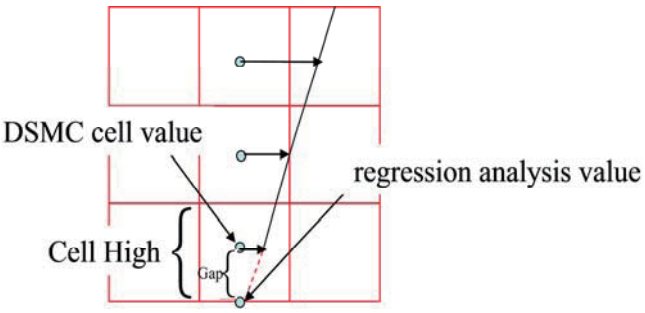


Figure 3. Schematic of DSMC meshes

Figure 3 demonstrates the DSMC cell value and regression analysis value distribution in the local mesh. In DSMC method, the slip velocity is usually obtained as the flow speed at the closest cell to the wall. Actually the slip velocity is the flow speed at the wall. The closest mesh to the wall is still having a gap with the wall that can not represent the slip velocity. Since the DSMC method is based on the statistical method, regression analysis can be used to derive the velocity at wall. This study tried three numerical interpolation methods such as Lagrange polynomial, quadratic polynomial, and least square method to better approximate the slip velocity at the wall.

Figure 4 shows the comparisons of DSMC calculated values of slip velocity from closest mesh, by three interpolation methods and the first order solution from Eq.(18). It illustrates that the values obtained by least square method agree best to the analytical first order solution.

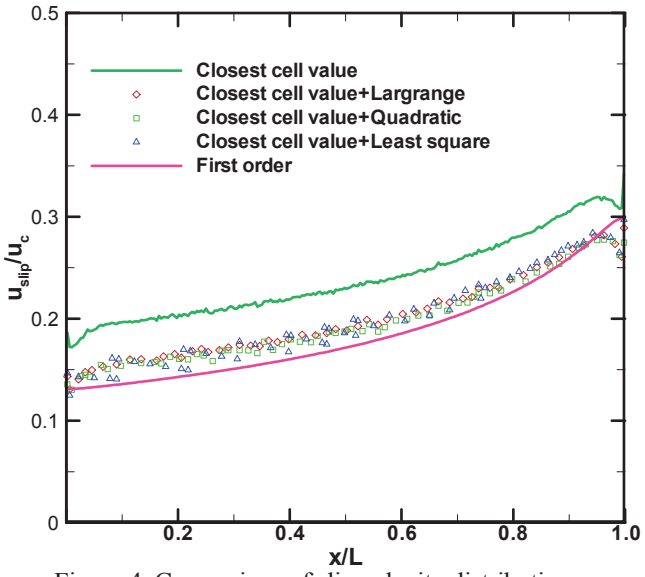


Figure 4. Comparison of slip velocity distribution.

In the preliminary trial of this numerical investigation, the solid boundary conditions with IS-BC and PS-BC in simulation procedure of DSMC are compared at higher Knudsen number regime. Inlet and exit stream boundary temperature configured as cases 2-4. The results of centerline velocity distribution as shown in Figure 5.indicated that the two boundary conditions are very close to each other. PS-BC is then selected for the following simulations because it needs less computer time.

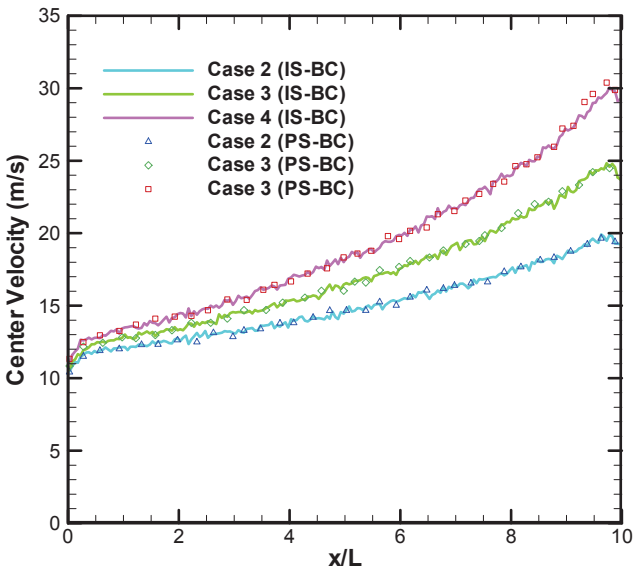


Figure5. Comparison of centerline velocity distribution for IS-BC and PS-BC

Figure 6 illustrates the centerline velocity distribution for cases 1-3 with PS-BC. For the adiabatic walls, it is thought that the increased velocity of the bulk flow along the channel direction is caused by coupling effects of pressure gradient, wall friction and inlet/outlet temperature difference. Increasing inlet/exit temperature difference notably gives rise the centerline velocity.

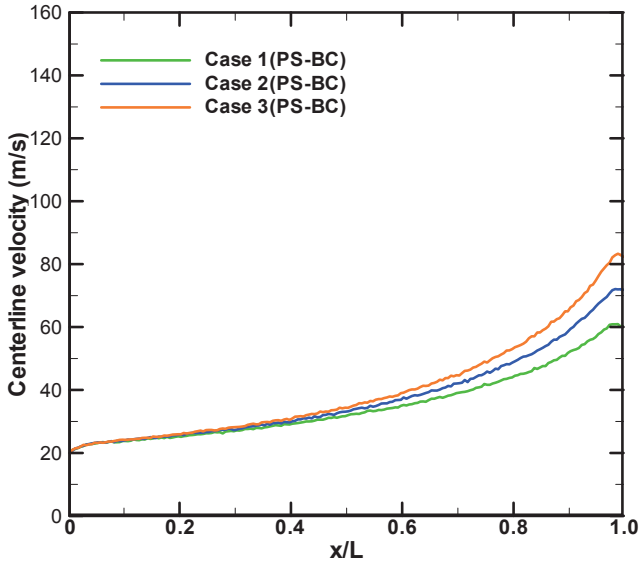


Figure 6. Centerline velocity distribution for cases 1-3 with PS-BC.

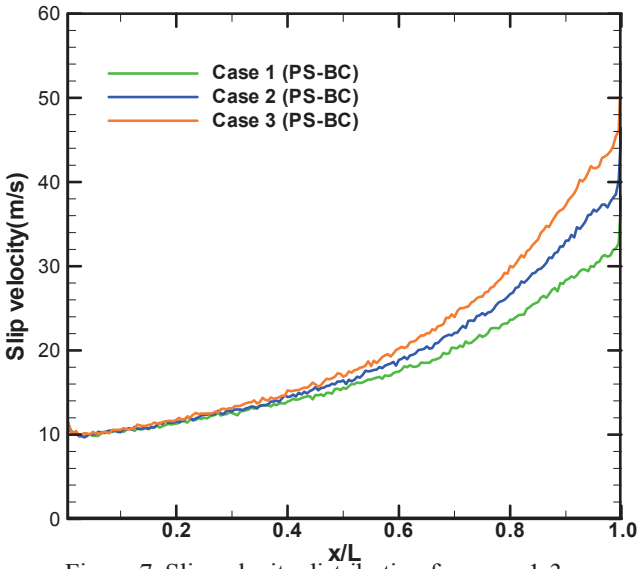


Figure 7. Slip velocity distribution for cases 1-3 with PS-BC

Figure 7 demonstrates the slip velocity distribution by least square regression for cases 1-3 with adiabatic walls. It is clear that the increase of the slip velocity along the channel direction is owing to the increase of local Kn and also the rising of inlet/outlet temperature difference. Cases 2 and 3 added the influence of tangential temperature gradient and the slip velocity associated with this particular factor is the so called thermal creep flow phenomena.

Figure 8 shows the centerline velocity distribution for cases 2-4 with linear temperature increment (LTI-BC) along the wall using diffuse reflection rule for DSMC calculation. Compared to adiabatic wall condition, the effect of wall heat transfer in these LTI-BC cases provides more energy for the mean flow in the channel and the values of centerline velocity are apparently higher than that in the corresponding PS-BC in Figure 6.

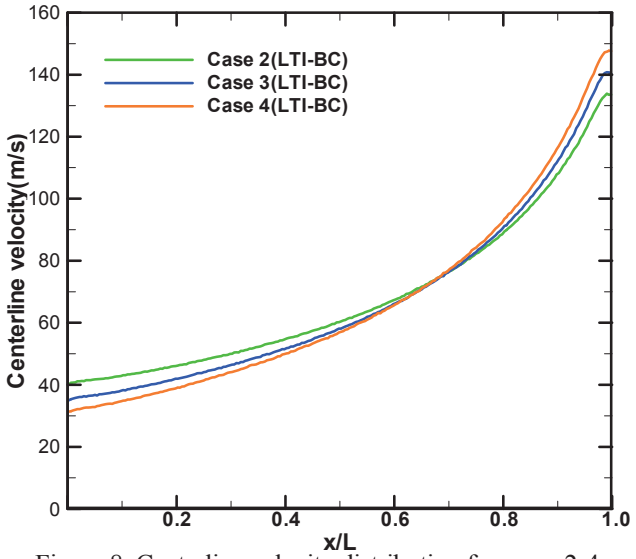


Figure 8. Centerline velocity distribution for cases 2-4 with LTI-BC.

Figure 9 demonstrates the slip velocity distribution for cases 2-4 with LTI-BC. The results show that slip velocity increases as temperature gradient increases. As comparing to the results for adiabatic wall shown in Figure 7, the wall heat transfer effect in these LTI-BC cases also gives more rapid increase of slip velocity along the channel wall. The thermal creep effect for LTI-BC is seen to be stronger than PS-BC, especially in downstream region.

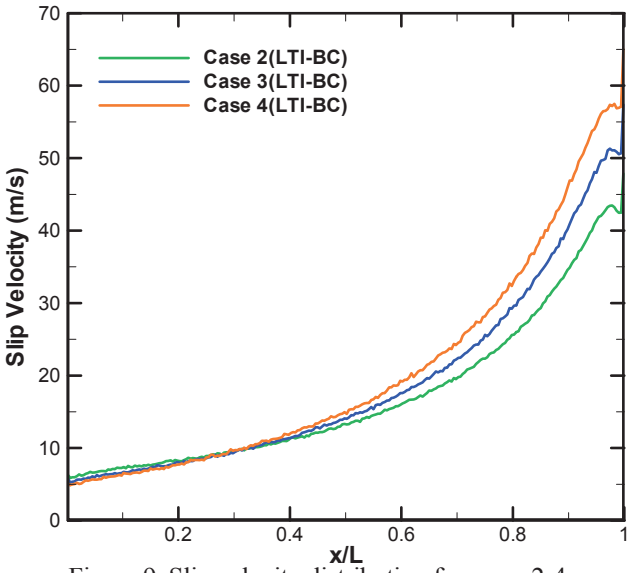


Figure 9. Slip velocity distribution for cases 2-4 with LTI-BC.

CONCLUSIONS

In the present research, The DSMC calculations are performed for microchannels with length to height ratio 10. The VHS molecule model and the theory of the particles flux conservation to modify the stream boundary conditions are employed. The simulated slip regime is obtained by adjusting the entrance and exit pressure. The results show that employing the least square method improved the accuracy of calculated value of slip velocity. The slip velocities are found in quantitative agreement with the trends of analytical first order solution. Under different solid boundary conditions, thermal effect of the high Knudsen number enhances the degree of rarefaction and the slip velocity. The flow properties such as centerline and slip velocities are strongly dependent on Kn, heat transfer and tangential temperature gradient along channel walls.

NOMENCLATURE

AR aspect ratio
A area, m^2
C molecular velocity, m/s
c_p specific heat, J/kg°C
Ec Eckert number
H channel heighy, μ m
Kn Knudsen number
k Boltzmann constant, J/K
L channel length, μ m
m mass, kg
n number density
$\dot{N}$ molecular number flux
P pressure, Pa
Re Reynolds number
s speed ratio
T temperature, K
$\triangle T$ temperature difference, K
u velocity, m/s
θ angle
γ specific heat ratio
λ mean free path, m
σ_v tangential momentum accommodation coefficients
ρ density, kg/m^3

Subscripts

c centerline
e exit
i inlet
m cell number
mp most probable
s slip velocity, m/s
w wall

ACKNOWLEDGEMENT

The study was partially supported by the National Science Council of the Republic of China through Grants NSC 101-2221-E-606-003.

REFERENCES

(1)Pfahler, J., Harley, J., Bau, H., and Zemel, J.N., (1991): "Gas and liquid flow in small channels" , *winter Annual meeting of the American Society of Mechanical system*, ASME, pp.49-60, .

(2)Liu, J.Q., Tai, Y.C., and Ho, C.M. , (1995): "MEMS for pressure distribution studies of gaseous flows in microchannels" , *IEEE international conference on micro electro mechanical systems*. Amsterdam, Netherlands, pp. 209-215, .

(3)Chen, C.S., Lee, S.M., and Sheu, J.D., (1998):"Numerical analysis of gas flow in microchannels," *Numerical Heat Transfer, Part A Applications*, Vol. 33, No. 7, pp. 749-762.
(4)Xue, H., Fan, Q., and Shu, C., (2000):"Prediction of micro-channel flows using direct simulation Monte Carlo," *Probabilistic Engineering Mechanics*, Vol. 15, No. 2, pp. 213-219.
(5)Arkilic, E.B., Breuer, K.S., and Schmidt, M.A., (1994):"Gaseous flow in microchannels," *ASME Fluids Engineering Div Publ Fed, ASME, NEW YORK, NY,(USA),* Vol. 197, pp. 57-66.
(6)Arkilic, E.B., Schmidt, M.A., and Breuer, K.S., (1997):"Gaseous slip flow in long microchannels," *Microelectromechanical Systems, Journal of*, Vol. 6, No. 2, pp. 167-178.
(7)Pong, K.C., Ho, C.M., Liu, J.Q., and Tai, Y.C., (1994):"Non-linear pressure distribution in uniform microchannels," *ASME-PUBLICATIONS-FED*, Vol. 197, pp. 51-51.
(8)Bird, G.A., (1994): Molecular gas dynamics and the direct simulation of gas flows, Clarendon Press . Oxford
(9)Piekos, E.S. and Breuer, K.S., (1996):"Numerical modeling of micromechanical devices using the direct simulation Monte Carlo method," *Journal of Fluids Engineering*, Vol. 118, No. 3, pp. 464-469.
(10)Oh, C.K., Oran, E.S., and Sinkovits, R.S., (1997):"Computations of high-speed, high Knudsen number microchannel flows," *Journal of Thermophysics and Heat Transfer*, Vol. 11, No. 4, pp. 497-505.
(11)Liou, W.W. and Fang, Y.C., (2001):"Heat transfer in microchannel devices using DSMC," *Microelectromechanical Systems, Journal of*, Vol. 10, No. 2, pp. 274-279.
(12)Fang, Y.C. and Liou, W.W., (2002):"Computations of the flow and heat transfer in microdevices using DSMC with implicit boundary conditions," *Journal of heat transfer*, Vol. 124, No. 2, pp. 338-345.
(13)Fan, J. and Shen, C., (2001):"Statistical simulation of low-speed rarefied gas flows," *Journal of computational Physics*, Vol. 167, No. 2, pp. 393-412.
(14)Hassan, M. L. and Esmail, N. (2006):"DSMC simulation of subsonic flows in parallel and series microchannels," *Journal of Fluids Engineering*, Vol. 128, No. 6, pp. 1153-1163.
(15)Kursun, U. and Kapat, J.S., (2007):"Modeling of microscale gas flows in transition regime Part I: Flow over backward facing steps," *Nanoscale and microscale thermophysical engineering*, Vol. 11, No. 1-2, pp. 15-30.
(16)Bird, G.A., Gallis, M.A., Torczynski, J.R., and Rader, D.J., (2009):"Accuracy and efficiency of the sophisticated direct simulation Monte Carlo algorithm for simulating noncontinuum gas flows," *Physics of Fluids*, Vol. 21, pp. 017103.
(17)Gallis, Michail A., Torczynski, J.R, Rader, D.J., and Bird, Graeme A, (2009):"Convergence behavior of a new DSMC algorithm," *Journal of computational Physics*, Vol. 228, No. 12, pp. 4532-4548.
(18)Ikegawa, M. and Kobayashi, J., (1990):"Development of a Rarefield Gas Flow Simulator Using the Direct-Simulation Monte Carlo Method: 2-D Flow Analysis with the Pressure Conditions Given at the Upstream and Downstream Boundaries," *JSME international journal. Ser. 2, Fluids engineering, heat transfer, power, combustion, thermophysical properties*, Vol. 33, No. 3, pp. 463-467.
(19)Nance, R.P., Hash, David B., and Hassan, H.A., (1998):"Role of boundary conditions in Monte Carlo simulation of microelectromechanical systems," *Journal of Thermophysics and Heat Transfer*, Vol. 12, No. 3, pp. 447-449.
(20)Wu, J.S, Lee, F., and Wong, S.C, (2001):"Pressure Boundary Treatment in Micromechanical Devices Using The Direct Simulation Monte Carlo Method," *JSME International Journal Series B*, Vol. 44, No. 3, pp. 439-450.
(21)Tzeng, P.Y, Soong, C.Y, Liu, M.H, and Yen, T.H, (2008):"Atomistic simulation of rarefied gas natural convection in a finite enclosure using a novel wall-fluid molecular collision rule for adiabatic solid walls," International Journal of Heat and Mass Transfer, Vol. 51, No. 3, pp. 445-456.
(22)Tzeng, P.Y, Chou, I.W, Liu, C.H, and Li, W.K, (2011):"Improvement of the Gas-Surface Collision Rule for Adiabatic Walls in DSMC Modeling of Rarefied Gas Convection in a Micro Enclosure," Journal of Aeronautics, Astronautics and Aviation, Vol. 43, No. 3, pp. 147-157.
(23)Beskok, A.K., George E., and Trimmer, W., (1996):"Rarefaction and compressibility effects in gas microflows," *Journal of Fluids Engineering*, Vol. 118, No. 3, pp. 448-456.