

Analysis of Condensation Characteristics of Moist Air in a Laval Nozzle

Chunwei Gong, Jianyong Wang *

College of Mechanical and Electrical Engineering, Shaanxi University of Science and Technology, Xi'an Shaanxi, 710021, China

* **Corresponding author:** Jianyong Wang (Email: jywang@sust.edu.cn)

Abstract: This study investigates the non-equilibrium condensation characteristics of moist air in a Laval nozzle using CFD methods coupled with a nucleation model and a droplet growth model. The results indicate that both the nucleation rate and the droplet growth rate are influenced by the supersaturation ratio. The nucleation rate exhibits a peak only within a short region near the nozzle throat, while the distribution of the growth rate is similar to that of the supersaturation ratio. In addition, it is found that the presence of shock waves leads to an increase in temperature and a reduction in supersaturation, thereby affecting the droplet growth rate.

Keywords: Laval Nozzle; Multiphase Flow; Condensation; Nucleation Model.

1. Introduction

In high-speed compressible flows, non-equilibrium phase-change phenomena often occur due to fluid expansion, cooling, and rapid pressure variations. Among these, condensation inside nozzles is a typical problem of great theoretical significance and engineering impact. This phenomenon is widely encountered in steam nozzles, rocket and aero-engine nozzles, supersonic wind tunnels, and organic Rankine cycle systems, and it can significantly affect flow structures, thermodynamic performance, and equipment safety.

When superheated steam or condensable gases accelerate and expand within a nozzle, the temperature may rapidly drop below the saturation temperature, resulting in a supercooled state. Once certain conditions are satisfied, spontaneous nucleation occurs and condensation develops rapidly, releasing latent heat. This process leads to abrupt changes in the temperature, pressure, and Mach number distributions in the flow field. Such condensation processes are usually accompanied by variations in shock-wave structures, efficiency losses, and non-equilibrium thermodynamic effects, and are therefore referred to as non-equilibrium condensation flows.

Wegener et al. [1] experimentally investigated the condensation process of water vapor in nozzles. By analyzing the thermodynamic processes of the gaseous phase inside the nozzle, they inferred the latent heat released during vapor condensation and thereby estimated the amount of condensate. By comparing experimental results with first-principles calculations, they found that first-principles theory was unable to quantitatively predict the occurrence of condensation, although qualitative agreement between theory and experiment was observed. They attributed the quantitative discrepancies to the insufficient understanding of material properties at low temperatures at that time. Cheng et al. [2] numerically studied the flow disturbances generated by the sudden injection of water vapor at the inlet of a nitrogen nozzle. They found that condensation occurred downstream of the nozzle throat, initially triggering shock waves propagating both upstream and downstream, and forming an expansion region downstream of the condensation front.

Subsequently, flow unsteadiness intensified, the expansion region disappeared, and the strengths of both upstream and downstream shock waves were significantly weakened and appeared repeatedly. Wiśniewski et al. [3] employed homogeneous and heterogeneous multiphase flow models to simulate phase-change processes under supersonic flow conditions, concluding that condensation effects cannot be neglected in transonic and supersonic flow regimes. Xiao et al. [4] conducted CFD simulations of the condensation process of moist natural gas in a Laval nozzle using nucleation and droplet growth theories, and found that spontaneous condensation of water vapor does not occur immediately upon reaching saturation, but instead begins downstream after the flow enters a supersaturated state.

For nozzle condensation problems, extensive studies have been carried out from multiple perspectives, including theoretical analysis, numerical simulation, and experimental investigation. Classical research has primarily been based on classical nucleation theory and droplet growth models to predict the onset location of condensation, nucleation rates, and droplet size distributions in nozzles. With the development of computational fluid dynamics (CFD), an increasing number of studies have coupled non-equilibrium condensation models with the compressible Navier–Stokes equations to simulate condensation flows under complex nozzle geometries and operating conditions. However, because the condensation process involves multi-scale and multi-physics coupling, its accurate description still faces significant challenges.

Therefore, a systematic investigation of the onset conditions, evolution characteristics, and effects of nozzle condensation on flow behavior is of great significance for improving non-equilibrium phase-change flow theory and enhancing the design and operation of related engineering systems.

2. Numerical Model

2.1. Geometric Model

The geometric model used in this study is shown in Fig. 1. Since the Laval nozzle is axisymmetric, the computational domain is simplified to a two-dimensional model. An

extended region is modeled at the nozzle outlet to simulate the actual atmospheric environment. The gas inlet is located at the left end of the nozzle, while the outlet consists of all the walls of the rectangular extended region.



Fig 1. Geometric model

The mesh is generated using Meshing software. A multi-zone approach is adopted to produce a high-quality mesh, and the nozzle region is locally refined to more accurately capture the flow behavior variations induced by changes in nozzle geometry. The mesh schematic is shown in Fig. 2.

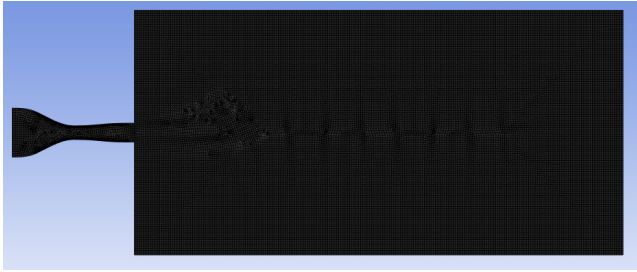


Fig 2. Nozzle mesh

2.2. Mathematical Model

In this study, a classical nucleation model and a droplet growth model are employed to describe droplet formation and growth during the condensation process.

The nucleation model is expressed in Eq. (1). The nucleation rate is jointly affected by the molecular collision term and the thermophysical properties of the condensable component. Among these factors, the gas-phase molecular number density, supersaturation ratio, surface tension, and critical radius play dominant roles.

$$J = \frac{vn^2}{\Delta T_s} \sqrt{\frac{2\sigma}{\pi m}} \exp\left(\frac{-4\pi\sigma r_{cr}^2}{3kT_g}\right) \quad (1)$$

The droplet growth model is given in Eq. (2). The droplet growth rate is mainly influenced by the supersaturation ratio and the ratio of the droplet radius to the critical radius.

$$\frac{dr}{dt} = \frac{P}{h_{1v}\rho_1\sqrt{2\pi RT_v}} \frac{\gamma+1}{2\gamma} c_p \left(1 - \frac{r_{cr}}{r}\right) (T_{sat} - T) \quad (2)$$

The calculation of the phase-change variable is given in Eq. (3). This variable is jointly contributed by two processes: droplet formation and droplet growth.

$$M = J\rho_1 \frac{4}{3}\pi r_{cr}^3 + n\rho_1 4\pi r^2 \frac{dr}{dt} \quad (3)$$

2.3. Boundary Conditions

In this study, the inlet pressure of the nozzle is 300 kPa, and the outlet back pressure is set to one atmosphere. The working fluid at the nozzle inlet is a mixture of air and water vapor, where the water-vapor mass fraction corresponds to saturation at the specified inlet pressure and temperature.

Except for the nozzle inlet and outlet, all other boundaries are treated as adiabatic walls. The numerical simulations are performed using ANSYS Fluent.

3. Results and Discussion

In this study, the condensation process of moist air inside the nozzle is numerically simulated. The results are analyzed from two aspects: contour distributions and quantitative data.

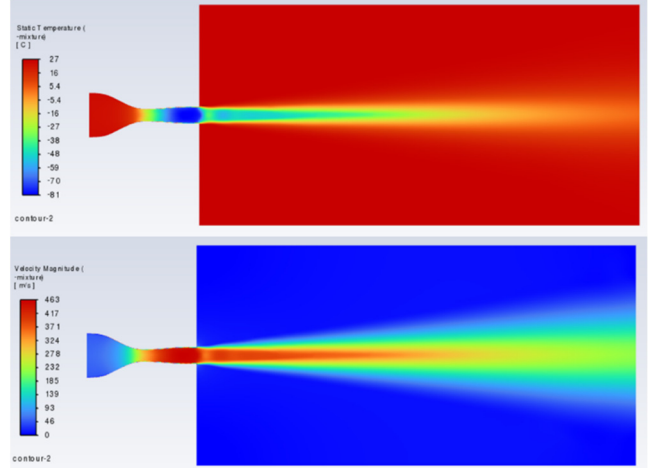


Fig 3. Contours of thermodynamic temperature and velocity in the nozzle (top: temperature; bottom: velocity)

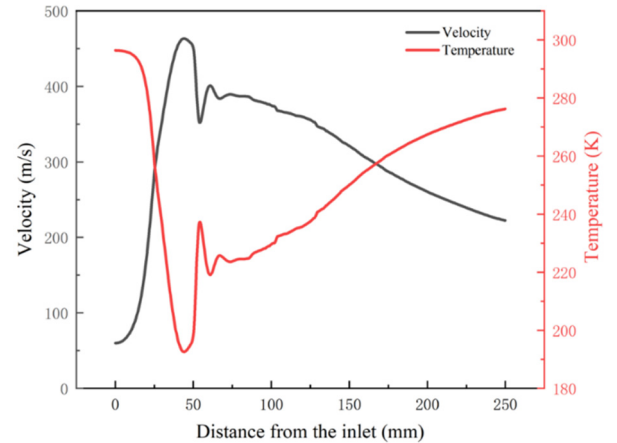


Fig 4. Variations of velocity and temperature along the nozzle centerline with distance from the inlet

As shown in Figs. 3 and 4, in the divergent section of the nozzle, the velocity of the moist air increases while the temperature decreases. A pronounced shock wave can be observed near the end of the nozzle divergent section, where the velocity drops sharply and the temperature rises, which may have a negative impact on the condensation process. After passing through the shock-wave region, the temperature gradually increases and the velocity gradually decreases.

Figure 5 presents the key condensation parameters in the nozzle. As shown in Fig. 5, regions with high supersaturation are mainly located in the nozzle divergent section and part of the extended wake region. In contrast, the nucleation rate is high only in a region slightly downstream of the nozzle throat, while it is zero elsewhere. The distribution of the droplet growth rate is similar to that of the supersaturation ratio, exhibiting relatively high values in the divergent section and within a limited wake region. The large discrepancy between the distributions of nucleation rate and supersaturation is presumed to be due to the excessively low thermodynamic

temperature in the divergent and wake regions, which reduces the surface tension of liquid water and thereby results in a low

nucleation rate.

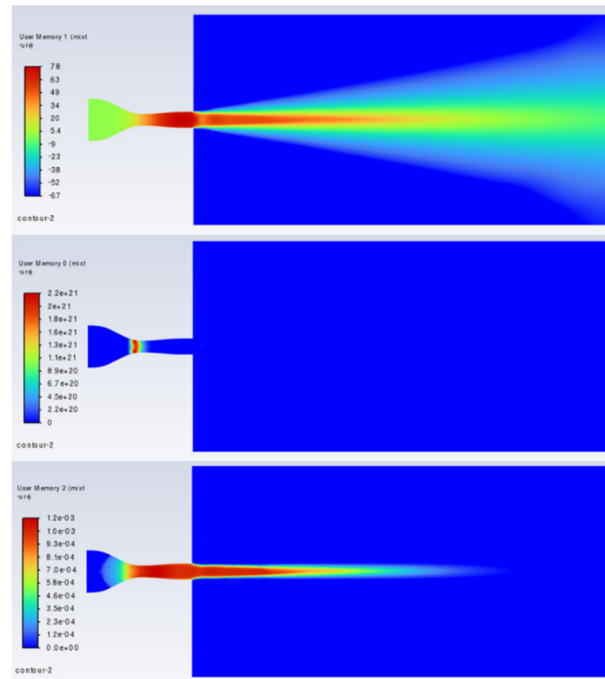


Fig 5. Key condensation parameters in the nozzle, from top to bottom: supersaturation ratio, nucleation rate, and droplet growth rate

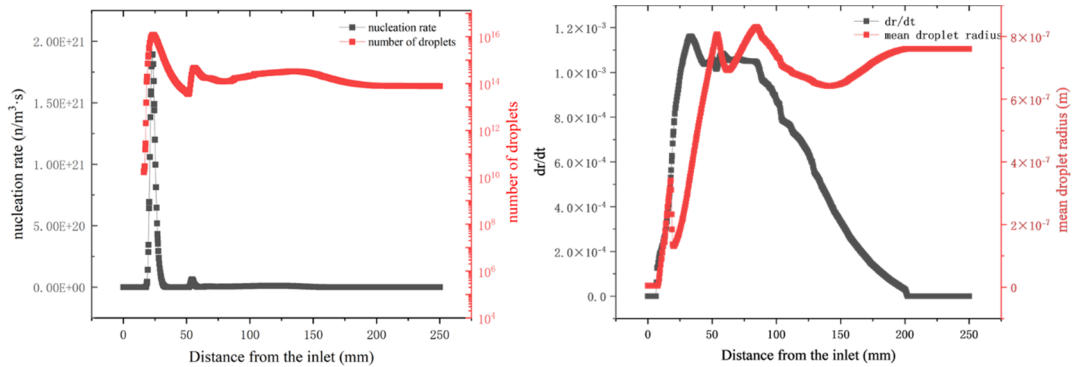


Fig 6. Nucleation and growth curves along the nozzle centerline with distance from the inlet

As shown in Fig. 6, the nucleation rate inside the nozzle exhibits only brief peaks in the throat region and at the location of the subsequent shock wave. The number of droplets also increases rapidly in this region and then tends to stabilize. In the wake region, the droplet number shows a limited decreasing trend due to dilution caused by the increasing wake width. In contrast, the region covered by a high growth rate is much larger: the growth rate remains relatively high in both the divergent section and the wake region. Correspondingly, the mean droplet radius exhibits several peaks of increase.

4. Conclusion

In this study, the condensation process of moist air inside a nozzle is numerically investigated using CFD coupled with a nucleation model and a droplet growth model, with emphasis on the temperature, velocity, and condensation characteristics in the nozzle. The following conclusions are drawn:

At the shock-wave location, velocity and temperature are mutually converted in the form of energy. The shock wave causes a decrease in velocity and an increase in temperature; however, after passing through the shock-wave region, both

velocity and temperature exhibit a slight recovery.

Nucleation inside the nozzle primarily occurs in the throat region, whereas droplet growth mainly takes place in the divergent section and the wake region. The magnitude of the growth rate is also influenced by the shock wave, which causes a slight reduction in the droplet growth rate.

References

- [1] Wegener P P, Pouring A A. Experiments on condensation of water vapor by homogeneous nucleation in nozzles[J]. *Physics of Fluids*, 1964, 7(3): 352–361. DOI: 10.1063/1.1711206.
- [2] Cheng W, Luo X, Van Dongen M E H. On condensation-induced waves[J]. *Journal of Fluid Mechanics*, 2010, 651: 145–164. DOI: 10.1017/S0022112009993879.
- [3] Wiśniewski P, Dykas S, Majkut M, et al. A comprehensive analysis of the moist air transonic flow in a nozzle with a very low expansion rate[J]. *Applied Thermal Engineering*, 2022, 217: 119185. DOI: 10.1016/j.applthermaleng.2022.119185.
- [4] Xiao R, Jin W, Han S, et al. Numerical simulation on condensing flow of water vapor of wet natural gas inside the nozzle[J]. *Frontiers in Heat and Mass Transfer*, 2017, 9(1): 1–8. DOI: 10.5098/hmt.9.6.