Numerical model of a two-phase microchannel heat sink electronics cooling system

Tom Saenen^{a,*}, Martine Baelmans^a

^a*Katholieke Universiteit Leuven, Department of Mechanical Engineering, Celestijnenlaan 300A B-3001 Heverlee, Belgium*

Abstract

Microchannel heat sinks using two-phase flow boiling have excellent potential for cooling high heat flux electronic devices. A numerical model of a two-phase cooling system using microchannel heat sinks is presented. This is supplemented with transient heat conduction models of the heat sink and heat exchanger and with various state of the art empirical correlations to close the equations. Finite volume discretization and the SIMPLE algorithm are used to solve the mixture conservation equations of mass, momentum and energy. The numerical code is verified using the method of manufactured solutions. This method reveals that the numerical order in space and time is consistent with the expected values from theory, second order and first order, respectively. Further some illustrative results of the model are validated using experimental measurements. These results agree well with each other and indicate good predictive capability of the system model.

Keywords: Two-phase cooling, System model, Code verification

1. Introduction

In recent years, research into small scale, high performance cooling solutions for electronic applications has increased tremendously. This is in part due to the continuous increase in speed, complexity and miniaturization of the electronic components. This leads to an increase in heat generation of the electronics that needs to be efficiently dissipated. Furthermore, transitioning from conventional finned heat sinks using air to more advanced, compact cooling solutions, results in overall energy saving of large scale electronic systems. These advanced cooling solutions also exhibit a smaller overall form factor than conventional heat sinks which in turn improves portability.

One of the possible high performance heat sink designs, incorporates liquid flow, usually water, in microchannels situated on or near the silicon die of the electronics. Another high performance cooling solution is the use of flow boiling in microchannels. This technique promises an even higher cooling potential, because the heat transfer coefficient associated with flow boiling is typically an order of magnitude larger than single-phase liquid heat transfer coefficients.

*Corresponding author. Tel.: +32-16-322549; fax: +32-16-322985
Email address: tom.saenen@mech.kuleuven.be (Tom Saenen)

Furthermore, capacitive heating of the coolant is almost non-existent due to the latent heat of the boiling process.

In spite of all these advantages, some hurdles still exist to effectively use this cooling technique for electronics. As such conventional correlations do not hold in the microchannels. However, because of recent research many new correlations have been developed specifically for two-phase flow in microchannels [1, 2, 3].

A second issue is that the system level modeling of two-phase flow in microchannels has not received much attention. In the past many detailed system level models have been developed for nuclear reactor cooling and refrigeration systems, a.o. [4]. Because the system level design and control strongly influence the performance of the two-phase microchannel heat sink [5], it is paramount that a detailed, transient system model is available for the designers of such a cooling system.

In this paper a detailed, transient system model of a two-phase microchannel electronics cooling system is elaborated. First, the general formulation of the system equations for the conservation of mass, momentum and energy are explained for all typical components of a two-phase cooling system. The heat conduction models for the heat sink and heat exchanger are also presented. Moreover, the correlations to solve the various equations are described. Subsequently, the numerical solution of these equations is elaborated. This solution is based on a finite volume discretization and implicit time integration. Furthermore, the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) is used to solve the hydraulic equations. The numerical code is verified using the method of manufactured solutions. Finally, some illustrative results of the model are validated using experimental data.

2. The two-phase system model

In this section, the mathematical formulation of the system model is presented. First, a general overview of the system model itself and the two-phase mixture flow equations is given. Second, the discretized forms of these equations are presented. Third, heat conduction in the solid elements is elaborated. Finally, the empirical correlations and models used to close the equations are given.

The main assumptions for the flow modeling are: transient one-dimensional flow, the two-phase modeling is based on a diffusion, drift-flux, model, gravity effects are neglected, thermal equilibrium between liquid and vapor and local thermodynamic equilibrium is assumed, the various components in the system are thermally well insulated from the environment. Therefore, convective heat transfer is only modeled at the heat sink and heat exchanger. Heat conduction in the heat sink is modeled enabling a transient three-dimensional temperature distribution. A two-dimensional heat load map is used as input for the heat sink. The heat exchanger is modeled using a transient one-dimensional axial temperature variation. The heat flux distribution in the finned heat exchanger is modeled in two dimensions, axial and radial.

2.1. Model equations

A basic configuration for a two-phase cooling system is presented in Fig. 1. Such a system typically consists of a pump, microchannel heat sink, a heat exchanger with air, an accumulator and connecting tubes. All these components are incorporated in the numerical model. Furthermore, the model needs to be capable for use in dynamic simulations both with single-phase flow, vapor-liquid flow and the transition between these regimes.

Figure 1: Schematic representation of a two-phase cooling system, with (a) the pump, (b) the accumulator, (c) the microchannel heat sink, (d) the heat exchanger with air and (e) the connecting tubing.

The governing equations for flow-area averaged two-phase flow can be formulated in different forms with a varying degree of accuracy. Going from less accurate to more detailed these can be divided into (i) homogeneous models, (ii) diffusion, drift-flux, models and (iii) multifluid models [6]. In the present system model the diffusion model is used. This model is selected because of the added accuracy with respect to the homogeneous model while not being computationally costly as the multifluid models. These latter also require numerous interfacial transfer models and correlations [6]. The liquid and vapor phases are assumed to be in thermal equilibrium, and viscous heating is neglected. In total the model consists of one mass equation, one mixture momentum equation and one mixture enthalpy equation, neglecting potential energy changes in time. These are given in Eqs. (1) to (3), from [6].

$$\frac{\partial \rho}{\partial t} + \frac{\partial G}{\partial s} = 0 \quad (1)$$

$$\frac{\partial G}{\partial t} + \frac{1}{A} \frac{\partial}{\partial s} \left(\frac{AG^2}{\rho'} \right) + \frac{\partial P}{\partial s} + F + \rho g \sin \theta = 0 \quad (2)$$

$$\frac{\partial \rho h^*}{\partial t} + \frac{1}{A} \frac{\partial AGh}{\partial s} - \frac{\partial P}{\partial t} - \frac{\partial}{\partial s} k \frac{\partial T_c}{\partial s} = q \frac{P_m}{A} \quad (3)$$

In Eqs. (1) to (3), s is the coordinate in the direction of the flow, $\rho = \alpha \rho_g + (1 - \alpha) \rho_l$ the mixture density, A the cross sectional area of the flow, $G = \alpha \rho_g u_g + (1 - \alpha) \rho_l u_l$ the mixture mass flux with u_g the vapor velocity and u_l the liquid velocity. ρ' is the apparent momentum density

defined by Eq. (4), from [6].

$$\rho' = \left(\frac{(1-x)^2}{\rho_l(1-\alpha)} + \frac{x^2}{\rho_g\alpha} \right)^{-1} \quad (4)$$

In Eq. (4), x is the vapor quality, i.e. mass fraction of vapor, α the void fraction, i.e. volume fraction of vapor, and ρ_l, ρ_g the liquid and vapor density respectively at mixture temperature and pressure. Furthermore, P is the pressure, F the frictional and local losses, g is the gravitational acceleration, θ is the inclination angle of the flow with respect to the horizontal plane. h^* is the mixture specific enthalpy defined by Eq. (5), from [6].

$$h^* = \frac{\rho_l(1-\alpha)h_l + \rho_g\alpha h_g}{\rho} \quad (5)$$

In Eq. (5), h_l, h_g are the saturated liquid and vapor enthalpy. Additionally, $h = xh_g + (1-x)h_l$ is the mixed-cup enthalpy, k the thermal conductivity, T_c the mixture temperature, q the heat flux and P_m the perimeter of the flow channel.

In order to close the equations the friction and local losses need to be calculated from empirical correlations. Furthermore, the void fraction needs to be calculated from a suitable expression. The choice of these correlations will be detailed further.

2.2. Empirical correlations

In order to solve the discretized equations empirical correlations are needed to calculate the void fraction α , the frictional pressure losses f , local pressure losses K and the heat transfer coefficient h_f . To calculate the void fraction from the vapor quality, the widely used and well validated CISE correlation is used [7] in the macro-scale channels. There are not many void fraction correlations available specifically developed for microchannel flow. However, the correlation of Zhang et al. [8] performs the best in their comparison with other void fraction correlations, using experimental data from 6 different data sets. Apart from these void fraction correlations, the homogeneous void fraction model and the popular Zivi [9] correlation is also available in the system model.

To calculate the single-phase pressure losses, fully developed flow is assumed in the larger components of the loop. For laminar flow in circular channels, the following formula is used : $f = \frac{64}{Re}$, with Re the Reynolds number defined as: $Re = \frac{GD_h}{\mu}$, with μ the dynamic viscosity. For turbulent flow, the Colebrook correlation is used [10]. The two-phase pressure losses in the components with macro-scale channels are calculated using the conventional and widely used Lockhart-Martinelli correlation [11].

Because the microchannels can be relatively short, flow development can be an important part of the pressure losses. Therefore, the correlation for developing, laminar flow of Yilmaz-Bender [12] is used to calculate the single-phase pressure losses in the microchannels. This correlation is a generalization of the well known Shah-Bender correlation [13]. The two-phase pressure losses in the microchannels are calculated using the pressure drop correlation of Mishima and Hibiki [14]. In the study of Zhang et al. [8], this correlation performed well in a comparison with other pressure drop correlations using 13 different experimental data sets. Ali et al. [15] also found that the correlation of Mishima and Hibiki performed best, in their comparison with other pressure drop correlations using their own experimental data. Finally, Lee and Garimella [16] also noted that the correlation of Mishima and Hibiki agreed reasonably well with their experimental

data. Apart from this correlation, the homogeneous flow model using various two-phase viscous mixture models is also available, as this shows a good performance in many studies, such as Lee and Mudawar [17] and Zhang et al. [8]

The main local pressure losses in the system consist of contraction and expansion losses at the in- and outlet of the microchannels. As has been shown in literature [18] these losses can be a significant portion of the total pressure losses in microchannels. For single-phase flow, the formulas derived by Kays [19] for the contraction and expansion losses are used.

For two-phase flow, the irreversible contraction and expansion losses cannot be so easily identified. This is mainly because the reversible pressure losses already require the use of a void fraction correlation. However, the following expressions for the irreversible pressure losses [6] based on a similar derivation as their single-phase counterparts have shown reasonable agreement with experimental data for microchannels [20].

$$K_e = \frac{G^2}{2\rho_l} (1 - \sigma^2) \left[1 + x \left(\frac{\rho_l}{\rho_g} - 1 \right) \right] \quad (6)$$

$$K_c = \frac{G^2}{C_c^2} (1 - C_c) \left[-\frac{C_c}{\rho'} + \frac{1 + C_c}{2} \frac{\rho_h}{\rho''^2} \right] \quad (7)$$

In Eqs. (6) and (7), K_e and K_c are the expansion and contraction loss coefficients, referenced to the mass flux in the smallest channel, σ is the flow area contraction ratio, always the ratio of the smallest area to the largest area, and C_c is the contraction coefficient. C_c is calculated using Eq. (8), from Geiger [21].

$$C_c = 1 - \frac{1 - \sigma}{2.08(1 - \sigma) + 0.5371} \quad (8)$$

Finally, ρ'' is the apparent energy density. This can be calculated using Eq. (9), from [6].

$$\rho''^2 = \left(\frac{(1 - x)^3}{\rho_l^2 (1 - \alpha)^2} + \frac{x^3}{\rho_g^2 \alpha^2} \right)^{-1} \quad (9)$$

The heat sink is divided into three regions to calculate the heat transfer coefficient. These consist of the single-phase region, the subcooled boiling region and the saturated two-phase region. The last region is identified by a local thermodynamic vapor quality larger than zero. To separate the subcooled boiling region from the single-phase region, the onset of boiling (ONB) model of Liu et al. [22] is used. This analytical and easy to use model has been validated using a microchannel test section and shows good agreement with the experimental results [22].

The laminar heat transfer in the single-phase region is calculated with the Muzychka and Yovanovich correlation [23]. This is a conventional correlation that takes into account flow development and channel geometry. For the turbulent single-phase regime, the Gnielinski correlation is used [24]. For the two-phase saturated boiling heat transfer coefficient, the correlation of Bertsch et al. [3] is used. This correlation was constructed using a wide range of data points from 14 different experimental studies. Additionally, the correlation performed well in the independent study of Costa-Patry et al. [25]. The three-zone model of Thome et al. [2] is also implemented, because of its good performance in many studies, e.g. Costa-Patry et al. [25]. However in the current study all the results are obtained using the correlation of Bertsch et al. To account for subcooled and partial boiling, a spline interpolation is performed between the convective single-phase heat transfer coefficient and the two-phase saturated boiling heat transfer

for a saturated liquid. This allows for a smooth transition from the single-phase regime to the saturated boiling regime.

In the heat exchanger the primary flow is divided into two regions to calculate the heat transfer coefficient. These consist of a condensation region and a single-phase region. The separation between these regions is set by a local thermodynamic vapor quality equal to zero.

In the single-phase region, the same correlations as in the single-phase region of the heat sink are used. The heat transfer in the condensation region is calculated using the simple and widely used correlation of Shah [26].

For the air side heat transfer of the louvered fin heat exchanger, the correlation of Kim and Bullard [27] is used, as this correlation was specifically developed for louvered fin heat exchangers.

Figure 2: Schematic representation of the heat sink discretization, with • the central nodes and ◦ the staggered nodes.

3. Numerical discretization

A first step to numerically solve Eqs.(1) to (3) consists of discretizing the equations based on the finite volume technique. This is done using a staggered 1-dimensional grid to avoid odd-even pressure-velocity decoupling. Mass and enthalpy are discretized over control volumes (CVs) around the central nodes, whereas the momentum equation is discretized over a staggered grid. The central nodes are positioned so that the discretized pressures coincide with the entrance and exit of the components, see Fig. 2 for the discretization of the heat sink. This way the individual pressure losses of each component are accurately represented by the calculated pressures. This also means that the entrance and exit of the components coincide with the faces of the momentum control volumes. Special consideration is given to the connection of the accumulator to the flow loop. The connection is made with a perpendicular splitter, or T-connection, with the accumulator

tubing perpendicular to the flow loop tubing. A momentum CV completely encompasses the splitter and is thus the only momentum CV with 2 velocities in the entire flow loop. Because both velocities are perpendicular there exists no momentum transfer towards the accumulator. As a result the accumulator interacts with the loop through its pressure only. The discretization of each equation is further elaborated in the next subsections.

3.1. Discretized equations

As previously mentioned, the momentum equation is discretized over the staggerd grid. A semi-discretized form of the resulting equation is shown in Eq. (10).

$$V \frac{\partial \bar{G}}{\partial t} + G_e A_e \frac{G_e}{\rho'_e} - G_w A_w \frac{G_w}{\rho'_w} + (p_e - p_w)A + \left(\frac{fL}{D_h} + K_{tot}\right) \frac{\bar{G}^2}{2\rho} A + V \bar{\rho} g \sin \theta = 0 \quad (10)$$

In Eq. (10), V is the volume of the CV, $\bar{G}$ is the mean value of the mass flux in the CV, the subscripts e and w denote the east and west face of the CV respectively, $\dot{m}$ is the mass flow, A is the cross sectional area of the CV, f is the Fanning friction factor, K_{tot} is the total local pressure loss factor, L is the length of the CV, D_h is the hydraulic diameter of the CV and $\bar{\rho}$ is the mean mixture density in the CV.

Because of the small microchannels and the high flow speed associated with them, an explicit scheme for the time integration would impose very small time steps in order to be numerically stable. Therefore, an implicit Euler scheme is used to integrate the equations. This ensures stability at the cost of a higher computational load per time step. The convective terms in Eq. (10) can be evaluated using various discretization schemes. Because several sudden changes in flow properties can arise in the system, e.g. transition between different components, stable, total variation diminishing (TVD) schemes are used in the model. The schemes that are incorporated include: Min-mod, MUSCLE, SMART, UMIST and SUPERBEE. However, because the grid is non-uniform, with different CV sizes in different components, some modifications of the schemes are necessary to ensure their boundedness. This is done based on the symmetric formulation of Waterson [28]. The friction factors and loss factors are evaluated based on correlations specified in section 2.2. The local pressure losses are always incorporated in the CV downstream of the area changes. The face mass flows are calculated from the nodal values using a central difference scheme (CDS). Furthermore, the density, ρ_p , is also calculated using CDS from the nodal values.

The pressure correction equation is used in the SIMPLE method [29] to incorporate the effect of the continuity equation on the mass fluxes and update the pressures. Since it is based on the continuity equation it is evaluated on the main grid. Based on the continuity equation the pressure correction equation is constructed. A detailed description can be found in [29].

Similarly as the momentum equation, the enthalpy equation is solved on the main grid. The heat flux q is calculated using correlations, as discussed in section 2.2. Also $\bar{h}^*$ can be converted to h by multiplying it by Eq. (11).

$$I = \frac{\alpha(h_g \rho_g - h_l \rho_l) + h_l \rho_l}{(\alpha(\rho_g - \rho_l) + \rho_l)((h_g - h_l)x + h_l)} \quad (11)$$

The convective terms h_e and h_w are calculated using a TVD scheme. The central total volumetric flux is calculated using CDS from the nodal mass fluxes.

3.2. Heat conduction in the solid elements

Heat conduction is modeled in the heat sink and for the heat exchanger. In the heat sink this is necessary to calculate the electronic junction temperatures accurately. For the heat exchanger this correctly couples the coolant temperature with the temperature of the air.

The configuration for the heat sink under consideration, contains parallel microchannels etched in the backside of the same silicon piece that houses the active electronic layer, see Fig. 3. Additionally two collectors or manifolds are also included upstream and downstream of the microchannels. The collectors are modeled as a single flow channel with the same height as the microchannels and being as wide as the complete heat sink. The heat conduction is modeled by discretizing using the finite volume method. The heat sink is fully discretized in three dimensions, Fig. 2 and Fig. 3. In this way, a detailed temperature map of the full heat sink can be obtained. A two dimensional heat input map can also be used. Finally, the heat sink is assumed to be insulated from the environment and heat transfer is only possible with the coolant in the microchannels.

Figure 3: Cross sectional view of a microchannel heat sink, with representative presentation of a conduction CV.

For each CV, the discrete heat exchanges with neighbouring CV's are modeled as either solid heat conduction or as heat convection with the coolant. For the heat conduction, the general form of Eq. (12) is used. For the heat convection, the form of Eq. (13) is used.

$$\dot{Q}_f = \frac{k_f A_f}{d_f} (T_f - T_p) \quad (12)$$

In Eq. (12), k_f is the heat conduction coefficient, d_f the distance between the temperature nodes, A_f the face area between the nodes and T_f and T_p the temperatures of the neighboring node and the current node.

$$\dot{Q}_f = h_f A_f (T_c - T_p) \quad (13)$$

In Eq. (13), h_f is the convective heat transfer coefficient to the coolant and T_c is the local coolant temperature.

Although a full three dimensional heat sink conduction model is incorporated, in many instances only the stream wise variation of the junction temperature is significant. In that case the three dimensional conduction model is modified to a simpler, computational less costly, one dimensional model. This model still takes into account axial conduction and replaces the conduction through towards the coolant by a simple one dimensional thermal resistance network, taking into account the fin efficiency of the channel walls. Streamwise variation of heat input is also incorporated in this model. However, conduction in the third direction, to the sides of the heat sink, is neglected.

The heat exchanger is modeled as a cross-flow multi-louvered fin heat exchangers using air, see Fig. 4. The discretization for the heat exchanger is performed in the stream wise direction. Thus, a one dimensional temperature profile of the heat exchanger is obtained. The secondary flow is assumed to have a uniform temperature and a constant flow speed.

Figure 4: Cross sectional view of a multi-louvered fin heat exchanger.

Similar to the heat conduction in the heat sink, the discrete heat exchanges with neighbouring CV's are modeled as either solid heat conduction, Eq. (12) or as heat convection with the coolant, Eq. (13). However, for the convection with air, the fin efficiency, η , of the louvered fins is also taken into account. From [27] Eqs. (14) to (16) are obtained for the fin efficiency.

$$\eta = 1 - \frac{A_{fin}}{A_{ru}}(1 - \eta_f) \quad (14)$$

$$\eta_f = \frac{\tanh mH_{fin}/2}{mH_{fin}/2} \quad (15)$$

$$m = \sqrt{\frac{2h_{ru}}{k_{fin}t_f} \left(1 + \frac{t_f}{L_{air}}\right)} \quad (16)$$

In these equations, A_{fin} is the total surface of the fins per CV, H_{fin} is the fin height, k_{fin} is the heat conduction coefficient of the fins, t_{fin} is the fin thickness and L_{air} is the flow depth of the secondary flow.

In the vertical direction for the heat sink, the input heat flow is set with a selected two dimensional heat input map and the coolant temperatures are obtained from the node temperatures in the two-phase flow model equations. For the heat exchanger, the primary coolant temperatures

are also obtained from the flow model equations. The secondary air flow is set to a constant input temperature and flow speed. As mentioned before, the heat sink and heat exchanger are assumed to be isolated from the environment. Thus, for the various boundaries with the environment, the various heat exchanges are set to zero.

3.3. Numerical solution of discretized equations

The numerical solution of the flow is handled using the SIMPLE algorithm [29]. This is an iterative procedure that requires multiple iterations each time step to fully resolve the flow. After establishing the hydraulic properties of the flow using the momentum equation and the pressure correction equation, the enthalpy equation is solved to obtain the flow enthalpy. Once the flow properties are obtained during each iteration, the heat conduction equations are solved for the heat sink and heat exchanger. The solution of these equations produces the temperatures in the heat sink and the discretized heat fluxes towards the coolant in the microchannels. The wall temperatures of the heat exchanger and the heat fluxes between the primary flow and the secondary flow are also obtained. Both these heat fluxes are used as input for the coolant enthalpy equations in the next iteration. This process is repeated until convergence is reached before advancing in time. During each iteration other thermodynamic properties, e.g. vapor quality, are calculated with the REFPROP program [30] using the local calculated pressure and enthalpy. An overview of the algorithm is given in Fig. 5. Convergence is monitored by calculating the residuals of the continuity and enthalpy equation. These are normalized using the residuals of the third iteration of each time step. Convergence is set when these normalized residuals reach 10^{-8} in value.

3.4. Initial and boundary conditions

The pressure, mass fluxes, enthalpy and solid temperatures all need to be initialized to specified values. The boundary conditions for the flow are set in the pump and at the accumulator. At the pump the mass flux is set to a given input value. In the accumulator the initial pressure of the inert gas is set. Further volume changes of the coolant in the loop will change the volume of inert gas in the accumulator and hence modify the absolute pressure of the coolant. To take these pressure changes into account, the inert gas is modeled as an ideal gas. To take into account the cooling of the inert gas, a lumped capacitance model of the accumulator is used.

Writing the first law of thermodynamics for the inert gas, and modeling the heat transfer using a lumped capacitance model results in Eq. (17).

$$m_{ig} c_{v,ig} \frac{dT_{ig}}{dt} = UA_{ig} (T_{env} - T_{ig}) - P_{ig} \frac{dV_{ig}}{dt} \quad (17)$$

In Eq. (17), m_{ig} is the mass, $c_{v,ig}$ the heat capacity at constant volume, T_{ig} the temperature, P_{ig} the pressure and V_{ig} the volume of the inert gas. U is the global heat transfer coefficient between the gas and the environment, A_{ig} is the heat transfer surface area and T_{env} is the environmental temperature. Equation (17) can be rewritten using the ideal gas law to Eq. (18).

$$\frac{dP_{ig}}{dt} = \frac{UA_{ig}}{c_{v,ig}} \left(\frac{r_{ig} T_{env}}{V_{ig}} - \frac{P_{ig}}{m_{ig}} \right) - \frac{c_{p,ig}}{c_{v,ig}} \frac{P_{ig}}{V_{ig}} \frac{dV_{ig}}{dt} \quad (18)$$

In Eq. (18), $c_{p,ig}$ is the heat capacity at constant pressure and r_{ig} is the specific gas constant. Equation (18) is numerically integrated in time to obtain the accumulator pressure. The heat transfer between the accumulator and the environment is modeled taking into account natural convection on the outside of the accumulator and conduction through the accumulator walls. The numeric integration is performed with the common fourth order Runge-Kutta scheme.

Figure 5: Flowchart of the system model.

4. Verification of the code

The verification of the computational code is realized using the method of manufactured solutions. This flexible method has been used to verify a wide range of numerical codes [31, 32], without the need of an exact physical solution for a particular problem. Since two-phase flow is often quite complex, a non-trivial exact physical solution is difficult to obtain and thus hard to compare against. Therefore, the method of manufactured solutions is ideally suited to verify the system model described in this paper.

4.1. Method of manufactured solutions

The partial differential equations of the model are described in Eqs. (1) to (3), with the frictional losses in the momentum equation approximated by $\frac{G^2}{2\rho} \frac{P_m}{A}$ to capture the main effect of mass flux and mixture density on the pressure losses. Because the system model uses REFPROP to calculate the material properties, arbitrary equations are used to calculate these properties from the pressure and enthalpy with the following general form:

$$\xi = a_{\xi,1} + a_{\xi,2}P + a_{\xi,3}h \quad (19)$$

The values for the coefficients $a_{\xi,n}$ are selected to be somewhat realistic for each material property and to facilitate convergence of the code. They typically vary between 0.1 and 10. The manufactured solutions for G , p and h for this set of differential equations are:

$$G = a_{g,1} + a_{g,2} \sin\left(\frac{2\pi s}{L_t}\right) + a_{g,3} \cos\left(\frac{2\pi t}{T_t}\right) \quad (20)$$

$$P = a_{p,1} + a_{p,2} \sin\left(\frac{2\pi s}{L_t}\right) + a_{p,3} \cos\left(\frac{2\pi t}{T_t}\right) \quad (21)$$

$$h = a_{h,1} + a_{h,2} \sin\left(\frac{2\pi s}{L_t}\right) + a_{h,3} \cos\left(\frac{2\pi t}{T_t}\right) \quad (22)$$

Again the values of the coefficients are selected to facilitate convergence. These typically vary between 0.5 and 1.5. The spatial domain is set from 0 to L_t and the time domain varies from 0 to T_t for these solutions. The solutions are then inserted in Eqs. (1) to (3), together with the material property equations, Eq. (19), to obtain the source terms for continuity, momentum and enthalpy equation respectively. For brevity's sake these source terms are omitted here.

Code verification is performed using the order of accuracy test [31]. In the system model, implicit, first order Euler is used for time integration. For spatial discretization, TVD schemes are used for the convective terms and a central difference scheme is used for the diffusive terms. Therefore, on a smooth grid, the spatial formal order of accuracy is of the second order. To calculate the observed order of accuracy, numerical solutions are needed on different mesh sizes for the spatial accuracy and with different time steps for the temporal accuracy. Therefore, numerical solutions are obtained on 7 different meshes using a small time step, to minimize the error induced from the time discretization, and with 6 different time steps on a fine mesh, to minimize the error induced from the spatial discretization. The refinement factor, r or the ratio between the coarse mesh and fine mesh element size, in both cases is set to $r = 4$. Additionally, the grid spacing and the time steps are normalized by the grid spacing of the finest mesh and the smallest time step respectively. The different meshes and time steps used are tabulated in table 1 and table 2.

Mesh	Number of nodes	Normalized grid spacing
1	40960	1
2	10240	4
3	2560	16
4	640	64
5	160	256
6	40	1024
7	10	4096

Table 1: Different meshes used to calculate the observed order of spatial accuracy

Time step size	Normalized time step
0.00390625	1
0.015625	4
0.0625	16
0.25	64
1	256
4	1024

Table 2: Different time step sizes used to calculate the observed order of temporal accuracy (note that T_t is always set at 4 s)

The discretization error of G , p and h is evaluated using the L_∞ and L_2 norms of the difference between the calculated results and the manufactured solutions. Using these error norms, the observed order of accuracy can be calculated from Eq. (23).

$$p = \frac{\ln\left(\frac{L_{k+1}}{L_k}\right)}{\ln r} \quad (23)$$

In Fig. 6 the observed order of spatial accuracy is plotted for the different normalized grid spacings. From Fig. 6 it is clear that the L_∞ and L_2 norms of all the variables closely match the formal order of accuracy, namely $p = 2$. Even at the higher normalized grid spacing, the observed order of accuracy matches closely the formal order of accuracy. In Fig. 7 the observed order of temporal accuracy is plotted for the different normalized time step size. At a higher normalized time step, the observed order of accuracy differs somewhat from the formal order of accuracy for the pressure and mass flux. This is probably due to the quite large time steps in that range and the solutions might not be in the asymptotic grid convergence range. However, the formal order of accuracy, being $p = 1$, is approached at smaller time step sizes.

Figure 6 and Fig. 7 indicate that the code solves the underlying equations correctly. In addition both the spatial and temporal order of accuracy clearly reflect the formal order of accuracy at smaller time steps or grid spacing. Therefore, for these particular discretization schemes the code is reasonably well verified.

Figure 6: Observed order of spatial accuracy in function of the normalized grid spacings.

5. Validation of illustrative results

Single-phase and two-phase results obtained from the model are validated with experimental data acquired from a microchannel flow loop facility at Purdue university. This test setup has been used in previous studies to obtain experimental data on two-phase flow boiling in microchannels [33]. The coolant used for the experiments is FC-77. A detailed description of the test setup can be found in [33].

For the current measurements, a 12.663 mm wide by 12.670 mm long, silicon heat sink consisting of 24 microchannels with 365 μm height and 395 μm width is used. The thickness of the heat sink base is 285 μm and the average wall thickness is 105.5 μm . The channels are cut into the silicon base by precision saw cutting. Heat input to the heat sink is provided by a test chip consisting of an array of 5 x 5 heating elements. Furthermore an array of 5 x 5 thermal diodes to measure a complete 2 dimensional temperature map of the chip is also incorporated [33]. Figure 8 shows a diagram of the heat sink and diode map. The uncertainties on the diode temperature measurements are ± 0.3 °C.

Experimental single-phase results are obtained by varying the heat input 3 times ($Q_{low} = 11.530 \pm 0.001$ W, $Q_{med} = 20.041 \pm 0.002$ W and $Q_{high} = 30.097 \pm 0.003$ W). The volumetric flow rate is set to $1.428 \pm 0.014 \cdot 10^{-6} \frac{m^3}{s}$, the inlet temperature of the heat sink is kept at 24 ± 0.3 °C and the absolute pressure at the inlet of the heat sink is 1.022 ± 0.003 bar. This corresponds to a boiling temperature of about 97°C. Between each increase, the temperature reached steady state before increasing the heat input further. The heat losses to the environment are estimated using a

Figure 7: Observed order of temporal accuracy in function of the normalized time step size.

specifically developed correlation with the junction temperatures as input. This correlation was developed by measuring heat losses of the heat sink and test chip assembly before it was plugged in the flow loop [33].

The numerical model is set up to simulate the test loop used to obtain the experimental results. The various components of the test loop are incorporated in this simulation. These include a pump, preheater, inlet and outlet manifolds, microchannel heat sink, air-liquid heat exchanger, an accumulator and various tubing connecting these components. The layout of the system is similar to Fig. 1, except with the addition of a preheater before the heat sink. Since the focus of the current validation is on the microchannel heat sink, the other components in the loop are discretized with a low number of CV's, using on average 40 CV's per component. The heat sink itself is discretized using 120 CV's. The total number of flow CV's is 400. For the simplified conduction model, an additional 160 conduction CV's are added. For the full 3D conduction model, an additional 43240 CV's are added. It is clear that the full 3D simulation will be much more computationally costly than the simplified model. For the current simulations, the simplified model reaches a steady-state solution using a single Intel Core 2 E6700 2.66 GHz CPU, on average, in 2 cpu minutes. The full 3D simulation reaches a solution, on average, in 5 cpu hours.

The numerical results of the model are verified using the grid convergence index, or GCI, from Roache [31, 32]. This method which is based on the Richardson extrapolation error estimation provides an estimate for the uncertainty on the numerical results. In particular it gives an estimate for the 95 % confidence intervals for the solution. The formula for the GCI is given in

Figure 8: Diagram of the microchannel heat sink and the heater and diode map. The numbers on the map indicate the row number.

Eq. (24).

$$GCI = \frac{F_s}{r^p - 1} \left| \frac{f_2 - f_1}{f_1} \right| \quad (24)$$

In Eq. (24), F_s is a safety factor, f_2 and f_1 are the results from a coarse grid and a fine grid respectively. In order to properly set the safety factor the observed order of accuracy, p , has to be checked with the formal order of accuracy. The observed order of accuracy is calculated from Eq. (25).

$$p = \frac{\ln\left(\frac{f_3 - f_2}{f_2 - f_1}\right)}{r} \quad (25)$$

By obtaining the temperature of the active silicon layer for three different grid sizes, with a grid refinement factor of $r = 2$, and using Eq. (25) the observed order of accuracy can be calculated. This results in a mean order of accuracy over the entire heat sink of 1.877, which is fairly close to the formal order of accuracy of 2. Therefore a safety factor of $F_s = 1.25$ is selected [31, 32]. Using Eq. (24) to calculate the GCI gives a mean GCI of about 0.1% which is quite low. Although the uncertainties associated with the numerical convergence are low, uncertainties on the heat sink dimensions, coolant temperature and heat input of the heating elements will result in uncertainties on the numerical results. Following a standard uncertainty analysis the total modeling error is estimated to be 4 %. Note that the uncertainties arising from the used correlations are not included, because of the difficulty in accurately quantifying these. Therefore the modeling error estimate is underestimated.

The comparison between the numerical and experimental results is plotted in Fig. 9. The numerical results are for the simplified conduction model. The error bars of the experimental results are omitted because of their small size. As can be seen from Fig. 9, the temperatures from the model show good agreement with the experimental temperatures. The overall temperature trend is captured by the model and the absolute values are also well predicted.

Additionally two-phase results are also obtained by setting the inlet temperature of the heat sink to 92 ± 0.3 °C using a preheater. Again the heat input is varied 3 times ($Q_{low} = 24.876 \pm 0.002$ W, $Q_{med} = 33.082 \pm 0.003$ W and $Q_{high} = 42.879 \pm 0.004$ W). The other parameters are the same as for the single-phase results. The results are plotted in Fig. 10, again using the simplified conduction model. The 95 % confidence intervals of the numerical results are omitted for clarity.

Figure 9: Streamwise variation of the junction temperatures in the microchannel heat sink (single-phase flow). The markers show experimental results, the lines show numerically predicted results using the system model. With case A being low heat input, B medium heat input and C high heat input. The dashed lines show the 95 % confidence intervals of the numerical results.

From Fig. 10 it is clear that the model again predicts the temperatures fairly well. Although not shown on the figure, all of the experimental results fall in the 95 % confidence intervals of the numerical results.

The increased deviation between the experimental and numerical results, in comparison with those of the single-phase experiments, could be due to several factors. First, of all the two-phase results are at an elevated temperature in comparison to the single-phase results. This means that there will be more losses to the environment and these are difficult to quantify and incorporate in the numerical model. Some estimates of these losses is possible, as mentioned in the description of the experimental setup, but in general the error on the losses would be quite high. Second, the influence of the in- and outlet manifolds on the temperature distribution can be quite large. If these manifolds are thermally well connected to the silicon heat sink, these will act as large fins extending into the flow. Therefore, these could alter the streamwise temperature distribution significantly. In the numerical model, these manifolds are weakly connected to heat sink. However, the true thermal resistance between these is difficult to accurately quantify. Third, the streamwise conduction in the heat sink becomes more important. In many experimental studies, the two-phase heat transfer coefficient increases with the heat flux, this is reflected in the correlations used in the model [3]. Therefore, if the streamwise conduction improves, the heat flux distribution can shift more to the two-phase region, without having an increase in the local wall temperatures. The net effect of this is that the overall junction temperature will decrease. This ef-

Figure 10: Streamwise variation of the junction temperatures in the microchannel heat sink (two-phase flow). The markers show experimental results, the lines show numerically predicted results using the system model. With case A being low heat input, B medium heat input and C high heat input.

fect can be quite large and uncertainties in the streamwise cross section of the heat sink will have a large influence on the junction temperature. Finally, the two-phase correlations themselves inherently are not as precise as the, typically laminar, single-phase correlations. Especially the transition from single-phase flow to two-phase flow is difficult to model. In the numerical model, this transition is done using a spline interpolation to ensure a smooth transition. In all of the two-phase results, this transition typically encompasses 30 % of the total heat sink length and thus has a large influence.

In Fig. 11, the two-phase temperature map is plotted with $Q_{med} = 33.082 \pm 0.003 \text{ W}$ heat input. The numerical results obtained are with the full 3D conduction model. Because the heat input is quite uniform, the radial temperature difference, of both the experimental and numerical results, is quite small. Again the numerical model predicts the temperatures fairly well. The larger radial temperature difference in the experimental results could be due to the radial heat losses to the environment. These losses are not included in the full 3D conduction model. The discussion on the increased deviation of Fig. 9 also hold for this figure. Also note that the results in Fig. 11 correspond well with the results obtained with the simplified model in Fig. 10, for similar conditions. This is again due to the almost perfectly uniform heat input.

Figure 11: Map of the junction temperatures for two-phase flow using medium heat input. The markers show experimental results, the lines show numerically predicted results using the system model. The row number correspond to the numbers in Figure 8.

6. Conclusion

A dynamic system model for electronics cooling using two-phase flow boiling and microchannels is developed. This model is able to simulate the transient and steady state behavior of such a cooling system.

Conservation of mass, momentum and enthalpy of the flow is modeled. To close the equations state of the art empirical correlations are used. The choice of the correlations is based on their performance in various experimental studies.

The conservation equations are numerically solved using the SIMPLE algorithm. TVD schemes are used to discretize the convective terms to ensure boundedness at sudden flow changes while still maintaining second order accuracy where the solution is smooth. Additionally these schemes are modified to take into account the non-uniform grid of the model. Finally these higher-order schemes are incorporated using the deferred correction method to ensure numerical stability.

The heat conduction in the heat sink is modeled using a transient three dimensional conduction model of the heat sink. While this model is accurate it is computational costly. So a second transient one dimensional heat conduction model of the heat sink is also included in the system model. Also the transient one dimensional heat conduction in the heat exchanger is incorporated in the system model. Furthermore the heat exchanger itself is modeled as a multi-louvered finned heat exchanger with air.

The numerical code itself is verified using the method of manufactured solutions. The observed order of accuracy matched closely the formal order of accuracy both for the spatial and the temporal discretization. Thus the code itself is well verified to produce a correct solution of the underlying partial differential equations.

Finally some illustrative results from the model are validated using experimental data. The numerical results show a good agreement with the experimental temperatures.

Acknowledgements

The authors are grateful to Prof. S. Garimella and Dr. T. Harirchian for their support while collecting the experimental data from the Purdue test facility. This work is sponsored by the IWT, The Institute for the Promotion of Innovation by Science and Technology in Flanders, Belgium, project SBO 60830, HyperCool-IT.

Nomenclature

Variables

A	cross sectional area [m^2]
a	general coefficient [-]
C_c	contraction coefficient [-]
c_p	heat capacity at constant pressure [$J/(kgK)$]
c_v	heat capacity at constant volume [$J/(kgK)$]
d	distance between temperature nodes [m]
D_h	hydraulic diameter [m]
F	frictional and local pressure losses [N/m^3]
f	Darcy friction factor [-]
F_s	safety factor [-]
G	mixture mass flux [$kg/(m^2s)$]
g	gravitational acceleration [m/s^2]
H	height [m]
h	convective heat transfer coefficient [$W/(m^2K)$]
h	mixed-cup enthalpy [J/kg]
h^*	mixture specific enthalpy [J/kg]
I	factor to convert h^* to h [m]
K	local pressure loss factor [-]

k	thermal conductivity [$W/(mK)$]
$\dot{m}$	mass flow [kg/s]
m	mass [kg]
P	pressure [Pa]
p	order of accuracy [-]
P_m	perimeter of flow channel [m]
q	heat flux [W/m^2]
r	refinement factor [-]
r	specific gas constant [$J/(kgK)$]
Re	Reynolds number [-]
s	streamwise coordinate [m]
t	thickness [m]
t	time [s]
T_c	mixture temperature [K]
U	global heat transfer coefficient [$W/(m^2K)$]
u	velocity [m/s]
V	volume [m^3]
x	vapor quality [-]

Greek Symbols

α	void fraction [-]
η	fin efficiency [-]
μ	dynamic viscosity [$Pa\cdot s$]
ρ	mixture density [kg/m^3]
ρ'	apparent momentum density [kg/m^3]
ρ''	apparent energy density [kg/m^3]
σ	flow area contraction [-]
θ	inclination degree [-]

Subscripts

e	east direction
-----	----------------

<i>f</i>	arbitrary direction
<i>fin</i>	fins
<i>g</i>	vapor
<i>env</i>	environment
<i>ig</i>	inert gas
<i>l</i>	liquid
<i>p</i>	current node
<i>w</i>	west direction

- [1] W. Qu, I. Mudawar, Flow boiling heat transfer in two-phase micro-channel heat sinks - 2. annular two-phase flow model, *International Journal of Heat and Mass Transfer* 46 (2003) 2773–2784.
- [2] J. Thome, V. Dupont, A. Jacobi, Heat transfer model for evaporation in microchannels. part 1: presentation of the model, *International Journal of Heat and Mass Transfer* 47 (2004) 3375–3385.
- [3] S. Bertsch, E. Groll, S. Garimella, A composite heat transfer correlation for saturated flow boiling in small channels, *International Journal of Heat and Mass Transfer* 52 (2009) 2110–2118.
- [4] M. Willatzen, N. Pettit, L. Ploung-Sorensen, A general dynamic simulation model for evaporators and condensers in refrigeration. part 1-2, *International Journal of refrigeration* 21 (1998) 398–414.
- [5] T. Saenen, M. Baelmans, Modeling size effects of a portable two-phase electronics cooling loop with different refrigerants, in: *Proceedings of the International Heat Transfer Conference, IHTC 14, ASME, Washington DC (USA)*.
- [6] S. Ghiaasiaan, *Two-Phase Flow, Boiling, and Condensation in Conventional and Miniature Systems*, Cambridge University Press, New York, 2008.
- [7] A. Premoli, D. Francesco, A. Prina, A dimensionless correlation for determining the density of two-phase mixtures, *Lo Termotecnica* 25 (1971) 17–26.
- [8] W. Zhang, T. Hibiki, K. Mishima, Correlations of two-phase frictional pressure drop and void fraction in mini-channel, *International Journal of Heat and Mass Transfer* 53 (2010) 453–465.
- [9] S. Zivi, Estimation of steady state steam void-fraction by means of principle of minimum entropy production, *ASME Trans. Series C* 86 (1964) 237–252.
- [10] F. White, *Fluid Mechanics*, McGraw-Hill, New York, 5 edition, 2003.
- [11] R. Lockhart, R. Martinelli, Proposed correlation of data for isothermal two-phase, two-component flow in pipes, *Chemical Engineering Progress* 45 (1949) 39–48.
- [12] T. Yilmaz, General equations for pressure drop for laminar flow in ducts of arbitrary cross sections, *Journal of Energy Resources Technology* 112 (1990) 220–223.
- [13] R. Shah, A correlation for laminar hydrodynamic entry length solutions for circular and non-circular ducts, *ASME J. Fluids Eng.* 100 (1978) 177–179.
- [14] K. Mishima, T. Hibiki, Some characteristics of air-water two-phase flow in small diameter vertical tubes, *Int. J. Multiphase Flow* 22 (1996) 703–712.
- [15] R. Ali, B. Palm, M. Maqbool, Experimental investigation of two-phase pressure drop in a microchannel, *Heat Transfer Engineering* 32 (2011) 1126–1138.
- [16] P. Lee, S. Garimella, Saturated flow boiling heat transfer and pressure drop in silicon microchannels arrays, *International Journal of Heat and Mass Transfer* 51 (2008) 789–806.
- [17] J. Lee, I. Mudawar, Two-phase flow in high-heat-flux micro-channel heat sink for refrigeration cooling applications: Part 1 - pressure drop characteristics, *International Journal of Heat and Mass Transfer* 48 (2005) 928–940.
- [18] G. Celata, M. Cumo, S. McPhail, G. Zummo, Characterization of fluid dynamic behaviour and channel wall effects in microtube, *International Journal of Heat and Fluid Flow* 27 (2006) 135–143.
- [19] W. Kays, Loss coefficients for abrupt changes in flow cross section with low reynolds number flow in single and multiple tube systems, *Transactions of the ASME* (1950) 1067–1074.
- [20] F. Abdelall, G. Hahn, S. Ghiaasiaan, S. Abdel-Khalik, S. Jeter, M. Yoda, D. Sadowski, Pressure drop caused by abrupt flow area changes in small channels, *Experimental Thermal and Fluid Science* 29 (2005) 425–434.
- [21] G. Geiger, *Sudden Contraction Losses in Single and Two-Phase Flow*, Ph.D. thesis, University of Pittsburgh, 1964.
- [22] D. Liu, P. Lee, S. Garimella, Prediction of the onset of nucleate boiling in microchannel flow, *International Journal of Heat and Mass Transfer* 48 (2005) 5134–5149.

- [23] Y. Muzychka, M. Yovanovich, Laminar forced convection heat transfer in the combined entry region of non-circular ducts, *Journal of Heat Transfer-Transactions of the ASME* 126 (2004) 54–61.
- [24] V. Gnielinski, New equations for heat and mass transfer in turbulent pipe and channel flow, *International Chemical Engineering* 16 (1976) 359–368.
- [25] E. Costa-Patry, J. Olivier, B. Michel, J. Thome, Two-phase flow of refrigerants in 85 μ m-wide multi-microchannels: Part 2 - heat transfer with 35 local heaters, *International Journal of Heat and Fluid Flow* 32 (2011) 464–476.
- [26] M. Shah, A general correlation for heat transfer during film condensation inside pipes, *International Journal of Heat and Mass Transfer* 22 (1979) 547–556.
- [27] M. Kim, C. W. Bullard, Air-side thermal hydraulic performance of multi-louvered fin aluminum heat exchangers, *International Journal of Refrigeration* 25 (2002) 390–400.
- [28] N. Waterson, A symmetric formulation for flux-limited convection schemes, *International Journal for Numerical Methods in Fluids* 56 (2008) 1575–1581.
- [29] S. Patankar, *Numerical Heat Transfer and Fluid Flow*, McGraw-Hill, New York, 1980.
- [30] E. Lemmon, M. Huber, M. McLinden, NIST Standard Reference Database 23: Reference Fluid Thermodynamic and Transport Properties-REFPROP, Version 8.0, National Institute of Standards and Technology, Standard Reference Data Program, Gaithersburg, 2007.
- [31] C. Roy, Review of code and solution verification procedures for computational simulation, *Journal of Computational Physics* 205 (2005) 131–156.
- [32] P. Roache, *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, Albuquerque, NM, 1998.
- [33] T. Harirchian, S. Garimella, Microchannel size effects on local flow boiling heat transfer to a dielectric fluid, *International Journal of Heat and Mass Transfer* 51 (2008) 3724–3735.