Numerical simulation of moisture extraction under the influence of an acousto-convective flow from a thin cavern

I A Bedarev\textsuperscript{1}, A A Zhilin\textsuperscript{1}, S A Lavruk\textsuperscript{1}, V I Trushlyakov\textsuperscript{2}

\textsuperscript{1}Khristianovich Institute of Theoretical and Applied Mechanics SD RAS, Institutskaya 4/1, 630090, Novosibirsk, Russia
\textsuperscript{2}Omsk State Technical University, 11 Mira ave., Omsk, 644050, Russia
lavruks@mail.ru

Abstract. With using the mechanics of multiphase media, the process of extraction of moisture from a dead pore is considered. The study was carried out by numerical simulation. Two-dimensional approach of interaction of the air flow with droplets without taking into account the processes of heat and mass transfer was considered.

1. Introduction

The processes of liquid evaporation are described in application to various tasks. Some evaporation process problems are connected with fuel evaporation during the combustion of liquid droplets. Models for solving such problems were constructed in [1–4], where the processes of evaporation of single droplets of hydrocarbon fuel both in a stationary and non-stationary setting were considered. Their ignition was also considered, as well as the transition from subsonic to supersonic combustion. The technology of acousto-convective drying was developed at ITAM SB RAS. It showed a significant intensification of the process of liquid extraction from various porous materials. The research results were published for meat [5], rice [6], pine nuts [7], granular silica gel [8], etc. One of the main advantages of this technology is the absence of heating of the material to be dried, i.e. drying takes place at room temperature [9].

In [10], theoretical and experimental studies on the evaporation of liquid from the surface of an aluminum base due to the supply of a jet of hot gas are presented. In [11], a numerical simulation of gas flows in the experimental vessel described in [10] is carried out. A study on modeling the evaporation of liquid films in pipes is presented in [12]. In [12], when using the Ansys Fluent software package for modeling an air flow with a liquid film in a pipe, the results on the pressure drop coincide with the data of the experimental studies, and the results are consistent with the studies of other authors.

The authors [13–15] carried out theoretical and experimental studies on the evaporation of liquid under conditions of low ambient pressure, as well as acoustic action. The agreement between experimental data and calculation results is obtained. The equations of conservation laws are used as a mathematical model, supplemented by semi-empirical models to describe heat transfer and acoustic impact.

In this paper, the process of liquid mass transfer from a thin cavern will be considered. The calculation is carried out similarly to the studies carried out in [10,11], however, in contrast to [10,11], the interaction of two phases (a jet of gas and liquid in a pore) was taken into account using the mechanics of heterogeneous media. The simulation is carried out in the ANSYS Fluent.
2. Physical and numerical model

Requirements for the mathematical model
For the considered mathematical model, the following requirements are considered:
1. The physical and mathematical model is based on models of the mechanics of multiphase media with taking into account the interaction of continua.
2. The basic equations follow from the laws of conservation of mass, momentum and energy and are closed by the equations of state of an ideal gas and the laws of heat and mass transfer, as well as the drops drag laws.
3. The volume of fluid consists of droplets of circular cross-section, the size of which can vary.
4. Modeling is carried out taking into account the total pressure for both phases, supplemented, if necessary, by an equation for describing chaotic motion.
5. The acoustic-convective effect is modeled using a periodic boundary condition for the static pressure at the channel inlet.

Equations and boundary conditions
Gas dynamics of the gas-liquid interaction process is considered. The Euler - Euler model is used to simulate a two-phase flow. The ANSYS Fluent software package is used as a software program.
During simulation two-phase flows and interphase interaction, the particles of the second phase are modeled as circular droplets, for which the initial diameter is specified. For this study, the droplet size varied in the range \(d_p=10^{-4} \ldots 10^{-6}\) m.
To simulate the flow in the volume, we use the method of numerical solution of the Favre averaged Navier-Stokes equations, supplemented by a two-parameter model of k-\(\omega\) turbulence in the SST modification. The basic equations can be found in [16].
The acoustic effect on the flow in the capillary at this stage of research can be described using a periodic boundary condition on the static pressure:
\[
P = P_{op} + A \cdot \sin(2\pi \cdot f \cdot t),
\]
where \(P_{op}\) is reference pressure, \(A\) is amplitude of pressure oscillation, \(f\) is vibration frequency.

3. Simulation of flow in a single pore on a plate

3.1. Simulation of gas flow
Calculations of the air flow over the plate at various flow velocities have been performed. A single pore (capillary) was located in a certain section of the plate. At the initial stage of modeling, the pore was not filled with water. The calculations were carried out in a two-dimensional setting. Capillary depth - 10 mm, width - 1 mm. Fig. 1 shows the geometry of the channel and a fragment of the computational grid in the pore region. The computational grid was refined in the boundary layer to provide the value of the near-wall parameter \(y + \approx 1\).

![Figure 1. Geometry of calculation area](image)

Calculations of the flow around the plate from a single capillary for three characteristic values of the longitudinal flow velocity at the channel inlet: 30, 60, and 120 m/s. The range of operating parameters of the acoustic drying installation, in which experiments will be carried out at the next stages of the
study, fits into such velocity distribution. The velocity flow in the working part of the installation for acoustoconvective drying, which is described in the second section of this report, of 30 m/s corresponds to an overpressure in the prechamber of 1 atm, a speed of 60 m/s corresponds to a pressure of 3 atm, and a speed of 120 m/s corresponds to pressure of 6 atm. Let's move on to discussing the simulation results. Fig. 2 shows the transverse velocity fields for the incoming flow velocities of 30 (a), 60 (b), and 120 (c) m/s. It can be seen from the figures that the maximum vertical speed at an incoming flow speed of 120 m/s is 8 m/s. In the upper part of the pore, at the interface with the main flow, a vortex flow is formed, which will contribute to the mass transfer of the liquid from the surface, when the capillary is filled with water. Moreover, for the maximum speed of the main flow, the vortex penetrates approximately 3 mm into the depth of the capillary.

Figure 2. Transverse velocity fields in the pore area for the main flow velocities 30 (a), 60 (b) and 120 (c) m/s.

Let us consider the problem of an acoustoconvective flow near a capillary. In contrast to the previous formulation of the problem, a periodic boundary condition for the static pressure was set on the left boundary of the region according to formula (1). In this calculation, the case of the most intense convective effect was considered, when the oncoming flow velocity was 120 m/s. In this case, the excess static pressure at the channel inlet was calculated with the following parameters in formula (1): $P_{op} = 370$ Pa, $A=10$ Pa, $f=100$ Hz. These parameters approximately correspond to the regime of weak acousto-convective interaction, at zero cavity depth and noise of the order of 130 dB. Fig. 3 shows the pressure distribution at the channel inlet at different times. It can be seen that the inlet pressure varies within the limits in accordance with the specified parameters in formula (9) with a frequency of 100 Hz.
Figure 3. Distribution of static pressure at the channel inlet at different points in time. Fig. 4 shows the pressure fields in the vicinity of the capillary during one cycle of the static pressure changes at times from 90 ms to 100 ms.

Figure 4. Pressure fields in the vicinity of the capillary during one cycle of change in static pressure.

3.2. Convective Drying

Let’s consider the problem of convective drying of a blind pore in a two-dimensional formulation based on the previously stated mathematical model. In this problem, a single pore with a width of 1 mm and a depth of 10 mm is considered, on which an air flow runs at 30 m/s. This problem does not take into account the phase transitions of the fluid in the flow. This is due to the low air temperature in the flow. The air temperature is 21°C, the water temperature is 19°C. The initial position of the liquid is shown in Fig. 5a.

During interaction of the incoming air flow, the water in the pore oscillates. As a result of this interaction, there is a gradual mass loss of the liquid. During the first second of interaction of the flow with the pore, a rather intensive extraction of liquid from water occurs (Fig. 5b–5d). The high intensity can be explained by the formation of a vortex structure in the upper part of the pore.
After the first second, liquid is slowly extracted from the pore. Fig. 6 shows the results of moisture extraction at time points 2 s (Fig. 6a) and 5 s (Fig. 6b). It can be seen that the change in the level of the liquid column is practically insignificant.

3.3. Evaluation of the effect of gas velocity on the liquid extraction process

The estimation of the influence of the gas flow rate in the channel on the process of fluid extraction from the pore is carried out. The gas velocity in the channel was taken as 30 m/s (Fig. 7a), 60 m/s (Fig. 7b), and 120 m/s (Fig. 7c). In fig. 7 shows the fields of the volumetric concentration of liquid at time 1 s for drops with a diameter of $10^{-5}$ m from a capillary for various velocities. It can be seen from the figure that with an increase in the gas velocity in the channel, the intensification of liquid extraction in the channel occurs, which is associated with the formation of a larger vortex structure inside the capillary. The size of the formed vortex in the channel (Fig. 2) is consistent with the volume of liquid, which is extracted by increasing the gas flow velocity.
Figure 7. The distribution of the liquid volumetric concentration fields at the time step 1s (a - c) at different gas flow rates in the channel. a) 30 m/s, b) 60 m/s, c) 120 m/s.

4. Conclusions
In this study a physical and mathematical model of liquid extraction under the influence of a convective gas flow. Calculations of the flow of gas and liquid in a single capillary on a plate, which simulate the surface porosity of a metal surface, have been performed. It is shown that in the calculation, the main mechanism of liquid extraction from the pore is the separation of droplets from the liquid surface.

5. Acknowledgment
The work was carried out within the framework of the state assignment of Ministry of Science and Higher Education of the Russian Federation No. 2019-0251 "Investigation of the processes of evaporation of liquids in the fuel tanks of launch vehicles to improve the environmental safety and economic efficiency of launch vehicles with liquid propellant rocket engines"

References
[1] Smirnov N N, Betelin V B, Nikitin V F, Phylippov Y G and Koo J 2014 Detonation engine fed by acetylene – oxygen mixture Acta Astronaut. 104 134–46
[2] Guendugov V M, Smirnov N N and Tyurenkova V V. 2013 Solving the problem of diffusion combustion of a droplet with allowance for several independent reactions Combust. Explos. Shock Waves 49 648–56
[3] Smirnov N N, Penyazkov O G, Sevrouk K L, Nikitin V F, Stamov L I and Tyurenkova V V. 2018 Onset of detonation in hydrogen-air mixtures due to shock wave reflection inside a
combustion chamber *Acta Astronaut.* **149** 77–92  
[4] Smirnov N N, Nikitin V F, Dushin V R, Filippov Y G, Nerchenko V A and Khadem J 2015 Combustion onset in non-uniform dispersed mixtures *Acta Astronaut.* **115** 94–101  
[5] Zhilin A A and Fedorov A V. 2016 Acoustoconvection Drying of Meat *J. Eng. Phys. Thermophys.* **89** 323–33  
[6] Fedorov A V. and Zhilin A A 2014 Mathematical modeling of moisture extraction from rice grains *J. Appl. Mech. Tech. Phys.* **55** 1016–9  
[7] Zhilin A A and Fedorov A V. 2014 Acousto-convective drying of pine nuts *J. Eng. Phys. Thermophys.* **87** 908–16  
[8] Fedorov A V., Zhilin A A and Korobeinikov Y G 2011 Investigation of the processes of impregnation and drying of granular silica gel *J. Eng. Phys. Thermophys.* **84** 965–74  
[9] Gosteev Y A, Korobeinikov Y G, Fedorov A V. and Fomin V M 2005 Heating of dry samples under an acoustic-convective action *J. Appl. Mech. Tech. Phys.* **46** 711–6  
[10] Tryshlyakov V I and Lavruk S A 2015 Theoretical and experimental investigations of the interaction of hot gases with liquid in closed volume *Acta Astronaut.* **109** 241–7  
[11] Fedorov A V., Bedarev I A, Lavruk S A, Trushlyakov V I and Kudentsov V Y 2018 Determination of the Flow Field in the Propellant Tank of a Rocket Engine on Completion of the Mission *J. Eng. Phys. Thermophys.* **91** 324–31  
[12] Hussein M M, Al-Sarkhi A, Badr H M and Habib M A 2019 *CFD modeling of liquid film reversal of two-phase flow in vertical pipes* vol 9 (Springer International Publishing)  
[13] Trushlyakov V I, Novikov A A, Lesnyak I Y and Panichkin A V. 2019 Study of acoustic and low-pressure exposure on the temperature of the evaporation of a liquid with free interface before it freezes *J. Acoust. Soc. Am.* **146** 3333–8  
[14] Trushlyakov V I, Lesnyak I Y, Novikov A A and Panichkin A V. 2018 Investigation of heat and mass transfer process in the closed volume with different types of impact on liquid *J. Phys. Conf. Ser.* **1050**  
[15] Trushlyakov V I, Novikov A A, Lesnyak I Y and Panichkin A V. 2019 Study of evaporation for liquid with free interface in the enclosed tank: acoustic and low-pressure exposure on the liquid *Thermophys. Aeromechanics* **26** 255–66  
[16] ANSYS Fluent Theory Guide 2013 *ANSYS Fluent Theory Guide* vol 15317