Reconsideration of Flows through Constriction Microchannels Using the DSMC Method

A.A. Alexeenko*, S.F. Gimelshein†, and D.A. Levin‡,
Pennsylvania State University, University Park, PA 16802

Abstract
Gaseous flow through a microchannel is treated numerically and analytically in order to assess pressure and mass flow losses due to a constriction of a finite length. Numerical modeling of microchannel flow in the slip and transitional regimes is carried out using the direct simulation Monte Carlo (DSMC) method. The prediction of pressure losses and mass flow based on a simple analytic model for constriction microchannel flow are in excellent agreement with DSMC simulations.

Nomenclature

a - speed of sound, m/sec  
\( \dot{m} \) - mass flow rate, kg/s  
p - pressure, Pa  
\( \mathcal{P} \) - inlet to outlet pressure ratio  
u - X-component of velocity, m/sec  
v - Y-component of velocity, m/sec  
R - gas constant, J/kg·K  
T - temperature, K  
\( \rho \) - density, kg/m³  
\( \sigma \) - accommodation coefficient

Subscripts

int - interior cell  
\( \varepsilon \) - boundary  
in - inlet  
out - outlet

1 Introduction
Experimental and numerical investigation of micro- and nanoscale flows has drawn much attention over the last few years. Most of the areas of interest are related to microchannels, micropumps, and micronozzles, with their subsequent application to propulsion devices, chemical and pressure sensors, and laminar flow control (Ref. [1] provides a comprehensive review of the field). A channel with constrictions and contraction/expansion transitions is an important geometry that is a common integral part of many micromechanical devices. The goal of this work is the numerical modeling of flows in microchannels aimed at studying the impact of constrictions in microchannels on the mass flux and pressure losses in nonreacting, subsonic flows.

The study of flows through microchannels with constrictions has been discussed by a number of researchers. Karniadakis and Beskok[2] studied the flows over a backwards facing step, through a grooved channel, and into a cavity. Zohar and coworkers[3]-[4] measured pressure and mass flow rates through microchannel devices with contraction and expansion sections as well as constrictions. Beskok et al[5] discussed the rarefaction and compressibility effects in microflows emphasizing that both need to be considered if one is to correctly model the physics of micro flows. Although the specific configurations of the microchannel devices differ among these researchers,[2]-[5] the flows share common features in that they were subsonic, of relatively low Reynolds number, and have a Knudsen number in the slip-flow regime (typically between 0.01 and 0.5). It was recognized that due to the finite Knudsen number, the flow field macroparameters, such as pressure, temperature, and mass flow rate, could not be calculated with the standard continuum, Navier-Stokes formalism. In terms of kinetic approaches, analytic formulas for the free molecular regime are not generally applicable either. However, the Navier-Stokes equations may be used with reasonable accuracy if the surface velocity-slip conditions are appropriately modified.

Direct Simulation Monte Carlo (DSMC) method is rigorous because the gas-surface interaction is
modeled without approximation. For this reason, DSMC calculations were performed\textsuperscript{2} for selected microchannel configurations, but the primary emphasis of these calculations was to validate slip-flow correction models rather than to use DSMC as a simulation tool to study the physics of the flow. Finally, the analytic expressions of Arkilic \textit{et al.}\textsuperscript{17} for \textit{ṁ}_{\text{base}} flow rate and \textit{p}_{\text{base}} that include rarefaction effects such as \textit{v}elocity and temperature jump conditions at the gas-wall boundary and compressibility effects were used to interpret full numerical simulations as well experimental data.\textsuperscript{3}-\textsuperscript{7}

The emphasis on predicting mass flow rate and pressure through microchannel systems is to establish a design criteria similar to that of macro devices. In macro systems the total head loss for a system of components with straight and bended or branching portions is well established in terms of tabulated major and minor losses. A concern exists in the design of microdevices that if the flow physics is not entirely understood, it may be difficult to predict whether a specific configuration (composed of straight sections coupled to sections containing constrictions) is optimized from the point of view of minimizing pressure losses. Based on the macrosystem analog, flow separation in microsystem flows containing constring or bend components could potentially cause large mass flow or pressure losses.\textsuperscript{4} Hence, more detailed knowledge about the microflow in terms of accurate velocity fields that illustrate the occurrence of flow separation and the formation and structure of recirculation zones is crucial.

To that end, researchers have tried to elucidate the nature of flow separation and recirculation that can occur in low Reynolds number, subsonic to transonic flows in microdevices. The flow separation is understood to be steady and due, in some nature, to the need for the flow to turn to accommodate a sharp corner ("geometric")\textsuperscript{4, 2, 8}. During this process an adverse pressure gradient can develop, creating a flow condition that may precede separation. However, it was also noted that for such low Reynolds numbers the flow should exhibit ideal, Hele-Shaw type, features with no flow separation even from sharp corners.\textsuperscript{10} Simulation predicted flow separation in shear-driven, grooved microchannels\textsuperscript{5} and a backwards step\textsuperscript{8} even for low Reynolds numbers, but direct experimental evidence with particle velocity imaging techniques was ambiguous. Hence it has been difficult to sort out the effects of rarefaction and compressibility factors that may lead to flow separation and the degree of flow recirculation. An important reason for performing DSMC calculations is that the gas-surface interaction may be exactly specified and controlled to allow one to assess the impact of rarefaction on recirculation. It will be shown in the DSMC calculations of this work that for subsonic flows, recirculation will occur due to classical boundary layer arguments.

A series of investigations were undertaken to assess "minor" losses in microflows through non-parallel plate channel configurations.\textsuperscript{3}-\textsuperscript{7} The work of Zohar and coworkers\textsuperscript{3}-\textsuperscript{7} combined MEMS microchannel component fabrication with measurements of not only mass flow rate, but pressure as well. These two simultaneous measurements provide the modeler with important, redundant data that allows one to check for consistency. In particular it was found that in constriction microchannels, the pressure drop just downstream of the constriction decreased to a level much larger than predicted values.\textsuperscript{4} The authors, however, were not able to reconcile the large pressure drop with classical flow physics and an explanation for the discrepancy is not provided. It will be shown in this work that the existence of flow separation and the measured pressure drops are consistent. A simple model will be proposed to analyze the pressure distributions for a microchannel with constrictions and its accuracy will be assessed by comparison with exact DSMC simulations.

## 2 DSMC Method

The DSMC method\textsuperscript{11} has been applied in this work to obtain numerical solutions for low Reynolds number microchannel flows. The DSMC method is a numerical approach that is feasible for solving the Boltzmann equation in the near continuum to free molecular flow regimes. The DSMC-based software SMILE \textsuperscript{12} is used for all computations. The code uses the \textit{majorant} frequency scheme\textsuperscript{13} of the DSMC method for modeling of collisional process. The intermolecular potential is assumed to be the variable soft sphere model\textsuperscript{14}. Although, vibrational and rotational energy exchange does not play a significant role at the conditions of interest here, the Larsen-Borgnakke model\textsuperscript{15} with temperature-dependent \textit{Z}_r and \textit{Z}_0 and discrete rotational and vibrational energies is used for the energy exchange between the translational and internal modes.

The more important physical mechanism in the low-speed microchannel flows is the gas-surface interaction. The Maxwell model is used here to model the momentum and heat transfer to the wall. The model assumes that a fraction \((1 - \alpha_0)\) of incident
particles is reflected specularly while the remaining fraction α_d experiences a diffuse reflection on the wall. The diffuse reflection implies that the particle's translational and internal energies are distributed according to the Maxwell-Boltzmann distribution regulated by the energy accommodation coefficient α_E. The parameter α_d is the tangential momentum accommodation coefficient

\[ \alpha_d = \frac{P_{tr} - P_{rr}}{P_{tr}}, \]

where \( P_{tr} \) is the tangential momentum and the subscripts \( r \) and \( r \) refer to the incident and reflected particles, respectively. The energy accommodation coefficient \( \alpha_E \) is

\[ \alpha_E = \frac{E_i - E_r}{E_i - E_w}, \]

where \( E_i \) is the energy that the reflected molecule would acquire if there was a thermal equilibrium between the wall and the gas. The full accommodation of tangential momentum, \( \alpha_d = 1 \), and energy, \( \alpha_E = 1 \), was assumed with the constant wall temperature of 300 K.

3 Subsonic Inlet and Outlet Boundary Conditions

3.1 Implementation

The numerical simulation of a subsonic flow in a pressure-driven microchannel requires a special treatment of the inlet and outlet boundary conditions. If the flow is in the subsonic regime then the one-dimensional characteristic theory dictates that there will be two incoming characteristic lines along which information propagates at the inlet and one incoming characteristic line at the outlet. Therefore, only two flow parameters out of three (pressure, temperature and velocity) can be freely specified at the inlet and only one at the outlet. On the other hand, in the DSMC method all three flow parameters must be specified for incoming molecules at the domain boundaries, i.e. density, temperature and velocity. In this case, boundary conditions have to use flow properties in the interior flow domain.

The implementation of subsonic inlet and outlet boundary conditions for various computational fluid dynamics (CFD) methods has been intensively studied in the past. In the present implementation, the Whitfield characteristic formulation was used in the cell subject to condition that the pressure in the cell is larger than the specified outlet pressure. Otherwise, extrapolation of velocity and temperature has been applied. The inlet and outlet conditions used in DSMC are updated based on the above procedure every 20,000 time steps to avoid statistical scatter in the flow macroparameters in the interior cells.

3.2 Validation

To validate the implementation of the subsonic boundary conditions in the DSMC method, a test case of nitrogen flow in a straight microchannel has been calculated and compared with the analytic approximate solution. The microchannel aspect
ratio is 30 and Knudsen number based on the outlet conditions is 0.05. The pressure ratio between inlet and outlet is 2.47. The inlet temperature of the gas is 300 K which corresponds to a value of the viscosity coefficient of $1.77 \times 10^{-5}$ kg/m·s based on the hard-sphere model with a viscosity-temperature exponent of 0.24 and molecular diameter of $4.17 \times 10^{-10}$ m. The large aspect-ratio microchannel is in the slip-flow regime and the analytic approximate solution for such flow was obtained by Arkilic et al.[17].

Application of a numerical method to solve practical problems requires a reliable way to estimate the accuracy of the solution. The DSMC numerical solution depends on three parameters: the cell size $Ax$, time step $At$, and the number of particles $N_A$ in a volume with linear size equal to the local mean free path.[23] In the DSMC algorithm, the cell size has to be less than the local mean free path and the time step should be less than the average time between collisions. However, a more strict requirement is that the number of simulated particles has to be large enough to make the statistical correlations between particles insignificant.

The principal objective of the validation study presented below was to examine the sensitivity of the results to the above numerical parameters of the DSMC simulation and to compare the computed flow parameters with the analytic predictions.

The calculations with different time steps showed that the results do not change when $At$ is decreased from $5 \times 10^{-5}$ s to $1 \times 10^{-5}$ s. The time step of $3 \times 10^{-5}$ s was chosen for all calculations presented hereafter. The cell size was found to have a larger impact on the DSMC solution. The influence of the number of cells on the streamwise velocity profile along the axis is shown in Fig. 1 for 1.6 million particles. There is a difference of about 10% when the number of cells is changed from 50,000 to 100,000. The solution does not change when the number of cells is further increased to 200,000. Note that the pressure distribution is practically unaffected by the number of cells. As shown in Fig. 2 the maximum pressure difference amounts to less than 1%.

The most important parameter for the accuracy of the DSMC method is the number of simulated molecules. In the present validation study, the number of simulated molecules has been varied from 0.2 million to 12.8 million. Similar to the influence of the number of cells, the pressure distribution is insensitive to the variation of the number of molecules. The maximum difference between the results for 0.2 million and 12.8 million was less than 2%. The most sensitive parameter to the number of molecules is the streamwise velocity. Figure 3 shows that the streamwise velocity changes when the number of molecules is less than 3.2 million. The difference between the solution for 0.2 million and 3.2 million is over 20%.

The main reason for the strong dependence of the solution on the number of particles is related to the correlations between simulated particles. Such correlations always exist in the system of a finite number of particles used in DSMC modeling.[22] The magnitude of these correlations depends on the total number of simulated molecules in the system, and generally decreases when the number of molecules increases. It is necessary to estimate the level of statistical dependence between the simulated particles and its contribution to the results of DSMC computation. A significant level of statistical dependence, or particle correlations, means that the molecular chaos hypothesis, used in the Boltzmann equation, is no longer valid.

An important criterion that allows for a practical verification of the presence of the statistical dependence between simulated particles is the relative number of repeated collisions.[23] Repeated collisions are collisions between the same pair of particles during their lifetime in the computational domain. The lifetime of a molecule in DSMC modeling is determined by the time the molecule was introduced into the computational domain or reflected from a diffuse wall and the time it left the domain or collided with a diffuse wall. The number of repeated collisions was calculated by checking if the current collision partner of a marked particle coincides to one of the last four of its collision partners.

The number of repeated collisions is closely related to the number of particles $N_A$ in a volume with the linear size equal to the local mean free path. If $N_A \gg 1$, one can say that the simulation results are close to the solution of the Boltzmann equation. The number of repeated collisions may also depend on the time step and the number of particles per collision cell.

The flow under consideration is characterized by low average velocities and, as a result, relatively large particle lifetimes. The number of repeated collisions may therefore be larger than that in a typical supersonic flow with same $N_A$. The results presented in Table 1 for the test validation case discussed above show the average number of repeated collisions for various numbers of molecules and cells. From the comparison with Figs. 1-3, it can be seen that the number of repeated collisions in this flow has to be less then 5% for the results to be accurate within 2%.
Finally, the calculated DSMC macroparameters are compared to the analytic solution for a straight microchannel flow [17]. The total number of cells in the DSMC calculation was 200,000 with about 12.8 million simulated particles. The pressure distribution along the channel is given in Fig. 4. The difference between the two solutions does not exceed 1%. The comparison of DSMC computed and analytic contours of X-component of velocity are plotted in Fig. 5 and also shows excellent agreement. All the calculations presented in the following sections were performed with the same $N_\lambda = 256$.

<table>
<thead>
<tr>
<th>Number of particles, million</th>
<th>Number of cells, $10^3$</th>
<th>$N_\lambda$</th>
<th>Percentage of repeated collisions</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>80</td>
<td>4</td>
<td>32</td>
</tr>
<tr>
<td>0.8</td>
<td>160</td>
<td>16</td>
<td>8</td>
</tr>
<tr>
<td>1.6</td>
<td>50</td>
<td>32</td>
<td>2.5</td>
</tr>
<tr>
<td>1.6</td>
<td>100</td>
<td>32</td>
<td>3.5</td>
</tr>
<tr>
<td>1.6</td>
<td>200</td>
<td>32</td>
<td>5</td>
</tr>
<tr>
<td>3.2</td>
<td>200</td>
<td>64</td>
<td>3</td>
</tr>
<tr>
<td>6.4</td>
<td>200</td>
<td>128</td>
<td>2.5</td>
</tr>
<tr>
<td>12.8</td>
<td>200</td>
<td>256</td>
<td>2</td>
</tr>
</tbody>
</table>

1 Straight channel, $Kn_{out}=0.05$, $\kappa_n=2.47$, $\Sigma_i=30$.

4 Microchannels with Constriction: Theory

To obtain the distribution of the flow parameters in a microchannel with constriction in the form of a long microchannel (or a transition section of another shape), one can consider it as three different flow sections. The first one is a straight microchannel flow upstream of the constriction, the second one is the flow in the transition section, and, finally, the straight microchannel flow downstream of the constriction.

For the flow in a microchannel with constriction, the known parameters are inlet and outlet pressures, viscosity coefficient and Knudsen number at the exit as well as geometrical shape of the three sections. A schematic of a microchannel with constriction and notations used here are given in Fig. 6. The pressure downstream and upstream of the constriction are unknowns. We will denote them $p_1$ and $p_2$ (see Fig. 6).
Figure 3: Streamwise velocity distribution along the channel, $Kn_{out}=0.05$, $L/H=30$ for different numbers of particles.

Figure 4: Normalized pressure distribution along the channel, $Kn_{out}=0.05$, $L/H=30$.

Figure 5: Streamwise velocity (m/s) contours, $Kn_{out}=0.05$, $L/H=30$. Analytic solution [17]- dashed lines, DSMC solution - solid lines.

The flow quantity that has to be conserved in all three sections is the mass flow. The mass flow conservation yields a system of three non-linear algebraic equations for the unknown mass flow, $m^*$, and two unknown pressures, $p_1$ and $p_2$. The mass flow in a dimensional form for a high-aspect ratio straight microchannel for a given pressure ratio is equal to [17]:

$$m = \frac{H^3wp_{out}^2}{24\mu LRT} (P - 1 + 12\sigma Kn_{out}(P - 1)) \quad (4)$$

Applying the mass flow formula for each section, we get the system:

$$\dot{m}(p_{in,1}, L_1, H_1, Kn_1) = m^* \quad (5)$$

$$\dot{m}(p_1, p_2, L_2, H_2, Kn_2) = m^* \quad (6)$$

$$\dot{m}(p_2, p_{out}, L_3, H_3, Kn_0) = m^* \quad (7)$$

where $Kn_1 = Kn_{out}p_{out}/p_1$ is the Knudsen number at the location just upstream of the constriction. Similarly, $Kn_2 = Kn_{out}p_{out}/p_2 H_3/H_2$ is the Knudsen number downstream of the constriction based on the constriction height.

Then non-linear algebraic system can be solved by an optimization method. The least square method was used and iterations were carried out until the agreement for mass flow up to fifth significant digit was reached.
5 Microchannels with Constriction: Simulation

The DSMC calculations of low-speed microchannel flows with constriction of a finite length have been carried out in order to study the influence of the constriction on the pressure loss, mass flow rate and the overall flow structure in the channel. The results of the DSMC simulations are then compared with the simple model of constriction. The flow conditions and geometries of the microchannels are summarized in Tables 2 and 3.

Table 2: Geometric setup

<table>
<thead>
<tr>
<th>Shape</th>
<th>H2/H1</th>
<th>H3/H1</th>
<th>L1/H1</th>
<th>L2/H1</th>
<th>L3/H1</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.25</td>
<td>1</td>
<td>14.5</td>
<td>1</td>
<td>14.5</td>
</tr>
<tr>
<td>2</td>
<td>0.25</td>
<td>1</td>
<td>8.5</td>
<td>3</td>
<td>8.5</td>
</tr>
</tbody>
</table>

Table 3: Summary of cases considered

<table>
<thead>
<tr>
<th>Notation</th>
<th>p_in/p_out</th>
<th>K_nout</th>
<th>Shape</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>2.47</td>
<td>0.05</td>
<td>1</td>
</tr>
<tr>
<td>Case 2</td>
<td>2.53</td>
<td>0.05</td>
<td>2</td>
</tr>
<tr>
<td>Case 3</td>
<td>2.47</td>
<td>0.1</td>
<td>2</td>
</tr>
<tr>
<td>Case 4</td>
<td>1.25</td>
<td>0.05</td>
<td>2</td>
</tr>
</tbody>
</table>

5.1 Pressure Losses

Let us first consider Case 1 which corresponds to the same total pressure loss as in the case of a straight microchannel that was used for validation of the boundary conditions. The presence of the constriction significantly changes the pressure distribution inside the channel. The pressure distribution along X-axis is plotted in
Fig. 7 along with that for a straight channel and a theoretical prediction of Eq. (5)-(7). The DSMC calculated pressure distribution agrees very well with the theoretical prediction. The maximum difference is immediately downstream of the constriction but still is within 3% of the computed value. The pressure loss at the constriction section is about 60% of the total pressure loss and, therefore, can not be considered a "minor" loss.

Figure 9 shows the comparison of theoretical and simulated DSMC pressure distributions along the X-axis for the other cases given in Table 3. In all three cases, pressure values are in excellent agreement. Therefore, the flow in a microchannel with a constriction of finite length can be considered as three different microchannel flows and Eqs. (5)-(7) can be used to calculated the pressure loss in the constriction section.

5.2 Flow Structure

Let us now consider the flow structure near the constriction section as predicted by the DSMC simulations. Figure 11 shows the pressure contours near the constriction section. There is a small adverse pressure gradient at the corners downstream and upstream of the constriction. The pressure isolines are normal to X-axis everywhere in the flow except the inlet and outlet of the constriction section where the isolines have a nearly circular shape as the flow becomes fully developed. Overall, the flow structure in the microchannel with constriction of a finite length is very well approximated by three different channel flows.

The streamlines and contours of X-component of velocity are shown in Fig. 10 for Case 1. Far from the constriction, the streamlines of the flow are parallel to X-axis and the value of the streamwise velocity is about 12 m/sec at the centerline.

The flow in the microchannel with constriction separates in the transition section similarly to a flow around forward and rearward steps. The two separation zones are clearly identified at the corners immediately upstream and downstream of the constriction section. The size of the separation zone upstream is equal to about half of the constriction height. The separation zone downstream is slightly larger due to a larger flow velocity. The velocity in the constriction section increases to approximately 60 m/sec at the centerline. Far downstream of the constriction the value of the X-component of the velocity decreases to 22 m/sec.

A similar flow structure is observed in the other computed cases. The streamlines and X-component of velocity is plotted in Figs. 12-14 for Cases 2-4. In Cases 2-4 the size of the separation zone upstream of the constriction is less than in Case 2 due to a much lower velocity.

The flow separation observed in the four cases may be understood in terms of standard boundary layer theory. To illustrate this, Case 1 was re-run assuming a fully specular gas-surface interaction for the upstream channel. Although such a situation is physically unrealistic for microchannel flows, the simulation corresponds to one of ideal flow. In this case, no flow separation is observed and the streamlines follow the corner boundaries. Hence in the real transitional subsonic microchannel flow, recirculation will occur even for low Reynolds numbers.

5.3 Mass Flow Losses

The mass flow of in a microchannel for a specific inlet to outlet pressure ratio is an important characteristic for design of micro-devices. The mass flow of a microchannel with constriction can be significantly smaller
compared to that of a straight microchannel, $\dot{m}_{\text{un}}$, for the same pressure ratio. A monotonic decrease of the mass flow with the decreasing constriction-gap width has been observed experimentally[4].

Theoretical and calculated mass flow of a constriction microchannel divided by that of a corresponding straight microchannel is listed in Table 4 for Cases 1-4. Both theoretical and calculated mass flow in a constriction microchannel indicate very large mass flow losses compared to straight microchannel.

The results of the calculation agree within a few percent with the theoretical prediction for Cases 1, 2, and 4. The maximum difference (18%) between theory and DSMC is in Case 3, where the flow Knudsen number at the outlet is 0.1 and therefore, the slip flow approximation used in Eq. 5-7, may no longer be accurate.

Table 4: Theoretical and calculated mass flow losses

<table>
<thead>
<tr>
<th>Case</th>
<th>Theory $\dot{m}/\dot{m}_{\text{un}}$, %</th>
<th>DSMC $\dot{m}/\dot{m}_{\text{un}}$, %</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40.6</td>
<td>39.9</td>
</tr>
<tr>
<td>2</td>
<td>13.2</td>
<td>13.9</td>
</tr>
<tr>
<td>3</td>
<td>15.8</td>
<td>18.7</td>
</tr>
<tr>
<td>4</td>
<td>14.8</td>
<td>15.2</td>
</tr>
</tbody>
</table>

6 Conclusions

The subsonic gas flow through a microchannel with a finite-length constriction is studied with the goal of predicting the significant pressure and mass flow rate losses observed in earlier experimental studies. An analytic model is proposed that allows one to predict the pressure and mass flow losses due to the constriction. The model is validated by comparison with the numerical results obtained using the direct simulation Monte Carlo method. The agreement between the predicted analytic and DSMC solutions is found to be very good with the maximum difference smaller than a few percent.

The analysis of the sensitivity of the DSMC results to numerical parameters was also conducted, and the large impact of correlations between simulated particles in the low-speed flows under consideration was shown. The DSMC results indicate that the flow in the microchannel with constriction separates in the transition section similarly to a flow around forward and rearward steps. However, the separation region does not significantly impact the pressure distribution along the channel.

The pressure loss in the transition section is as large as 60% and, therefore, cannot be considered a minor loss. The mass flow rate ratio between the constriction and straight channel varies from 15 to 40% for the considered cases. Therefore, the constriction has a dramatic effect on the major system design parameters for a pressure-driven microchannel flow.

7 Acknowledgment

The research at Penn State University was supported by the Army Research Office Grants DAAG55-98-1-009 and DAAD19-02-1-0196. These programs are supported by the Science and Technology Directorate of the Missile Defense Agency. We would like to thank Dr. Clifton Phillips of SPAWARSYSCEN who provided us with computer time on DoD Maui High Performance Computing Center. The authors want to acknowledge Prof. William Liou for helpful conversations.

References

Figure 12: X-component of velocity contours and streamlines, Case 2.

Figure 13: X-component of velocity contours and streamlines, Case 3.

Figure 14: X-component of velocity contours and streamlines, Case 4.


