1D HOMOGENEOUS MODELING OF MICROCHANNEL TWO-PHASE FLOW WITH DISTRIBUTED LIQUID WATER INJECTION FROM WALLS

Sébastien Vigneron, Carlos H. Hidrovo, Fu-Min Wang, Eon-Soo Lee, Julie E. Steinbrenner, Theresa A. Kramer, John K. Eaton, Kenneth E. Goodson
Mechanical Engineering Department
Stanford University

ABSTRACT
This paper presents a theoretical model and a numerical simulation of a liquid-gas two-phase flow within a microchannel ($50 \mu m \times 500 \mu m \times 2 cm$) equipped with distributed liquid water injection through the side walls. The modeling and solution of the conservation equations provide pressure drop as a function of inlet velocity. The influence of different parameters involving water injection is investigated, such as the quantity of water that is injected and the profile that is used to inject it. The numerical results show that for small water injection rates ($1-10 \mu L/min$) the air flow velocity and pressure drop are not significantly perturbed by the presence of liquid water. But if water injection becomes important ($10-100 \mu L/min$) larger pressure drops are observed. The influence of inlet pressure is also investigated.

The model predictions are compared with experimental results obtained from testing a set of microchannels with a varying number of water injection slots on the side walls. Pressure drop distribution data from these experiments are consistent with model predictions.

INTRODUCTION
Two-phase flow in microchannels has recently attracted attention because of its wide applicability to technologies such as MEMS, integrated circuits cooling [1], chemical process engineering, medical/genetic engineering and bioengineering among others. For example, gas-liquid flow and transport is a subject of increasing importance in low-temperature Proton Exchange Membrane (PEM) fuel cells [2]. A good way to improve their performance is to use microchannels ($0.05 - 1 mm$) in the anode and cathode gas delivery systems [3], because this reduces species transport resistances within the channels. However, microchannels substantially complicate the issue of water management [4-6], particularly on the cathode side, where oxygen is consumed and water produced. This aspect is still poorly understood and needs to be investigated.

The present paper describes a theoretical model and a numerical simulation of a liquid-gas two-phase flow within a microchannel ($50 \mu m \times 500 \mu m \times 2 cm$) equipped with distributed liquid water injection through the side walls. The modeling and solution of the conservation equations provide pressure drop as a function of inlet velocity. The influence of different parameters involving water injection is investigated, such as the quantity of water that is injected and the profile that is used to inject it. The numerical results show that for small water injection rates ($1-10 \mu L/min$) the air flow velocity and pressure drop are not significantly perturbed by the presence of liquid water. The influence of the inlet pressure is also investigated. The model predictions are compared with experimental results obtained from testing a set of microchannels with a varying number of water injection slots on the side walls. Pressure drop distribution data from these experiments are consistent with model predictions.

PROBLEM STATEMENT
We consider a rectangular microchannel of 500 microns width, 50 microns depth and 2 centimeters length. An air flow is setup at the inlet of the channel. Between the inlet and the outlet liquid water is injected into the flow. The liquid water enters the gas channel through a uniform distribution of slots along the side walls. In other words we are considering a bi-component (air/water) two-phase (gas/liquid) flow inside a rectangular channel. We consider a one-dimensional formulation of this problem, so the liquid water injection is treated as if water was created in the core of the flow (as a source term in the equations). The flow is pressure driven (Poiseuille flow) so the value of the stagnation pressure at the inlet of the channel is an important parameter.
Our goal is to study the effects of water injection on the characteristics of the microchannel two-phase flow in comparison with a single-phase flow. We investigate the role of liquid water flow rate on the pressure drop across the channel, as well as the influence of the water injection spatial profile on pressure drop. We also study the influence of the inlet absolute pressure on the flow distribution. An interesting fact is that because we are considering a compressible flow inside a constant area channel, the flow accelerates along the channel. This means that the flow may choke at the outlet of the channel.

**ONE-DIMENSIONAL HOMOGENEOUS FLOW MODEL**

There are two approaches to modeling liquid-vapor two-phase flows. The first treats the flow as a homogenous fluid [7] with properties that are the mass weighted averages of the local properties of the liquid and the vapor. These properties are used to evaluate the incompressible Navier-Stokes and energy equations for a single fluid. The homogeneous model limits the physics that can be incorporated into the model. The two phases are considered to be uniformly distributed within each grid element. Local temperature and pressure differences between phases can not be captured by treating the two phases as a homogenous fluid. A two-fluid (separated flow) model treats the liquid and vapor phases separately. Mass, momentum, and energy conservation equations are required for each phase. In addition, closure models are required to account for the transfer of the mass, momentum, and energy between phases. This model is significantly more detailed and complex than the homogeneous model. The goal of the present work is to determine the extent to which the simpler homogeneous flow model can predict the influence of liquid water injection on the channel flow.

**Model Assumptions**

We consider a one-dimensional model of the above problem: fluid properties are averaged at each cross section along the axis of the channel and thus vary only in the direction of the flow. By doing this, we accept that we will not be able to get any information on the variation of properties normal to the flow direction. However this is a reasonable approximation since water injection will primarily alter the flow characteristics in the channel direction.

We also define area averaged values of velocity, density, enthalpy, and friction factor of the flow:

\[
\bar{u} = \frac{1}{A} \int_A u \, dA
\]

\[
\bar{\rho} = \frac{1}{A} \int_A \rho \, dA
\]

\[
\bar{h} = \frac{1}{A} \int_A h \, dA
\]

\[
\bar{\tau}_w = \frac{1}{P} \int_P \tau_w \, dl
\]

where \(A\) and \(P\) are respectively the cross sectional area and the perimeter of the channel.

These averaged variables vary only according to the coordinate \(z\) along the channel.

Then, as discussed above, we treat the two-phase flow as a homogeneous fluid. A homogeneous flow is defined as one where temperature is constant and liquid and gas velocities are equal over the flow cross-sectional area. The two phases also see the same pressure. The properties of the two-phase mixture are expressed as mass-weighted averages of the properties of the liquid and vapor phases.

In the governing equations, the mixture variables and properties are thus defined as:

- Homogeneous density:

\[
\bar{\rho} = \frac{1}{A} \int_A \rho \, dA
\]

- Homogeneous enthalpy:

\[
\bar{h} = \frac{1}{A} \int_A h \, dA
\]

- Homogeneous viscosity:

\[
\bar{\tau}_w = \frac{1}{P} \int_P \tau_w \, dl
\]

In the frame of the homogeneous assumption, there is a explicit relation between \(\bar{x}\) and \(\bar{\alpha}\), the average gas volume fraction:

\[
\bar{\alpha} = \frac{A_g}{A_g + A_l} = \frac{\rho_g \bar{x}}{\rho_L(1 - \bar{x})}
\]

In these equations, \(\bar{x}\) is the weight flow of gas over total weight flow, \(\bar{x} = \frac{m_g}{\bar{m}}\), \(\bar{m}\) is the total mass flow rate inside the channel, \(m_g = m_{g,1} + m_{g,2}\), where \(m_{g,1}\) is the gas phase mass flow rate, \(m_{g,2}\) is the liquid phase mass flow rate, \(\rho_g\) is the density of the liquid phase, \(\rho_L\) is the density of the gas phase, \(h\) is the enthalpy of the liquid phase, \(h_g\) is the enthalpy of the gas phase, \(A_g\) and \(A_L\) are the gas and liquid cross-sectional area, respectively. All the variables that have a bar on top of them are spatially averaged (one-dimensional model) and then expressed in terms of the liquid and gas variables (homogeneous model).

In this model, the Reynolds number is defined as follows:

\[
\text{Re}_{hit} = \frac{\rho_D D_h \bar{u}}{\mu_{hit}}
\]

where \(\mu_{hit}\) is the homogeneous viscosity of the flow.

As a first step, we consider that the gas phase is only made of air and that the liquid phase is only composed of saturated water. By doing this, we neglect mass exchanges between the gas and the liquid. This means that we do not take into account water condensing and evaporating between the two phases, nor air dissolving into the liquid phase. Nevertheless we want to simulate a flow configuration in which the two phases and the

Copyright © 2004 by ASME
channel walls are at the same temperature (20°C). In this case, the heat transfer between the two phases is negligible, and since we are operating at a relatively low temperature, the convective mass transfer is also small. This might become an issue when we will start heating the channel at higher temperatures (80°C). We will include this in subsequent versions of the model through the use of species conservation equations.

We are interested in a time independent formulation of this problem, so we consider a steady flow. We assume that the wall temperature is held constant. Finally we assume that the gas phase follows the ideal gas law.

**Governing equations**

Equation (10) expresses the mass conservation of the homogeneous flow (the H subscripts of the homogeneous variables have been omitted for brevity):

\[
\frac{d}{dz}(\bar{\rho} \bar{u}) = \frac{\delta \dot{m}}{A} \tag{10}
\]

Where \(\bar{\rho}\) is the mass density, \(\bar{u}\) is the mean axial velocity of the homogeneous mixture, and \(\delta \dot{m}\) is the liquid water mass flow rate per unit length that is injected in the channel. The momentum and energy equations for the homogeneous model are:

\[
\frac{d}{dz}(\bar{\rho} \bar{u}^2) = -\frac{dp}{dz} - \frac{P_w}{A} \bar{\tau}_w \tag{11}
\]

\[
\frac{d}{dz}\left(\bar{\rho} \left(\bar{h} + \frac{\bar{u}^2}{2}\right) \bar{u}\right) = \frac{h_{\text{conv}} P_h}{A} (T_s - T_f) + h_u \delta \dot{m} \frac{A}{A} \tag{12}
\]

In these equations, \(A\) is the channel cross sectional area, \(p\) is the local pressure, \(\tau\) is the wall shear stress, \(P_w\) is the wetted perimeter, \(h\) is the specific enthalpy of the homogeneous fluid, \(h_{\text{conv}}\) is the local effective convective heat transfer coefficient, \(P_h\) is the heated perimeter, \(T_s\) is the wall temperature, and \(T_f\) is the homogenous fluid temperature.

When liquid water is added into the channel, it comes with a certain quantity of enthalpy \(h_u\), which is treated as a source term in the energy equation. We also take into account the heat transfer between the walls of the channel and the fluid by using a convective heat transfer coefficient \(h_{\text{conv}}\).

Strictly speaking, these conservation equations are not exact. In our model, we consider the following approximations. First we consider that liquid water is injected into the channel with zero momentum in the \(z\) direction. This is a good approximation since the water injection velocity is very small and its main component is in the direction perpendicular to the axis of the channel, which is not taken into account in a one-dimensional model. We also neglect the gravity effects since the Bond number is very small, and the heat conduction in the \(z\) direction.

**Constitutive relations**

As a first approximation, the gas phase follows the ideal gas law:

\[
p = \rho_v RT_f \tag{13}
\]

For the friction factor, we use the value obtained by experiments [8], which is:

\[
f_H = \frac{110}{Re_H} = \frac{8 \tau_w}{\bar{\rho} \bar{u}} \tag{14}
\]

Equation (14) applies to the average pressure drop across the channel but is applied locally in the current model. It should be pointed out that we are using an experimental correlation and there is discrepancy with the expected theoretical value (which would be \(f_H \text{Re}_H \sim 85\) for a rectangular channel of aspect ratio 10:1). The sources for the discrepancy might be the following ones:

- the actual channel dimensions are smaller than expected (micro-fabrication issue: channel depth might be less than expected). This is certainly the most significant reason of discrepancy. For example, calculations based on a depth of 45 microns (which might be the real depth of the channel, instead of the desired 50 um) produce f-Re value in the line with the theoretical value of 85.
- entry length effects may also be important
- there might be an additional pressure drop due to the fact that we are dealing with a U-shape bended channel, not a straight one.

Nevertheless, we still use the experimental f-Re relationship since we want to compare the numerical results with experimental data.

The homogeneous viscosity of the flow still needs to be defined. In general, a weighted average with respect to the gas volume fraction of the flow is the most logical means of defining \(\bar{\mu}\). A mean value defined in this manner should approach the liquid viscosity and vapor viscosity as \(\alpha \rightarrow 0\) and \(\alpha \rightarrow 1\) respectively. The relation that we propose and that satisfies these criteria is:

\[
\bar{\mu} = \mu_v \alpha + \mu_L (1 - \alpha) \tag{15}
\]

where \(\mu_v\) and \(\mu_L\) are respectively the viscosity of the gas and liquid phase.

With regard to the enthalpy-temperature relation, since the gas phase follows the ideal gas law, we have the following relation between \(h_u\) and \(T_f\), the temperature of the homogeneous fluid:

\[
h_u = h^o_u + C_p \left(T_f - T^0\right) \tag{16}
\]

where \(C_p\) is the specific heat at constant pressure of air (which is also a function of the fluid temperature \(T_f\)), \(T^0\) is a reference temperature, and \(h^o_u\) is the enthalpy of air at this temperature.

As a first approximation, knowing that we will not look at high velocity water flow regimes, we consider the liquid phase as incompressible. Under this assumption, \(h_L\) is only a function of temperature \(T_f\). Its values are tabulated in [9] and [10].
For the convective heat transfer coefficient $h_{\text{conv}}$, we propose as a first approximation a convective thermal resistance analogy which leads to the following expression [11]:

$$\frac{1}{h_{\text{conv}}^H} = \frac{\bar{\alpha}}{h_{\text{conv}}^G} + \frac{(1 - \bar{\alpha})}{h_{\text{conv}}^L}$$  \hspace{1cm} (17)

where $h_{\text{conv}}^G$ and $h_{\text{conv}}^L$ are the convective heat transfer coefficients in the cases where the channel is full of air and full of water, respectively. As a better understanding of the thermal behavior of the flow is gained from modeling and experimental results, this relation can be modified accordingly.

We compute these two coefficients $h_{\text{conv}}^G$ and $h_{\text{conv}}^L$ by means of the Nusselt number:

$$Nu = \frac{h_{\text{conv}}}{k}$$  \hspace{1cm} (18)

where $D_h = \frac{4A}{P}$ is the hydraulic diameter of the channel, and $k$ is the thermal conductivity of the considered phase. We use the value of the Nusselt number in a rectangular channel of aspect ratio 10:1 [12],

$$Nu = 5.858$$  \hspace{1cm} (19)

The velocity and temperature profiles are assumed to be fully developed (no entry-length effects), and a constant wall temperature boundary condition is applied. These relations are then improved by means of a dual approach involving physical modeling and experimental data.

**Boundary conditions**

At the inlet of the channel, the velocity, pressure, temperature, and density of the two phases are specified (Dirichlet boundary conditions). At the outlet, we apply Neumman boundary conditions [13].

**Algorithm**

In order to obtain a numerical solution that is as accurate as possible, a fully implicit finite volume method is used [14]. The algorithm assumes a constant wall temperature. An upwind scheme is applied to the mass, momentum, and energy conservation equations to obtain the pressure and enthalpy distributions. The vapor quality distribution is calculated from the continuity equation. The solution for each step is iterated until all variables converge. Accuracy is confirmed under the assumptions in the model by global verification of the conservation of mass, momentum, and energy. For each iteration step, all variables that change with the solution are checked to confirm convergence of these variables between successive iterations.

**RESULTS AND DISCUSSION**

We specify different operating conditions to the model. In the current simulations, the inlet pressure is equal to 1 atmosphere, the inlet temperature of the two phases is 20°C and the wall temperature is held constant at 20°C. We vary the inlet velocity and the water injection mass flow rate. We are interested in examining the pressure drop between the inlet and the outlet of the channel.

**Influence of the water injection quantity**

First we compute the pressure drop for 5 different values of water injection flow rate: 0, 1, 10, 100, and 300 μL/min. For each case, we compute the variation of pressure drop with respect to the inlet velocity. Water is added uniformly along the channel keeping the liquid injected constant and equal to the prescribed value. The results are shown in Fig.2.

![Figure 2 – Inlet velocity vs. pressure drop for different water injection quantities (homogeneous injection profile)](image_url)

We can see that there is no significant difference between the three first curves representing the 0, 1 and 10 μL/min cases. However, a significant increase in the pressure drop appears when the water injection flow rate reaches the value of 100 μL/min, and a dramatic increase is noticed for the 300 μL/min case. Thus, the numerical results show that for small water injection rates (1−10μL/min) the air flow velocity and pressure drop are not significantly perturbed by the presence of liquid water. But if water injection becomes consequent (100−300μL/min) larger pressure drops are observed. This can be understood by noting that the homogeneous fluid has a larger inertia when a large amount of water is added into it; the pressure work required to make it move is consequently larger. The viscosity of the flow also increases as more water is injected: the work required to make it move is consequently larger. We can also notice that the pressure drop variations in the case of 300 μL/min behave more linearly than in the other cases. The non-linear behavior of the pressure drop is linked to compressibility effects in the homogeneous fluid. This brings us to the following conclusion: the compressibility of the homogenous flow is inversely related to the quantity of water injected. This is coherent since the water is assumed to be incompressible in the present model.

In order to better understand these phenomena, we modify the numerical code so that we can isolate two physical parameters that play a key role in the increase of pressure drop along the channel due to mass addition: we want to control the compressibility and the viscosity variations of the fluid. We run
two simulations: in the first one, we keep the gas density constant. The gas is thus assumed to be incompressible. By doing this, we look at the effects that viscosity variations due to water addition may have on the flow. In a second simulation, we keep the fluid viscosity constant. By doing this, we neglect the changes in viscosity of the homogeneous fluid as the flow moves along the channel due to water addition. The results are shown in Fig. 3 (variations of the inlet velocity of the gas phase with respect to the pressure drop across the channel).

**Figure 3 - Separation of the gas density variations due to compressibility effects and viscosity variations due to mass addition**

We can see on Fig. 3.b that the curved shape of the plots is due to compressibility effects (which add non-linear effects). If the gas is assumed incompressible (Fig. 3.a), the curves are linear. We can also notice on Fig. 3.a that the curves are shifted to the right when more water is injected. Thus we can conclude that the increase in pressure drop (which is observed when the water injection flow rate increases) is mainly due to the fact that the viscosity of the homogeneous fluid increases. The wall shear stress increases accordingly, then the work required to make the fluid move is larger. If the viscosity of the fluid is kept constant along the channel as we increase the water injection flow rate, the pressure drop does not change.

**Influence of the water injection profile**

Next we hold constant the total mass of liquid water that is injected and we vary the profile that is used to inject it. We simulate four different injection profiles along the channel: homogeneous, linearly increasing, linearly decreasing and “slot”. For example, in the linearly increasing case, the amount of water that is injected increases linearly along the channel, going from a null value at the inlet to a maximum value at the outlet. For the “slot” case we simulate the presence of a slot (20 \( \mu m \) wide, half on the way down from the inlet) in the channel through which water is injected. Everywhere else, no water is introduced. We are using the slot profile to simulate experimental testing where a single injection slot of this size is used. The total amount of water injected in the channel is kept constant for all four cases.

We first simulate a water injection rate of 1 \( \mu L/min \). The results are shown in Fig.4.

**Figure 4 – Pressure drop vs. inlet velocity for different water injection profiles (water injection mass flow rate: \( 1 \mu L/min \))**

We can see that the curves for these four injection schemes are basically identical. The air flow does not seem to be perturbed by the presence of liquid water, so that the variations of the pressure drop with respect to the inlet velocity remain constant between the four cases.

If we run the simulation again, but this time using a larger value for the water mass flow rate (100 \( \mu L/min \)), we can see that there are differences. The results are shown in Fig.5.

The pressure drop in the linearly increasing case is the smallest one, followed by the homogeneous, then the linearly decreasing cases. The slot case shows the largest pressure drops. It can be concluded from these plots that the air flow is perturbed not only by the quantity of water that is injected, but also by the profile that is used to inject it. If water is injected through a single slot, the pressure drop will be larger than if the water is continually added along the channel. These observations are physically meaningful: the pressure drop is proportional to the amount of water and length over which it must be convected.
Inlet velocity vs. pressure drop for different water injection profiles (water injection mass flow rate = 100uL/min)

We can understand the physics that is involved in this mechanism as follows: let consider that we measure the pressure drop between the inlet and a point 2/3 on the way down from the inlet. The water injection profile that corresponds to the largest quantity of liquid water that has been injected until this point is the slot one, then the linearly decreasing profile, then the homogeneous one, and finally the linearly increasing profile. The flow that has the largest quantity of liquid water injected into it at this point is also the one that has the largest inertia and largest viscosity. It follows that the pressure work required to make the flow move is the largest one, which is observed experimentally. Thus pressure drop distribution data from these simulations are consistent with physical predictions.

Influence of the inlet pressure

The influence of the inlet pressure on the flow behavior was also investigated. We computed the pressure drop for 2 different values of inlet pressure: 1 and 10 atmospheres. For each case, we ran the simulation for 5 different water injection mass flow rates: 0, 1, 10, 100, and 300 µL/min while keeping the water injection profile the same (homogeneous). The results are shown in Fig.6.

It is apparent that the compressibility effects of the flow diminish as the inlet pressure increases: the pressure drop-velocity relationship behaves more linearly as the inlet pressure is increased from 1atm to 10atm.

We can also notice that the curves for different water injection rates get closer to each other when inlet pressure is increased. Thus the pressure drop difference between two water injection quantities (for the same inlet pressure and velocity) decreases as the inlet pressure increases. In other words, there is an inverse relationship between the inlet pressure and the effects of liquid water injection on air flow velocity and pressure drop.

Comparison with experimental results

The model predictions are compared with experimental results obtained by testing a set of microchannels with a varying number of water injection slots on the side walls (see [8] for more details). A series of microchannels with water injection slots in the side walls are used for this purpose. These structures were microfabricated by plasma etching microtrenches in silicon and covering them with Pyrex glass as per previous work by Zhang et al. [1]. Fig. 7 shows the general layout of the test structures and three specific examples of test structures with different water injection geometries. Dry air is flown through the 500 µm wide and 50 µm deep U-shaped channel while water is introduced through the channel perpendicular to it.
Pressure drop distribution data from these experiments are compared with model predictions in Fig. 8 (inlet pressure = 2 atm in case (a) and inlet pressure = 3 atm in case (b)).

The numerical model is capturing the right physics: the orders of magnitude are correct, and we can see the same behavior when we inject more or less water: if the water injection rate is less than 10 \( \mu \text{L/min} \), there is no significant difference in pressure drop. But if water injection becomes consequent (10 – 100 \( \mu \text{L/min} \)), larger pressure drops are observed. The homogeneous model seems to overestimate the pressure drop. This is attributed to the fact that the experimentally observed flow regime is of stratified kind (liquid film and gas core flowing over it). This flow regime leads to a smaller pressure drop than the one that is obtained by means of the homogeneous flow model. The main surface of the channel is wetted by the gas phase which has a smaller viscosity than the liquid phase. The resulting wall shear stress is also smaller than the one that is actually computed.

We can also mention that the homogeneous assumption implies uniform mixing of the two phases: therefore the injected water is fully accelerated towards the air velocity as soon as it is injected. This leads to larger momentum and frictional pressure drop contributions from the liquid water than there really are. In reality, as mentioned above, we observe a stratified flow regime in which air and water are fully separated. In this regime, the two phases flow at different rates with the water moving slower than the air. The increase in pressure drop is mainly due to the reduced “effective” cross sectional area that is available for the air flow when water is introduced.

In order to improve the accuracy of this model, we need to take into account the fact that there are two separated fluids flowing along the channel: our task for the future is thus to increase the complexity of this model and to tackle the problem of separated flow models which permit different gas and liquid velocities and flow direction, as well as different gas and liquid temperatures.

**CONCLUSION**

We have developed a theoretical model and a numerical simulation of microchannel two-phase flow with distributed liquid water injection from walls by using a one-dimensional homogeneous flow model. The modeling and solution of the conservation equations provide pressure drop between the inlet and the outlet of the channel as a function of inlet velocity. We have investigated the influence of different parameters involving the water injection, such as the quantity of water that is injected and the water injection profile that is used to inject it. The numerical results show that for a small water injection quantity, the air flow velocity and pressure drop are not significantly perturbed by the presence of liquid water. But if the water injection becomes consequent, different behaviors come to light: a larger pressure drop is computed when the mass flow rate of injected water is set to 100 \( \mu \text{L/min} \) or when we use a linearly increasing injection profile.

The model predictions are compared with experimental results obtained from testing a set of microchannels with a varying number of water injection slots on the side walls. Pressure drop distribution data from these experiments are consistent with model predictions, even though the computed pressure drop rise between 5 \( \mu \text{L/min} \) and 100 \( \mu \text{L/min} \) is larger than the one observed experimentally. Our task for the future is to improve the complexity of this model in order to tackle the problem of separated flow models which permit different velocities and temperatures for the two phases.

**ACKNOWLEDGMENTS**

The authors would like to thank Honda R&D Co., Ltd. not only for their support but also for their input, feedback and insight into the project.

**REFERENCES**