Hot melt splash upon impingement of a water jet

S E Yakush¹, N S Sivakov², V I Melikhov¹,³ and O I Melikhov¹,³

¹Institute for Problems in Mechanics of the Russian Academy of Sciences, Ave. Vernadskogo 101 Bldg 1, Moscow, 119526, Russia
²Bauman Moscow State Technical University, Baumanskaya 2-ya 5, Moscow, 105005, Russia
³National Research University Moscow Power Engineering Institute, Krasnokazarmennaya 14, Moscow 111250, Russia

E-mail: yakush@ipmnet.ru

Abstract. As a pre-requisite for powerful steam explosion following the spread of molten high-temperature material over the bottom of water pool, a premixed zone must be formed with certain contents of melt, water, and vapour. Experimental observations indicate that melt spreading is featured by the formation of periodic melt splashes which can be responsible for the phase premixing. The paper considers one of the key phenomena involved in this process, namely, the splash of hot melt due to impingement of a high-velocity water jet. Numerical simulations are performed in the axisymmetric formulation, revealing the whole process of water-melt interaction, as well the development of melt flow leading to its splash. Parametric study is presented for the melt splash height dependence on the water mass and velocity.

1. Introduction

Interaction between hot and cold liquids can proceed in different regimes, including rapid phase transition with conversion of significant fraction of thermal energy in the mechanical work, known as steam explosion [1]. This phenomenon is encountered where a high-temperature molten material is released into a coolant with much lower boiling temperature, for example, in underwater volcano eruptions, metallurgy and, most notably, in nuclear energy. In the latter area, so-called fuel-coolant interaction (FCI) is considered one of the major dangers in severe accidents with the reactor core degradation and meltdown.

The phenomenon of steam explosion has been studied extensively over the past few decades, however, due to its complexity there are still many remaining uncertainties to be resolved, both experimentally, theoretically, and numerically [1]. Recently, interest to studying the stratified steam explosions (with melt spreading under a shallow water layer) was renewed after it was shown experimentally [2] that quite powerful impact of the explosion wave on the pool bottom is possible: strong steam explosions with the peak pressure up to 10 MPa were registered in the experiments with molten binary oxidic materials having the temperatures above 1000 K.

The exact mechanisms for the formation of premixed zone above the spreading melt layer are not clear at the moment. In the experimental work [2] it was argued that periodic formation and collapse of vapor bubbles near the boundary between water and the vapor film separating it from the hot melt, visible on the experimental video recordings, can produce cumulative water micro-jets directed towards the melt surface. Interaction of these jets with the underlying liquid melt can result in the melt...
perturbations visible as sporadic splashes, with the melt reaching height of few centimeters. This physical mechanism, due to rapid boil-up of water upon its direct contact with the high-temperature melt, can be quite efficient in mixing the phases, so that necessary pre-requisites to steam explosion wave propagation appear. No mechanistic model for the formation of the premixing zone is available so far (see, e.g., numerical simulations [3] where ad-hoc parameters were assigned to the premixed zone).

It should be noted that vapor bubble growth on the water-vapor interface and its subsequent collapse require special study; these events appear to be similar to the well-known cavitation phenomenon, however, they bear significant differences due to the major role of thermal processes. While cavitation bubbles contain initially the saturated vapor at low pressure (saturation pressure at the water temperature), the bubbles formed during the spreading of melt are likely to contain vapor at the temperature well above the saturation point (due to contact of vapor with the spreading melt); also, no substantial initial pressure drop between the bubble vapor and ambient water is expected. However, the pressure drop is likely to appear due to vapor cooling and condensation, driving the water flow in the way similar to cavitation bubble collapse.

In the current work we concentrate on the interaction between the water micro-jet and the melt surface onto which the jet is impinging, leaving the solution of the coupled problem (including the bubble formation, collapse, and interaction in the three-layer configuration) for the future research. Here, we assume that a short-duration water jet is impinging vertically on a horizontal melt surface, and focus the study on the subsequent three-phase flow and melt splashing. Note that such jets can originate not only from the collapsing bubbles, but also from the development of Rayleigh-Taylor instability on the vapor-water surface, resulting in the downward flow of water through the vapor film.

2. Mathematical model

Consider a three-phase system consisting of melt (denoted by subscript \( m \)), liquid water (subscript \( w \)), and vapor (\( v \)) separated by sharp interfaces where interphase processes can proceed. To describe the flow, we apply the Volume Of Fluid (VOF) method (e.g., [4]), introducing an effective fluid with properties depending upon the volume fraction \( \alpha_k \) of each constituent fluid (\( k = m, w, v \)): the density \( \rho \), specific enthalpy \( h \), dynamic viscosity \( \eta \), and heat conductivity \( \lambda \) are expressed as

\[
\rho = \sum_{k=1}^{K} \alpha_k \rho_k, \quad \rho h = \sum_{k=1}^{K} \alpha_k \rho_k h_k, \quad \eta = \sum_{k=1}^{K} \alpha_k \eta_k, \quad \lambda = \sum_{k=1}^{K} \alpha_k \lambda_k,
\]

where the respective properties of the constituent physical fluids are denoted by subscript \( k \), \( K = 3 \) is the total number of fluids considered. The phase volume fractions satisfy the compatibility condition \( \alpha_m + \alpha_w + \alpha_v = 1 \). The flow is described by the conservation equations:

\[
\frac{\partial \rho \alpha_k}{\partial t} + \nabla (\rho \alpha_k U) = \Gamma_k,
\]

\[
\rho \left( \frac{\partial U}{\partial t} + (U \cdot \nabla) U \right) = -\nabla P + \nabla \tau + \rho g + F_i,
\]

\[
\rho \left( \frac{\partial h}{\partial t} + (U \cdot \nabla) h \right) = \nabla \lambda \nabla T.
\]

Here, \( t \) is the time, \( P \) and \( T \) are the pressure and temperature, \( U \) is the velocity vector, \( \Gamma_k \) is the phase source term due to phase transitions; in the current problem \( \Gamma_m = 0, \Gamma_w = \Gamma_v = \Gamma \), where \( \Gamma \) is the evaporation rate. The problem is closed by specifying the dependencies of each phase properties on the pressure and temperature from the respective equations of state. In this work we use a simple model of constant phase densities, heat capacities, and transport coefficients, neglecting the material compressibility and thermal expansion. While these assumptions are well-grounded for liquid melt and
water due to small variation of phase pressure and temperature in the short interaction time, they are more arbitrary for the vapor. However, due to the large difference in densities the ambient vapor plays a minor role in the problem, and the vapor produced by water evaporation is close to the saturation conditions. Therefore, we take the vapor density as that of the saturated vapor at the ambient pressure. The constant density assumption simplifies the numerical procedure, still retaining the effects of vapor expansion upon water evaporation; the effects of density variation will be studied in more detailed in the future solution of the fully coupled problem.

The surface tension force acting on \( k \)-th liquid phase \((k = m, w)\) is described by the continuous surface force (CSF) model (see, e.g. [4]):

\[
F_{x,k} = \sigma_k \kappa_k \nabla \alpha_k ,
\]

where \( \sigma_k \) is the surface tension of \( k \)-th liquid at the interface with vapor, \( \kappa_k = -\nabla \left( \nabla \alpha_k / \nabla \alpha_k \right) \) is the surface curvature of \( k \)-th liquid.

The phase change rate \( \Gamma \) on the water-vapor interface was approximated by a simple model by Lee [5] for the evaporation and condensation rate (simulations were performed with \( C = 100 \text{ s}^{-1} \)):

\[
\Gamma = C \alpha_w \rho_w \frac{(T - T_s)}{T_s}, \quad T > T_s; \quad \Gamma = -C \alpha_v \rho_v \frac{(T - T_v)}{T_v}, \quad T < T_v .
\]

Numerical simulations were carried out in the open source CFD toolbox OpenFOAM-v1912 [6], with icoReactingMultiphaseInterFoam solver suitable for flows of arbitrary number of immiscible phases separated by interfaces tracked by the VOF method.

3. Geometry and parameters

The problem geometry is sketched in figure 1. The melt pool is of depth \( H_m = 25 \text{ mm} \) and radius \( R = 140 \text{ mm} \); the space above it is filled with vapor, the height of computational domain is \( H = 200 \text{ mm} \). The left boundary of the computational domain coincides with the axis of symmetry, the top boundary is open, with the fixed pressure \( P = 1 \text{ bar} \); the right boundary corresponds to a solid wall up to the height \( H_m \) (pool depth), whereas above the pool level it is also open, with fixed pressure \( P = 1 \text{ bar} \). The initial temperatures of the melt and vapor are \( T_m = 1322 \text{ K} \) and \( T_v = 373 \text{ K} \) (saturated vapor), respectively. Water jet was modeled by a cylinder of radius \( r_w \) and height \( z_w \) with the center located at a height \( z_w \) above the melt surface; its initial vertical velocity was \( v_w \) (directed downwards). The initial water temperature was \( T_w = 300 \text{ K} \) (subcooled water).

\[\text{Figure 1. Sketch of the computational domain, with initial position of water jet above the melt layer. Arrow shows initial velocity of water jet, the dashed-dotted line demonstrates the definition of maximum splash height.}\]
The material properties used in the simulations were taken from [3]: the melt density is $\rho_m = 7811$ kg/m$^3$, heat capacity $c_m = 380$ J/(kg K), dynamic viscosity $\eta_m = 3.4 \cdot 10^{-3}$ Pa s, conductivity $\lambda_m = 2$ W/m K, surface tension $\sigma_m = 0.4$ N/m; the corresponding water properties are $\rho_w = 1000$ kg/m$^3$, $c_w = 4181$ J/(kg K), $\eta_w = 8.5 \cdot 10^{-4}$ Pa s, $\lambda_w = 0.61$ W/m K, $\sigma_w = 0.06$ N/m; vapor properties are $\rho_v = 0.59$ kg/m$^3$, $c_p,v = 2100$ J/(kg K), $\eta_v = 1.8 \cdot 10^{-5}$ Pa s, $\lambda_v = 0.025$ W/m K; the heat of evaporation is $\Delta H_v = 2.26$ MJ/kg. Simulations were performed on a grid with 210×150 cells in the radial and vertical directions, refined in the melt region and neat the axis where the most significant interaction occurs.

4. Results

The flow in question is in many respects is similar to the classical problem of droplet splashing on a water surface (see e.g., [7]); the differences, though, are related to the large density ratio (about 7.8) not encountered for “ordinary” liquids, and also by the melt temperature being well above the boiling temperature of water, making co-existence of the hot and cold liquids thermodynamically impossible (although, non-equilibrium processes are expected upon direct contact of the two fluids, followed by rapid boil-up of water). While the evaporation model (5) is too simplistic to capture the complex physics of such non-equilibrium processes, it gives reasonable physical behavior (rapid evaporation of water heated above the saturation), provided that the constant $C$ is high enough.

Consider first the short phase of water-melt interaction starting with the first contact of water jet and melt. The baseline simulation was carried out for the following parameters: initial jet radius $r_0 = 2$ mm, length $z_0 = 15$ mm, height above the melt surface $z_v = 5$ mm. In figure 2, four instants are shown; for clarity, each liquid phase (water, melt) is presented by a single contour line where the respective volume fraction is $\alpha = 0.5$.

Figure 2. Interaction of melt with impinging water jet at times $t = 0.15, 0.3, 1$, and 2 ms.
It can be seen that water jet impact on the melt surface causes the development of a cavern, as is observed in the splashes of immiscible liquids of close densities. Also clear is the development of intensive vapor flow along a rather thin melt sheet thrown upwards and sideways by the impact. Figure 2 shows that liquid water is vanishing with time due to intensive evaporation, and there exists a thin vapor film separating water from melt. Water evaporation proceeds most intensively over the first 4 ms, by which time 90% of the original mass is converted to vapor.

After the liquid water has been evaporated, further evolution of melt is completely governed by the momentum gained during the impact interaction, gravity, and surface tension on the melt free surface. The thin high-velocity melt sheet is rising to form the known “crown”; in the 2D framework taken here, though, the shape of melt splash remains axisymmetric.

Further evolution of the melt is shown in figure 3 where four instants are presented demonstrating the melt rise to its highest point at time $t = 140$ ms and subsequent slumping due to gravity. It can be seen that the splashed melt sheet loses its integrity and looks like a swarm of droplets (although real droplets can only be reproduced in 3D simulations planned in the future work). It is these melt droplets that are thrown up by the impact to mix with water and form the premixing zone prior to steam explosion.

Subsequent gravity-driven flow of melt filling the cavity near the axis of symmetry, the converging flow results in the formation of a cumulative jet [8] seen in figure 4 as the secondary melt splash. The height of the secondary melt splash is smaller than that of the primary one, at least for the parameters of the simulation.

![Figure 3. Melt splash at times $t = 18, 110, 140, 188$ ms (left to right).](image)

![Figure 4. Secondary melt splash due to formation of cumulative jet, $t = 220, 250, 294, 380$ ms (left to right).](image)

Parametric study of melt splash occurring upon interaction with an impinging water jet was performed by varying the initial parameters of the problem, namely, the initial jet velocity $v_w$, jet height $z_0$, and radius $r_0$; the initial height of the jet above the melt surface was constant, $z_w = 5$ mm. In all the cases, the general features of the interaction and subsequent melt flow were qualitatively the same as in the baseline case presented in figures 2–4. The quantitative characteristics obtained in the simulations are summarized in table 1.
Table 1. Melt splash characteristics.

| Case | Jet velocity $v_w$ (m/s) | Jet length $z_0$ (mm) | Jet radius $r_0$ (mm) | $\tau_1$ (ms) | $\Delta z_1$ (mm) | $\tau_2$ (ms) | $\Delta z_2$ (mm) | $U_{av}$ (m/s) |
|------|------------------------|----------------------|----------------------|--------------|-----------------|--------------|-----------------|----------------|
| 1    | 50                     | 15                   | 2                    | 108          | 79              | 294          | 30              | 0.19           |
| 2    | 25                     | 15                   | 2                    | 62           | 53              | 254          | 16              | 0.18           |
| 3    | 80                     | 7.5                  | 2                    | 80           | 81              | 270          | 26              | 0.22           |
| 4    | 50                     | 7.5                  | 2                    | 62           | 61              | 248          | 19              | 0.16           |
| 5    | 50                     | 15                   | 1                    | 68           | 72              | 246          | 24              | 0.17           |

Here, $\tau_1$ is the first splash time, $\Delta z_1$ is the melt rise height for the first splash, $\tau_2$ and $\Delta z_2$ are the respective quantities for the second splash, $U_{av}$ is the maximum melt velocity averaged on the time interval from the initial instant to the first splash.

The data presented in table 1 indicate that, within the water jet parameter range studied, melt splashes can reach the height of few centimeters. The primary splashes are more pronounced than the secondary ones; the splash heights increase with the increase in the jet velocity (compare Cases 1 and 2, 3 and 4). Interestingly, the splash height depends weakly on the jet length (compare Cases 1 and 4), indicating the predominant role played by the water impact on the first encounter with the melt surface.

5. Conclusions
The numerical simulation results show that splashing of hot and heavy melt due to impingement of short-duration water jet is possible, despite the large density ratio of the two liquids. This result supports the idea that melt perturbations by cumulative jets from collapsing bubbles can be responsible for the melt splashes visible experimentally. Further studies are necessary to confirm this finding, they will be devoted to coupled simulation of water-vapor-melt interaction in the stratified configuration where the impinging jet parameters will be related to the processes on the water-vapor interface (bubble formation and collapse).

Acknowledgments
This research was funded by Russian Science Foundation (RSF) under Grant 18-19-00289.

References
[1] Berthoud G 2000 *Annu. Rev. Fluid Mech.* 32 573–611
[2] Meignen R at al. 2014 *Ann. Nucl. Energy* 74 125–33
[3] Kudinov P, Grishchenko D, Konovalenko A and Karbojian A 2017 *Nucl. Eng. Des.* 314 182–97
[4] Leskovar M, Centrih V and Uršič M 2016 *Nucl. Eng. Des.* 296 19–29
[5] Tryggvason G, Scardovelli R and Zaleski S 2011 *Direct Numerical Simulations of Gas-Liquid Multiphase Flow* (Cambridge: Cambridge University Press)
[6] Lee W H 1979 *A Pressure Iteration Scheme for Two-Phase Modeling* (Technical Report LA-UR 79-975, Los Alamos Scientific Laboratory, Los Alamos, New Mexico)
[7] OpenFOAM. The open source CFD toolbox 2019 URL https://www.openfoam.com
[8] Il’inykh A Y and Chashechkin Y D 2020 *Fluid Dynamics* 55 (2) 162–70
[9] Lavrentiev M A and Shabat B V 1977 *The Problems of Fluid Dynamics and their Mathematical Models* (Moscow: Nauka) (in Russian)