Numerical simulation of heat transfer during the liquid cooling of complex surfaces at unsteady heat release

A N Chernyavskiy and A N Pavlenko
Kutateladze Institute of Thermophysics SB RAS, 1 Lavrentiev ave., Novosibirsk, 630090, Russia

E-mail: cherny@ngs.ru

Abstract. The presented mathematical model allows simulating the heat transfer process during the liquid cooling of complex surfaces under conditions of unsteady heat release. The numerical simulation can be provided for the case of film cooling as well as for channel heat exchangers. The presented simulation approach is based on the Finite Volume Method (FVM) which makes it possible to perform the calculations for heat exchangers with complex geometry. Usage of the Volume of Fluid (VOF) method allows simulating the heater cooling by the liquid film with a free surface. The presented mathematical model was tested on a set of specific problems. The simulation results are in satisfactory agreement with the known solutions.

1. Introduction
The liquid cooling is widely used in different kinds of heat exchangers. It is known that the usage of complex heating surfaces makes it possible to increase the intensity of the heat transfer process and reduce the size of heat exchanges [1-3].

The area of high intensity of the heat transfer process is limited by the heat flux rate and depends on the heat release law [4]. Reaching the critical value of the heat flux rate leads to decay of the film flow with the formation of large-scale dry zones, reduction of heat transfer intensity, and as a consequence, to a drastic increase in temperature of the heating surface which can destroy the heater element. The ability to calculate the critical heat fluxes and maximal times of their impact on the given system is required for the development of the stably operating heat exchanging systems.

For the moment, there are a lot of different works devoted to liquid flow simulation. The detailed overviews of multiphase flow simulation methods are given in [5–8]. Unfortunately, there is no background for heat transfer simulation under conditions of unsteady heat release in areas of complex geometry.

The presented mathematical model makes it possible to simulate the heat transfer process during the liquid cooling of complex surfaces at unsteady heat release. The numerical simulations can be provided for different kinds of heat exchangers with different heating surface geometry. The results of calculations obtained using the presented mathematical model were tested on a variety of specific problems. The simulation results are in satisfactory agreement with the known solutions.

2. Mathematical model
Simulation of such a complex phenomenon like unsteady heat transfer in moving liquid means searching for the simultaneous solutions of hydrodynamic and heat transfer problems. In general, both of the presented problems can be described by a set of partial differential equations with appropriate initial and boundary conditions.
The most common approach to describe the flow dynamics of viscous liquid is based on the Navier-Stokes equation. For the purposes of the current work it is possible to use the incompressible fluid variation which looks like the following:

\[
\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla)\vec{u} = -\frac{1}{\rho} \nabla p + \nu \Delta \vec{u} + \vec{f}.
\] (1)

The presented Navier-Stokes equation can be supplemented by the continuity equation. In the case of incompressible fluid it looks like this:

\[
\nabla \vec{u} = 0.
\] (2)

Calculation of the pressure field can be performed using one of the modifications of SIMPLE method [8].

The temperature field can be calculated using the unsteady heat conduction equation:

\[
\rho C \left[ \frac{\partial T}{\partial t} + (\vec{u} \cdot \nabla)T \right] = \nabla (\lambda \nabla T) + q_v.
\] (3)

In the case of constant heat conduction coefficient, equation (4) can be presented in the following form:

\[
\frac{\partial T}{\partial t} + (\vec{u} \cdot \nabla)T = a \Delta T + \frac{q_v}{\rho C}.
\] (4)

The presented simulation approach is based on the Finite Volume Method (FVM) [9]. One of the most significant advantages of FVM is the possibility to perform the simulation of problems with complex geometry. This method can be used with regard to all differential equations which can be written in the divergence form. So, it is possible to use it for Navier-Stokes equations as well as for the heat conduction equation. The idea of FVM is to divide the calculation area on a set of finite volumes interacting by fluxes through their surfaces (see Figure 1).

![Figure 1. Control volume in FVM.](image)
Let’s describe the evolution of temperature in time using FVM approach for rectangular cell $P$ with four boundaries called «north», «west», «east» and «south». First of all, let’s integrate (3) inside all the selected cell volume:

$$
\int_V \rho C \frac{\partial T}{\partial t} dV + \int_V \rho C (\vec{u} \cdot \nabla) T dV = \int_V \nabla (\lambda \nabla T) dV + \int_V q_i dV .
$$

Equation (5) can be transformed using the following Gauss’s theorem (6) to look like (7).

$$
\int_S \nabla \cdot \vec{F} dV = \int_S (\vec{F} \cdot \vec{n}) dS ,
$$

$$
\int_V \rho C \frac{\partial T}{\partial t} dV + \int_S \rho C (\vec{u} \cdot \vec{n}) T dS = \int_V \nabla (\lambda \nabla T) dV + \int_V q_i dV .
$$

Taking into account the equivalence of mean value of some magnitude in the selected cell, let convert (7) to (8):

$$
\rho_1 C_1 V_1 \frac{\partial T_1}{\partial t} + \int_S \rho_1 C_1 (\vec{u}_1 \cdot \vec{n}) T_1 dS = \int_S (\lambda \nabla T_1 \cdot \vec{n}) dS + q_{1,v} V_1 ,
$$

where the heat-conducting term of the equation can be presented as a sum of heat fluxes on the edges of the neighbor cells:

$$
\int_S (\lambda \nabla T \cdot \vec{n}) dS = \sum_f \overline{(q_f \cdot \vec{n}_f) S_f} ,
$$

$$
\rho_1 C_1 V_1 \frac{\partial T_1}{\partial t} + \int_S \rho_1 C_1 (\vec{u}_1 \cdot \vec{n}) T_1 dS = \sum_f \overline{(q_f \cdot \vec{n}_f) S_f} + q_{1,v} V_1 ,
$$

where $f$ means the value of some magnitude in the middle of one of the neighbor cell edges.

There are several ways to calculate some value in the middle of some edge. The easiest way is to use the half-sum of the appropriate magnitude value of the current cell and the neighbor cell:

$$
A_f = \frac{A_i + A_j}{2} .
$$

This approach cares about only one neighbor and only one dimension. It is possible to increase the accuracy using some multi-cell and multi-dimensional approximation.

As for the heat flux sum, for a rectangular cell with four boundaries called «north», «west», «east» and «south» it looks like the following:

$$
\sum_f \overline{(q_f \cdot \vec{n}_f) S_f} = q_n S_n - q_x S_x + q_e S_e - q_w S_w = (q_n - q_x) S_x + (q_e - q_w) S_y .
$$

Setting of the boundary conditions of the first or the second kind seems to be a rather simple task in the case of the presented approach. The only thing to be done is specifying the temperature value or the heat flux from the appropriate edge:

$$
T_f = T_0 ,
$$

$$
q_f = q_0 .
$$

All other cell parameters can be specified in the same way.

Because of the analogy between the convective transfer of heat conduction and Navier-Stokes equation, it is possible to use the following discretization approach [10]:
$$F_f = S_f \left( \overline{n_f} \cdot \overline{u_f} \right), \quad (15)$$

$$\rho_p C_p V_p \frac{\partial T_p}{\partial t} + \sum_f \rho_f C_f F_f T_f - T_p \sum_f \rho_f F_f = \sum_f \left( \overline{q_f} \cdot \overline{n_f} \right) S_f + q_{v,p} V_p. \quad (16)$$

Equation (15) defines the liquid consumption through the given surface of the control volume. In terms of the presented approach, the Navier-Stokes equation looks like the following [7]:

$$\rho_p V_p \frac{\partial u_{i,p}}{\partial t} + \sum_f \rho_f u_{i,f} F_f - u_{i,p} \sum_f \rho_f F_f = RHS, \quad (17)$$

where $RHS$ is the sum of the diffusion terms and momentum sources.

Another method which makes it possible to simulate the flow of liquid with a free surface is the Volume of Fluid (VOF) method [11]. The idea of VOF is to consider the flow as a liquid-gas mixture. The idea of the method can be implemented by adding one extra property to all the finite volumes of FVM with the meaning of the amount of liquid in the volume, i.e. 1 means fully-liquid and 0 means fully-gaseous control volume. The current state of the free surface can be found by analyzing the volumes with the transitional value of such property.

It is necessary also to add one extra equation to the problem set to make it possible to track the evolution of phase concentrations. In case of the absence of phase transitions the noticed equation looks like the following one:

$$\frac{\partial C}{\partial t} + (\overline{u} \cdot \nabla)C = 0. \quad (18)$$

According to the FVM approach equation (18) can be presented in the following form:

$$V_p \frac{\partial C_p}{\partial t} + \sum_f C_f F_f - C_p \sum_f F_f = 0. \quad (19)$$

All the cell parameters like density, viscosity, heat conductivity, etc. become to be the weighted sums of the same parameters of pure gas and liquid:

$$A_p = C_p A_l + (1-C) A_g. \quad (20)$$

In a closed system without phase transitions, the total amounts of liquid and gas phases should be the constants.

3. Results and discussion
The presented mathematical model has been tested through a set of simple problems with known solutions. To demonstrate the possibility of using this model for solving the complex geometry problems, the following simulations have been performed:

- the first one is an example of unsteady heat transfer during the liquid cooling of a wavy-curved heating element;
- the second one is an example of liquid cooling of four heat-releasing shafts.

Both of the demonstration simulations have been performed using water at 20°C as a cooling liquid under the conditions of isothermal walls on the boundaries of the simulation domain. For both simulations, the evolution of the cooling liquid temperature in time has been retrieved. All the presented temperature fields are given in degrees Celsius.

The results of the numerical simulation of the unsteady heat transfer process during the liquid cooling of the wavy-curved heating element are presented in Figure 2.
Figure 2. Unsteady heat transfer during the liquid cooling of the wavy-curved heating element.

The results of numerical simulation of the unsteady heat transfer process during the liquid cooling of four heat-releasing shafts are presented in Figure 3.
Figure 3. Unsteady heat transfer during the liquid cooling of four heat-releasing shafts.

As it was shown above, the presented mathematical model makes it possible to simulate the unsteady heat transfer process in cases of problems with complex geometry. The simulations of the demonstration problems with free surface evolution are in progress now and will be available soon.
Acknowledgments
This research was carried out at the IT SB RAS with the financial support of the Russian Foundation for Basic Research (project No. 18-08-00402_a) and the Federal Program for Basic Research of State Academies of Sciences for 2013-2020 (project No. III.18.2.3, AAAA-A17-117030310025-3).

References
[1] Volodin O, Pecherkin N, Pavlenko A and Zubkov N 2020 Int. J. Heat Mass Tr. 155 119722
[2] Volodin O, Pecherkin N, Pavlenko A and Zubkov N 2019 J. Phys.: Conf. Ser. 1369 012046
[3] Volodin O, Pecherkin N, Pavlenko A and Zubkov N 2017 Interf. Phen. Heat Tran. 5 215–22
[4] Pavlenko A, Surtachev A and Matsekh A 2007 High Temp. 45 826–36
[5] Leonov A, Chudanov V and Aksenova A 2013 Methods of direct numerical simulation in two-phase media (Moscow: Nauka)
[6] Kolev N 2007 Multiphase flow dynamics (Berlin, Heidelberg: Springer-Verlag)
[7] Ferziger J and Peric M 2001 Computational methods fluid dynamics (Berlin: Springer)
[8] Fletcher C 1988 Computational techniques for fluid dynamics (Berlin: Springer-Verlag)
[9] Rapp B 2017 Microfluidics: Modelling, Mechanics and Mathematics (Elsevier)
[10] Khrabry A, Smirnov E and Zaytsev D 2013 Proc. Krylov state research centre 78 53–64
[11] Hirt C and Nichols B 1981 J. Comput. Phys. 39 201–25