

Features of shock-wave and vortex processes simulation at depressurization of circuits with superheated water coolant

M V Alekseev¹, M E Kuznetsova, S I Lezhnin, N A Pribaturin

Kutateladze Institute of Thermophysics SB RAS, Ave. Ak. Lavrenteva 1, 630090, Novosibirsk, Russia

E-mail: ¹alekseev@itp.nsc.ru

Abstract. Formation and evolution of pressure waves on the barrier were numerically simulated at non-stationary outflow of the superheated vapor from the vessel. The simulation conditions were fully consistent with the experimental conditions (1982) [1]. The calculated pressure fields were calculated both at the initial stage of non-stationary vapor outflow and at the quasistationary flow of superheated vapor on the barrier. The dynamics of interaction between shock waves and vortex structures on the barrier was studied at quasi-stationary outflow of the superheated vapor, and the experimental pressure profile on the target was compared with the calculated pressure profile. Comparison of the experimental data and the calculation results has showed acceptable agreement between the pressure profiles on the target up to a distance from the target to the nozzle approximately $X/D = 6$.

1. Introduction

The problem of depressurization of vessels or pipe lines with heated water coolant was formulated in the 60-80 years of the last century (see, e.g. [2, 3, 4]). This problem is related to prediction of emergency operation modes of power equipment in case of emergency depressurization. At that thermodynamic equilibrium and nonequilibrium models based on the hypothesis of heterogeneous nucleation were used to calculate the liquid water coolant boiling, caused by depressurization of high pressure vessel [5]. Accounting for possible bubble fragmentation due to the interfacial instability made the models too complicated [6]. Another approach to solving this problem is the use of non-equilibrium two-phase model, based on the experimental data for the relaxation time (transition) "non-equilibrium - equilibrium boiling" [7]. In [8] it was shown that the non-stationary relaxation model of boiling adequately describes the nonequilibrium boiling in subsonic and supersonic flows of boiling liquid in the channels. On the basis of the relaxation model and using the toolkit LCPFCT [9] the authors developed a software code for solving a wide class of problems [10, 11, 12] on depressurization of vessels or pipelines with water coolant. Averaging of the Euler equations [13] on the unit cell and the time interval is equivalent to the model of large-scale eddies with "subgrid turbulence model. The "subgrid turbulence model occurs implicitly, since the scale of large eddies is limited to the computational grid size. I.e., the scales of vortices for which direct modeling is impossible are calculated using a numerical viscosity.



The aim of this work is to test the program code for the problems of depressurizing vessels and channels with superheated vapor. As a test problem we selected the experimental work [1].

2. Calculation method

The calculation was performed using homogeneous model. At that the parameters of the vapor phase corresponded to saturation, and the liquid phase could be both metastable and equilibrium. The system of equations of the model includes the Euler equations in the axisymmetric approximation:

$$\frac{\partial}{\partial t}(\rho r) + \frac{\partial}{\partial z}(\rho r u) + \frac{\partial}{\partial r}(\rho r v) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\rho r u) + \frac{\partial}{\partial z}(\rho r u^2) + \frac{\partial}{\partial r}(\rho r u v) = -r \frac{\partial p}{\partial z} \quad (2)$$

$$\frac{\partial}{\partial t}(\rho r v) + \frac{\partial}{\partial z}(\rho r u v) + \frac{\partial}{\partial r}(\rho r v^2) = -r \frac{\partial p}{\partial r} \quad (3)$$

$$\frac{\partial}{\partial t}(E r) + \frac{\partial}{\partial z}(u r (E + p)) + \frac{\partial}{\partial r}(v r (E + p)) = 0 \quad (4)$$

supplemented by the equation of the mass source associated with the phase transition:

$$\frac{\partial(\rho X_\Gamma)}{\partial t} + \frac{\partial(\rho u X_\Gamma)}{\partial z} + \frac{1}{r} \frac{\partial(\rho r v X_\Gamma)}{\partial r} = \Gamma \quad (5)$$

$$\Gamma = \frac{\rho(X_{SAT} - X_\Gamma)}{\theta} \quad (6)$$

Here z, r are the axial and radial coordinates, u, v are the axial and radial components of velocity, ρ is the density, P is the pressure, E is the specific internal energy, X_Γ is the mass, and X_{sat} is the equilibrium mass content of vapor (at given P and E), Γ is the rate of vapor generation, and θ is the relaxation time [7].

The density of homogenous two-phase mixture is expressed through the density of the phases and the mass vapor content:

$$\rho = \left(\frac{X_\Gamma}{\rho_{2,SAT}(P)} + \frac{1 - X_\Gamma}{\rho_{1,M}(P, W_{1,M})} \right)^{-1} \quad (7)$$

and the specific enthalpy is determined by the following ratio:

$$W = X_\Gamma W_{2,SAT}(P) + (1 - X_\Gamma) W_{1,M} \quad (8)$$

To describe of phase transition within the homogeneous two-temperature approximation of a vapor-liquid medium used phenomenological relaxation model (5)-(6), based on processing the experimental data for the time of relaxation (transition) "non-equilibrium - equilibrium boiling" [7]. Calculation of vapor and liquid properties (7)-(8) was carried out using the software package TTSE [14]. To solve the system of equations (1-4) the authors applied software package LCPFCT [9], which used a differential monotone conservative method of flow correction (Flux Corrected Transport). The goal of this study is to investigate the pressure field evolution during the transition from the non-stationary to quasi-stationary outflow of the coolant. In this case, it is required to calculate unsteady motion of ordered and large-scale turbulent structures. The scale of large eddies is limited to the computational grid size in the direct simulation. The scales of vortices for which direct simulation is impossible are modeled as a subgrid turbulence using the

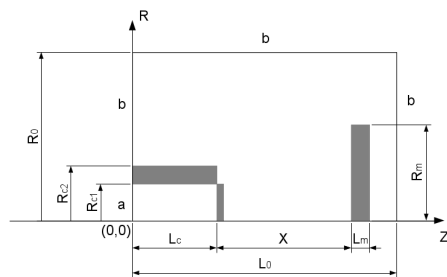


Figure 1. Calculation domain.

numerical viscosity. At finite difference approximation of Euler equations the viscosity terms of an order of $O(\Delta t, h^2)$ arise; at that the numerical viscosity $\varepsilon \approx |V|h$ allows modeling the subgrid turbulence viscosity.

The characteristic calculation domain is shown in figure 1. The calculation was carried out in the cylindrical coordinate system with a uniform grid. The radius of the computational domain was equal to R_0 , and the length of the computational domain was L_0 . The heated water coolant was filled in the pipe channel with an inner radius R_{c1} and the outer radius R_{c2} . The channel length was equal to L_{c1} . From the right end the pipe channel was closed by a diaphragm. From the left side a so-called "the Cauchy-Lagrange boundary condition" was set for the inflow in the channel [4]. The point of this "input section model" following. When the fluid flowing out from the vessel during the input section essentially not one-dimensional. The most significant feature is the inertia of the flow of fluid flowing into the tube, which affects the flow rate and pressure waves structure. It is proposed [4] on the approximation of the uniformly of the flow at the inlet and the potential flow to build analytic field and the velocity of the medium using the integral of the Cauchy - Lagrange, an equation for the fluid velocity in the inlet section to the outlet:

$$\frac{4D}{3\pi} \frac{\partial v}{\partial t} + \frac{v^2}{2} + \frac{p - p_0}{\rho_0},$$

where of D - the pipe diameter ; p_0 , ρ_0 is the pressure and density of the medium in the vessel; p , v - pressure and velocity in the inlet section. As shown in [4] the proposed "input section model" should adequately describe the outflow of cooled liquid or vapor-liquid mixture at small pressure drops and high ones.

At the distance X the axisymmetric target in the form of disc with L_m thickness and R_m radius was set. On the boundaries of the computational domain the condition b of free inflow and outflow at pressure P_0 was used.

To select the optimal parameters of the external liquid-vapor medium at a temperature of 100°C its density and sound velocity were compared with the density and sound velocity of standard air atmosphere at a temperature of 20°C . The sound velocity in air was compared with the "frozen" sound velocity in liquid-vapor medium, which is more adequate for fast wave processes. The analysis showed [14, 11, 12], that the mass vapor content of the external vapor-liquid medium should be chosen equal to 0.5.

The initial conditions and the geometry of the numerical model corresponded to the experimental conditions [1]. Superheated vapor pressure at the entrance to the channel $P_0 = 4.02$ MPa, $T_0 = 258.24^\circ\text{C}$, the inner radius of the circular channel $R_{c1} = 5$ mm, the outer radius $R_{c2} = 9$ mm, and the channel length was equal to $L_{c1} = 100$ mm. Diameter meseni was $20D$.

3. Shock-wave and vortex effects in unsteady outflow of superheated vapor on the barrier

Figure 2 shows the dynamics of formation of shock-wave and vortex structures in non-stationary outflow of superheated vapor (field of normalized pressure gradient) when displacing the target

to a distance of $X/D = 6.3$. It may be noticed that for the first $200 \mu s$ (figure 2(a-c)) of the process there is generation of compression waves that moves toward the target and is reflected from it. After the compression wave the structure of the Mach disk with barrel shocks begins to form. At the barrel shock due to the Kelvin — Helmholtz instability occurs the generation of vortex structures that interact with the primary compression wave and form the complex structure of the pressure fields ($t = 200 - 600 \mu s$) (figure 2(d-g)). After ($t = 600 \mu s$) the quasi-stationary jet flow occurs, and periodic vortex structures affecting the pressure field are generated.

Generation of periodic structures in the quasi-stationary outflow essentially depends on the distance between the target and the nozzle (figure 3). At a small distance $X/D = 0.5$ (figure 3.a) a narrow zone of supersonic flow is formed in the form of "disc" on the target and transits in a subsonic jet flow along the target. The pulsation of this boundary leads to the formation of periodic pressure waves along the target. When the distance $X/D = 3.3$ (figure 3.b) the formation of the Mach disk and the side barrel shocks occurs.

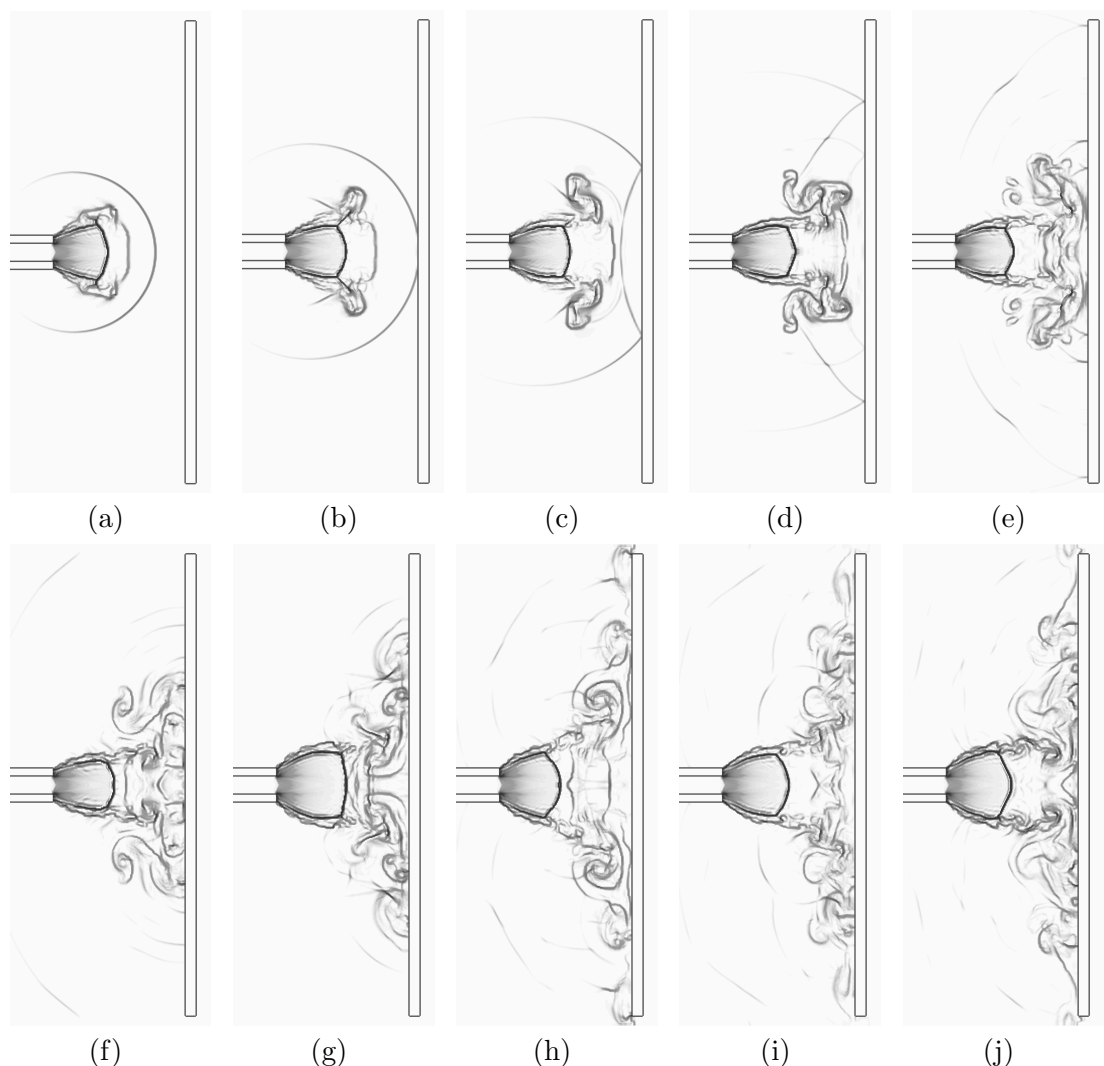


Figure 2. Normalized pressure gradient at various times for a fixed computational domain geometry, $P_0 = 4.02 \text{ MPa}$; $D = 10 \text{ mm}$; $X/D = 6.3$; time of calculation t : a - $100 \mu s$, b - $140 \mu s$, c - $180 \mu s$, d - $250 \mu s$, e - $350 \mu s$, f - $450 \mu s$, g - $600 \mu s$, h - $800 \mu s$, i - $1000 \mu s$, j - $1130 \mu s$

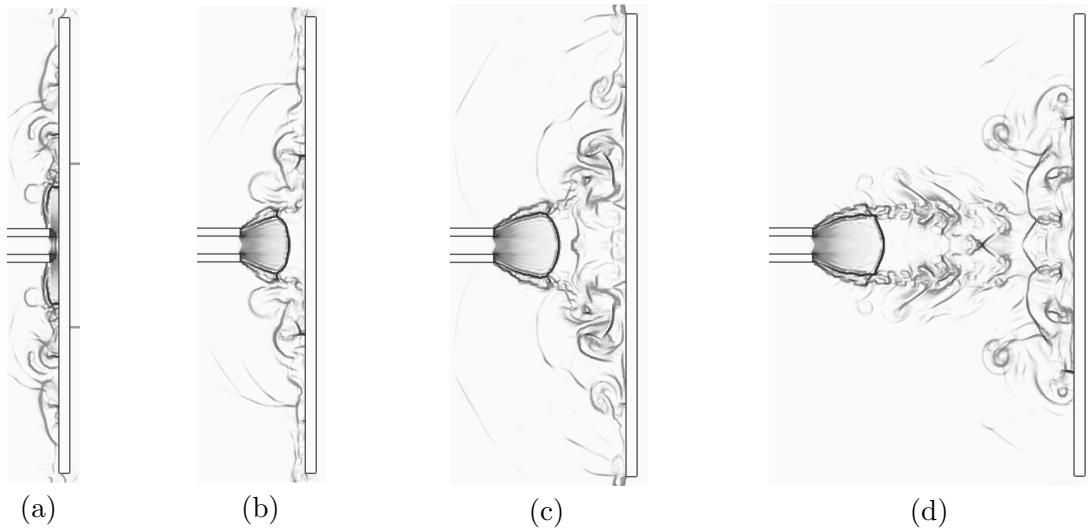


Figure 3. Normalized pressure gradient at the time of $850 \mu s$ for different computational domain geometry $P_0 = 4.02 \text{ MPa}$; $D = 10 \text{ mm}$; $t = 850 \mu s$; distance from the target to the nozzle: a - $X/D = 0.5$; b - $X/D = 3.3$; c - $X/D = 6.3$; d - $X/D = 13$.

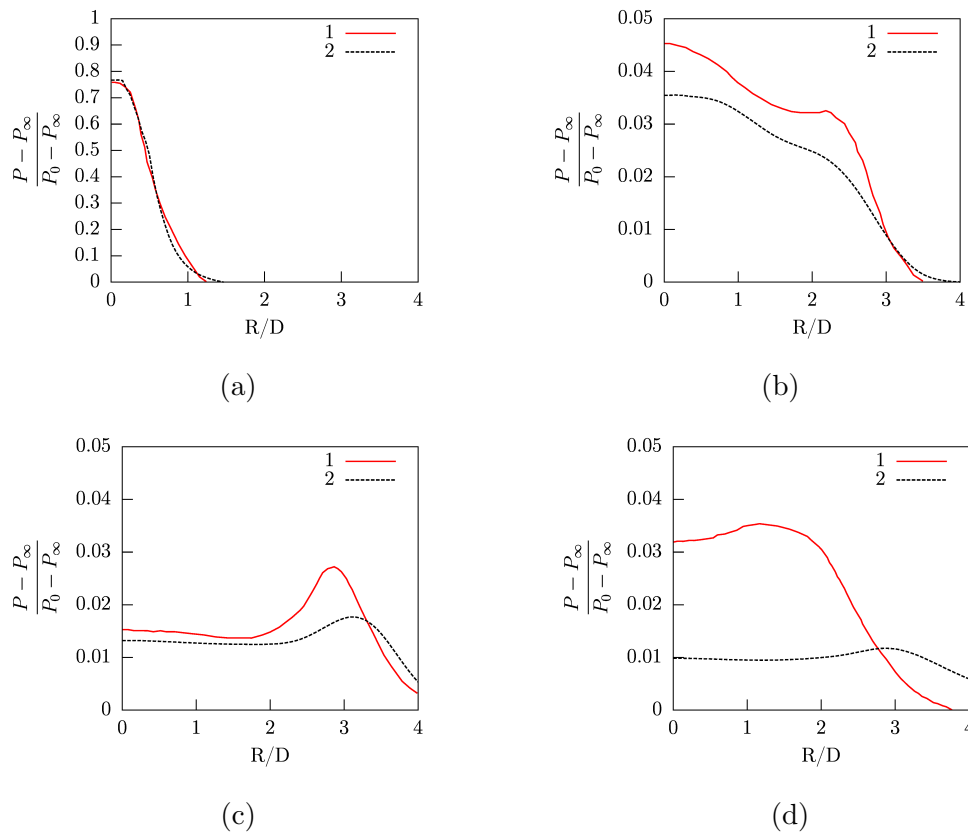


Figure 4. The profile of the dimensionless pressure on the target depending on the radius in calibers at the superheated vapor jet outflow: 1-experiment [1], 2-calculation; $P_0 = 4.02 \text{ MPa}$; $D = 10 \text{ mm}$; the distance of the target from the nozzle: a - $X/D = 0.5$; b - $X/D = 3.3$; c - $X/D = 6.3$; d - $X/D = 13$.

In addition to the formation of periodic pressure waves due to pulsation of the structure of the barrel shocks, the vortical structures are generated due to the Kelvin — Helmholtz instability on the side faces of the barrel shock structure. For a distance $X/D = 6.3$ (figure 3.c) the main contribution to the pressure pulsations is given by Kelvin — Helmholtz instability. When the distance $X/D = 13$ (figure 3.d) previous mechanisms of periodic structure generation are added with extra formation and collapse of off-design structures of pressure surges after the Mach disk and the target.

Figure 4 shows a comparison of the profiles of dimensionless pressure on the target depending on the radius in calibers, where 1 is the experiment [1], and 2 is the calculation. The calculation presents the pressure profile, obtained by averaging the non-stationary pressure profile over time of $100\mu s$, (the time of calculation of quasi-stationary process was $100ms$). The experimental pressure profile coincides with the calculated pressure profile if distance from the target is equal to $X/D = 0.5$ (figure 4.a). With increasing distance from the target to the nozzle to $X/D = 6.3$ the calculated pressure profiles on the target acceptably (but rather qualitative) repeat the profiles, obtained in the experiment (figure 4 (b,c)). When the distance between the calculated and experimental pressure profile on the target $X/D = 13$, the difference becomes very significant. This fact is connected with the methodological errors: a) the error resulting from the approximation of viscous effects in the numerical scheme; b) the error of modeling in "air atmosphere"; c) the absence of a boundary layer near the target. The strong divergence between the calculated and experimental data at a large distance from the nozzle due to the accumulation of errors mentioned above.

Acknowledgments

Authors express their gratitude to A.L. Sorokin for active participation in the development and numerical realization of the models.

The work was financially supported by the Russian Science Foundation (grant No14-29-00093).

References

- [1] Masuda F and et al 1982 *Nuclear Engineering and Design* **67**(2) 273 – 286
- [2] Edwards A R and O'brien T P 1970 *J. Br. Nucl. Energy Soc.* **9** 125–135
- [3] Nigmatulin B I and Soplenkov K I 1980 *High Temperature* **18**(1) 118 – 131 (in Russian)
- [4] Goffman G V, Kroshilin A E and Nigmatulin B I 1981 *High Temperature* **19**(6) 1240 – 1250
- [5] Bolotnova R K and Buzina V A 2014 *Computational continuum mechanics* **7**(4) 343 – 352
- [6] Ivashnev O E 2008 *Fluid Dynamics* **43**(3) 390 – 401
- [7] Zapolski P D, Bilicky Z, Bolle L and Franco J 1996 *Int. J. Multiphase Flow* **22**(3) 473 – 483
- [8] Artemov V, Minko K and Yan'kov G G 1996 *Thermal Engineering* **62**(12) 897 – 905
- [9] Boris J P, Landsberg A M, Oran E S and Garder J H LCPFCT - Flux-Corrected Transport Algorithm for Solving Generalized Continuity Equations. NRL/MR/6410-93-7192
- [10] Alekseev M V, Lezhnin S I, Pribaturin N A and Sorokin A L 2014 *Thermophysics and Aeromechanics* **21**(6) 763–766
- [11] Alekseev M V, Lezhnin S I and Pribaturin N A 2015 *Vestnik of TyumSU. Physical and mathematical modeling.* **1**(2 (2)) 75– 84 (in Russian)
- [12] Alekseev M V and Vozshakov I S 2015 *Vestnik of TyumSU. Physical and mathematical modeling.* **1**(4(4)) 6 – 14 (in Russian)
- [13] Belotserkovskii O M 1996 *U.S.S.R. Comput. Math. Math. Phys.* **22**(6) 166–183
- [14] Lezhnin S I and Pribaturin N A 1983 *Proceedings of Siberian branch of AS USSR. series "Technical sciences"* (2) 20 – 26 (in Russian)