Numerical simulation of flow with evaporation in triangular grooves

N Sibiryakov¹² and O Kabov¹
¹ Kutateladze Institute of Thermophysics, Russian Academy of Sciences, Prosp. Lavrentyeva 1, Novosibirsk, 630090, Russia
² Novosibirsk State Technical University, Prosp. K. Marksa 20, Novosibirsk, 630073, Russia

E-mail: kolyasibir@yandex.ru

Abstract. We study the flow of an evaporating liquid film in triangular open microchannel. Above the surface of the liquid flows a gas. The shear-stress on the gas-liquid surface drags the liquid and removes the vapor. It models either flow in a channel or a flow in a single groove. The novelty of this work is that we simulate gas-driven flow with evaporation. We use Boundary element method to solve Laplace equation with mixed boundary conditions. This method gives the normal derivative of vapor concentration at the surface of the liquid. The resulting evaporation flux is growing sharply near the contact line. Integral evaporation coefficient slightly depends on the depth of the liquid. The upper gas flow drives the liquid film, resisting a velocity drop and preventing the dry spot formation.

1. Introduction
The performance of modern computers highly depends on the processor frequency. Modern chips produce several billion elementary operations per second. Such a load on the processor causes serious heating of the electronics. Air fans can hardly cool personal computers, consume a lot of energy and cannot cope with the growing performance of processors. In addition, the control system on the space station or on the satellite is not profitable to cool the processor with a large and energy-intensive cooler. To divert large heat flows from a small area, it is necessary to create fundamentally new cooling systems.

Thin evaporating liquid films [1] can divert large heat fluxes from small arias. In the future, microchannels with a two-phase flow will become the basis for new heat removal systems. Even without evaporation [2], the flow in the triangular channel is more efficient than in rectangular due to the larger surface area of the channel. The heat flux reaches 50 W per cm square. Moreover, recent articles show that the flow in open triangular channels is very stable to external disturbances [3]. Cooling efficiency increases significantly if the fluid in the channel evaporates.

For the first time, the problem of a two-phase flow in a closed channel was solved by M. Soliman, J. R. Schuster, P. J. Berenson [4]. They considered a vertical channel of circular section, in which vapor condensation occurs on the walls. The fluid moved by gravity and the flow of steam, which flowed into the channel against the direction of liquid film. For channels of a triangular cross section, a similar problem was solved by V. Sartre et al. [5]. In their formulation, steam enters the channel and also condenses on the walls, but, unlike the cylindrical channel, vapor condenses in the corners and liquid films start there. A similar problem, but with a vertically arranged channel on which the vapor
condenses, was solved by T.S. Zhao, Q. Liao [6]. They examined in more detail how the process of condensation occurs on the walls of the tube at different thicknesses of the liquid film. The problem with evaporation in a triangular channel was solved by Y. Peles and S. Haber [7]. In their formulation, the liquid evaporates in a closed triangular channel, as a result, the associated vapor flow moves at a high speed above the surface. The theoretical works of V.S. Ajaev and G.M. Homsy [8, 9] are devoted for the flow of liquid and saturated vapor in channels of rectangular cross section. They considered in detail how the liquid evaporates in the region of the contact line and found which heat flux can divert a channel of similar geometry.

There are not many experimental works on the flow of liquid with evaporation in the channels of a triangular cross section. T.S. Zhao, Q.C. Bi [10] experimentally studied vapor condensation on a pipe of a triangular cross section. G.P Peterson, H.B. Ma [11] studied the maximum heat flux that can divert the flow in an open triangular channel. The most detailed experimental study of the flow with evaporation was carried out by K. Helbig et al. [12]. They studied theoretically, numerically and experimentally the flow under the action of gravity in a channel with ribbed walls. In their work, they found that before forming a dry spot, the fluid in the grooves begins to behave unstable and is broken into drops. They did not give any explanations for this phenomenon in their article, because they focused on the calculation of the flow until it breaks into a flow in separate grooves. In this paper, since the flow is considered in a separate groove, we can predict the moment at which the flow of fluid in the corners or grooves becomes unstable.

2. Theoretical model
In my formulation of the problem, a constant temperature is set on the channel walls, the channel is open on top and evaporation occurs freely from the surface of the liquid (figure 1). This simulates both the flow in a triangular open channel and in each individual groove at the bottom of a structured channel. Let z-coordinate be directed along the channel, x and y in a cross-section.

![Figure 1. Channel layout. Depth of the liquid decreases due to evaporation.](image)

Liquid flows into the channel with an initial velocity \( v_0 \) and with an initial thickness \( h_0 \). Evaporation from the surface of the liquid is proportional to the temperature of the liquid and the surface element:

\[
 dm = -\gamma(T, h) dS,
\]
where \(dm\) is evaporated mass of the liquid, \(C(T, h)\) is the evaporation rate of the liquid, which depends on temperature \((T)\) and on the depth \((h)\) of the liquid, \(dS\) is the surface element from which evaporation occurs.

Instead of finding the velocity at each point of the fluid, we search for the velocity \(v\) averaged over the section. Since the thickness \(h\) of the fluid is small (less than a millimeter), the surface of the fluid in the cross section is an arc of a circle of radius \(R\). Because of the curvature of the boundary and surface tension \(\sigma\), the pressure in the fluid \(P\) is equal to the Laplace pressure:

\[
P = P_0 - \frac{\sigma}{R}
\]

From geometric considerations:

\[
R = h \frac{\sin \varphi}{\cos (\varphi + \theta)}.
\]

The law of conservation of momentum in the integral form:

\[
\int_V \rho g_z dV + \int_S \left( P - \frac{P}{2} + \frac{\rho v^2}{2} + \frac{\rho(\varphi - v)^2}{2} \right) dS = \int_S \rho v(\mathbf{v} \cdot \mathbf{n}) ds.
\]

Where \(\rho\) is density of the liquid, \(g_z\) is projection of free fall acceleration projection onto the \(z\) axis, \(P\) is liquid pressure, \(\zeta\) is the coefficient of wall-liquid friction force, \(\zeta_a\) is the coefficient of gas-liquid friction force, \(v_a\) is the velocity of gas above the liquid, \(\mathbf{n}\) is normal vector. Averaging over the cross-section gives us following equation:

\[
\frac{\rho}{S} (v^2 S)' = -P' + \rho g_z - \frac{\rho v^2 L}{2} + \frac{\rho(\varphi - v)^2}{2} 2\alpha R
\]

where \(S\) is derived from geometric considerations:

\[
S = \frac{h^2}{2} \left[ \sin \varphi - \frac{\sin^2 \varphi}{\cos^2 (\varphi + \theta)} \right] (\alpha - \sin \alpha).
\]

For a small sized system, the flow is laminar, so the shear stress is well described by poiseuille law:

\[
\zeta = \frac{13.3}{\rho \nu D_t} \quad \eta = \frac{13.3 \eta}{\rho \nu D_t}
\]

where \(D_t\) is characteristic length of the system.

From the mass conservation law follows:

\[
(vS)' = -\frac{C(T, h)}{\rho} \frac{\sin \varphi}{\cos (\varphi + \theta)} v.
\]

Now we split variables and get system of two ordinary differential equations for velocity and depth distribution in the channel:

\[
v' = g_z + \frac{\sigma A}{2 \rho \nu h^2} - 2Ak \frac{v}{h} - B \frac{v}{h^2} + 2Bk \alpha \frac{v_a - v}{h^2},
\]

\[
h' = -\frac{A}{\nu \cos (\theta + \varphi)} - \frac{v h}{2v'}
\]

where:

\[
A = \frac{C(T, h)}{\rho} \left( \frac{\pi}{2} - \theta - \varphi \right)
\]

\[
B = \frac{\left( \sin \varphi - \frac{\sin^2 \varphi}{\cos^2 (\varphi + \theta)} \right) [\alpha - \cos (\theta + \varphi)]}{2.66 v}
\]

Note that from the first equation for velocity follows that \(v' \to \infty\) when

\[
\rho v^2 = \frac{\sigma}{2R}
\]

After the velocity becomes critical, the theoretical model does not work. This criterion is a condition for the onset of the Rayleigh – Plateau instability. Apparently, soon after the model stops...
working, the liquid breaks up into drops and there is no longer a continuous flow, as it was already
observed in the work of K. Helbig et al. [12].

3. Calculation of evaporation rate

Before solving a system of two ordinary differential equations, it is necessary to find the
evaporation coefficient of the liquid. To do this, it is necessary to solve the diffusion equation, since
the evaporation rate is determined by the vapor gradient on the surface of the liquid.

\[ \frac{dm}{dt} = D(n, \nabla c_v) \rho_v dS = D \frac{\partial c_v}{\partial n} \rho_v dS \]

From here we obtain the connection between the desired function and the solution of the equation
of vapor diffusion in the channel:

\[ C(T, h) = D \frac{\rho_v}{\rho_l} \frac{\partial c_v}{\partial n}. \]

Under stationary conditions and with negligible convective terms the diffusion equation is
simplified to Laplace equation:

\[ D \Delta c = 0. \]

On the surface of the liquid, the vapor concentration must be equal to the saturation concentration
\( c = c_s(T) \). At the upper boundary almost all of the steam is removed and the concentration \( c = 0 \).
Moreover, the solution is linear for these boundary conditions. Since we want to model the flow in one
of the trenches of the finned channel, the symmetry condition must be satisfied on the side walls (it is
also the periodic condition)

\( \frac{\partial c}{\partial n} = 0. \)

We know the value of the function at some part of the boundary, and we know the value of the
derivative on the other part. The best method to find the value of the normal derivative with the
greatest accuracy is boundary integral equations method, or rather its modification, which is called the
boundary elements method [13]. For the correct work of this method at the boundaries with a non-
continuous normal (in the corners), it is necessary either to round them or to use smooth, finite kernels
[14].

4. Results and discussion

We obtained the solution for normal derivative of vapor concentration at the liquid surface. The
integral method automatically gives a large normal derivative in those places where the thickness of
the liquid film is small. In real experiments, it is the microregion that accounts for the maximum
evaporation rate, which corresponds to the obtained numerical solution. To obtain the evaporation rate
of the fluid in the channel, it is necessary to integrate the normal derivative of the concentration over
the surface of the fluid in this section. Having carried out calculations for different values of the depth
of the fluid in the channel, we obtain the dependence of the integral evaporation coefficient on the
depth of the fluid (figure 2).

This dependence is linear a large depth range. Because the main evaporation occurs in a
microregion, evaporation rate integrated over the cross section weakly depends on liquid's depth in the
channel. This result is in agree with experimental observations for droplets evaporation [15, 16].
Figure 2. The dependence of the integral evaporation rate on the depth of the liquid in the channel.

Now, let us consider how the rate of flow of the associated gas flow affects the distribution of the thickness of the liquid in the channel. To do this, we take the flow rate of the same order with the initial velocity of the liquid (figure 3).

Figure 3. The liquid film thickness distribution in the channel. The initial velocity of the fluid is 2 m/s. For the dotted line the gas velocity is 4 m/s, for the solid line 2 m/s.

The velocity of the fluid grows to a limited extent, since the force of viscous friction against the walls gradually becomes greater than the capillary one. After reaching the maximum, the speed drops to a critical value, after which the simulation does not work (figure 4). Gas flow prevents a rapid drop in the velocity of the liquid film. This increases the effective cooling length.
Figure 4. The velocity of the fluid in the channel. The initial depth of the fluid is 1 mm. For the dotted line the gas velocity is 4 m/s, for the solid line 2 m/s.

Let us estimate the effective heat flux along the canal:

\[ q = \frac{\rho \, d \, (\nu \, S)}{2h} \, \frac{dz}{dx} \]

It practically does not change, slightly decreasing and reaches 29 kW per m square. This means that reducing the size of the cooling device and increasing the initial velocity of the fluid and gas in the channel increases the efficiency.

5. Conclusion

In this work we model the flow of a liquid with evaporation in micro-grooves for the first time, taking into account the co-current gas and uneven evaporation from the surface. This is a big improvement of the previous work [17].

The obtained integral evaporation rate (Fig. 2) weakly depends on the depth of the fluid in the channel. This is due to the fact that evaporation is intensified in the microregion [15,16]. This effect was obtained numerically, since the method of boundary integral equations has an advantage over other methods for solving partial differential equations in problems with a rapidly changing normal vector (the presence of angles).

Numerical simulation shows that the viscosity of the fluid plays a significant role in this problem limiting the maximum velocity of the fluid and preventing the film from rapid thinning. Upper gas flow increases the velocity of the fluid in the channel, thereby increasing the effective cooling length.

Simulation works only while velocity is greater than critical (3). As soon as the speed drops to a critical, in a real liquid, the Rayleigh–Plateau instability develops and the film breaks into drops [12].

Increasing the initial velocity and reducing the size of the channel also improves the effective heat flux. This means that it is possible to make smaller channels if the co-current gas flow will blow with sufficient speed.

Acknowledgment

This work was supported by the Russian Scientific Foundation (project No. 18-79-10258).

References

[1] Gatapova E Ya and Kabov O A 2008 Int. J. Heat and Mass Transfer 51 4797–10
[2] Tiselj I, Hetsroni G, Mavko B, Mosyak A, Pogrebnyak E and Segal Z 2004 Int. J. Heat and Mass Transfer 47 2551–565
[3] White N C and Troian S M 2018 Phys. Rev. Fluids, arXiv:1812.09816
[4] Soliman M, Schuster J R and Berenson P J 1968 ASME J. Heat Transfer 90 267–76
[5] Sartre V, Zaghdoudi M and Lallemand M 2000 *Int. J. Therm. Sci.* **39** 498–504
[6] Zhao T S and Liao Q 2002 *Int. J. Heat and Mass Transfer* **45** 2829–42
[7] Peles Y P and Haber S 2000 *Int. J. Multiphase Flow* **26** 1095–115
[8] Ajaev V S and Homsy G M 2001 *J. of Coll. and Interf. Sci.* **240** 259–71
[9] Ajaev V S and Homsy G M 2001 *J. of Coll. and Interf. Sci.* **244** 180–89
[10] Zhao T S and Bi Q C 2001 *Int. J. Heat and Mass Transfer* **44** 2523–34
[11] Peterson G P and Ma H B 1996 *ASME J. Heat Transfer* **118** 731–9
[12] Helbig K, Alexeev A, Gambaryan-Roisman T and Stephan P 2005 *Flow, Turbulence and Combustion* **75** 85–104
[13] Crouch S L and Starfield A M 1983 *Boundary Element Methods in Solid Mechanics* (George Allen & Unwill) p 322
[14] Sibiriakov E B 2017 *Physical Mesomechanics* **20** 98–104
[15] Gatapova E Ya, Semenov A A, Zaitsev D V and Kabov O A 2014 *Colloids and Surfaces A: Physicochem. Eng. Aspects* **441** 776–85
[16] Kabov O A, Gatapova E Ya, Semenov A A, Jutley M S, Ajaev V V, Kirichenko E O, Feoktistov D V, Kuznetsov G V and Zaitsev D V 2018 *Interfacial Phenomena and Heat Transfer* **6** 421–35
[17] Sibiryakov N, Kabov O and Belosludstev V 2019 *EPJ Web of Conferences* **196** 00051