COMPUTATIONAL FLUID DYNAMICS SIMULATION OF THE SUPERSONIC STEAM EJECTOR USING DIFFERENT CONDENSATION MODEL

by

***Lin CAI*[a*], *Miao HE*[a], *Ke-Zhen HUANG*[b], and *Wei XIONG*[b]**

[a] College of Mechatronics and Control Engineering Hubei Normal University, Hubei, China
[b] China Ship Development and Design Center, Hubei, China

Original scientific paper
https://doi.org/10.2298/TSCI180723236C

This paper addresses the non-equilibrium condensation (NEC) in supersonic steam ejector under the assumptions of no slip velocity between the droplets and vapor phase and homogenous nucleation. The experimental data carried out by Moore has been used to verify the numerical results. It is illustrated that the maximum value of the flow mach number of the NEC model is lower than that of the equilibrium condensation model, and NEC model increases the ejector's entrainment ratio in comparison equilibrium condesation model. When using the NEC model, the nucleation characteristics such as subcooling degree, nucleation rate could be obtained in ejector flow field.

Key words: *ejector, CFD, droplet, non-equilibrium condensation*

Introduction

The supersonic steam ejector is widely used in the field of steam power, oil, thermoelectric, steam power ship, refrigeration and so on. For the capacity of extracting large volumes of vapor within a relatively small space and at a low cost, supersonic steam ejector is widely used on condensing steam turbine to hold the vacuum of the condenser, also to pump the non-condensable air, thereby indirectly enhancing the efficiency of the Rankine cycle.

The theory of ejectors was first developed through 1-D aerodynamic design and experimental method [1], the ejector performance curve using in refrigeration system was introduced by Huang *et al.* [2] . Recent years, with the development of computer and CFD, many scholars use CFD method to study the ejector performance, lots of studies show that CFD method could save lots of test costs, especially in researching optimization of ejector structural parameters. Sriveerakul *et al*. [3] used a commercial CFD package to predict the performance of a steam ejector the effects of the primary nozzle, throat diameter, and throat length on entrainment ratio (ER) are discussed based on numerical results. Zang *et al*. [4] and Cai *et al.* [5] carried out the ejector nozzle parameters optimization work, and the total length of the ejector had been reduced 23.4% while ER increased 58.8%. Except the studies on ejector performance, many works have been done on the numerical model of CFD method, such as turbulence mode and equations of state.

[*] Corresponding author, e-mail: cailin03313@163.com

The real physical property of water vapor is more complex than that of ideal gas, on the other hand, phase transition and liquid droplet nucleation phenomena caused by supersonic expansion are neglected when using the ideal gas model, there by yielding different calculation results that differ from the factual data, which had been discussed in our previous study [4, 5], other scholars' work on ejector numerical calculations with the comparison of experimental also illustrate the importance of physical properties or condensation. The phase changed phenomenon of high speed steam flow in Laval nozzle or ejector is very complicated. According to the theory of steam nucleation, when the steam flow past the nozzle throat with high speed, the temperature and pressure fall sharply, the rate of cooling is greater than the reduction in saturation temperature, and therefore, the fluid turns into a *sub-cooled* state, but condensation will not happened immediately, when steam temperature is lower than saturation-line, it will continue expand as superheat steam state, which is called thermodynamic *non-equilibrium* state, as the flow developed, the non-equilibrium flow reaches its extreme point, and water vapor is condensed into small droplets with latent heat releasing, then the steam return to equilibrium state. Oswatitsch first applied the classic nucleation theory to moist gas-flow problem in Laval nozzle (see in Bakhtar *et al*. [6]). Gyarmathy [7] used the classic nucleation theory to solve the wet steam field in steam turbine, Young and Yan [8] fixed the droplet growth rate model on the basis of Gyarmathy's work, and the 1-D numerical results agreed with experiment data well. Gyarmathy and Young's droplet growth rate model is still widely used in wet steam flow problems such as Laval nozzle, cascades, steam turbine until now.

The recent listed studies have shown that steam condensation needs to be considered in ejector flow filed problem. In this paper, on the basis of a study done before, the focus is drop nucleation in supersonic steam ejector. Two condensation model NEC and EC model have been compared on the assumption of homogeneously flow using CFX 15.0. The NEC model is firstly validated through the experimental data reported for a single nozzle and then applied to the computational domain of the ejector, the differences of mach number, static pressure, static temperature and ER between two condensation models have been discussed.

Mathematical models

Basic equations

As the liquid droplet diameter is very small, and the steam velocity is high, the non-NEC models used in this paper are based on the assumption of no slip velocity between vapor phase and the liquid droplets, so the basic equations for NEC and EC model are given [9]:

$$\frac{\partial U}{\partial t}+\frac{\partial F}{\partial x_i}=0 \tag{1}$$

where U and F are defined:

$$U=\begin{bmatrix}\rho\\ \rho u_i\\ \rho H\end{bmatrix},\quad F=\begin{bmatrix}\rho u_i\\ \rho u_i u_j+p-\mu_t\dfrac{\partial u_i}{\partial x_j}\\ \rho u_i H+p-\Gamma_t\dfrac{\partial T_c}{\partial x_j}\end{bmatrix} \tag{2}$$

and ρ and H are functions of mixture phase related to mass fraction β of liquid phase.

Nucleation and droplet growth equations for NEC model

In the NEC model, the classical nucleation theory is adopted, liquid phase appears in small drop form through the process of nucleation. The classical nucleation theory holds that as the level of subcooling degree is large enough to overcome a free-energy barrier, a large number of small spherical droplets will be generated, after that, the heat and mass transfer between the released droplets and vapor phase brings the flow back to near equilibrium condition. The number of small droplets generated is given [9]:

$$J = \frac{q_c}{1+\eta}\left(\frac{2\sigma}{\pi m^3}\right)^{1/2}\frac{\rho_v}{\rho_1}\exp\left(-\frac{4\pi r^{*2}\sigma}{3KT_c}\right) \tag{3}$$

where q_c, σ, m, ρ_v, ρ_1, K, and T_c are the condensation coefficient, the surface tension of droplets, mass of a water molecule, the density of the vapor and water liquid, the Boltzmann's constant, and the temperature of vapor, respectively. The Kantrowitz's non-isothermal correction factor, denoted by η, is given:

$$\eta = 2\frac{\gamma-1}{\gamma+1}\frac{L}{\mathrm{R}T_c}\left(\frac{L}{\mathrm{R}T_c}-\frac{1}{2}\right) \tag{4}$$

where γ is the specific heat capacity of vapor, L – the latent heat, and R – gas specific constant.

The critical droplet radius, denoted by r^*, is defined by:

$$r^* = \frac{2\sigma}{\rho_1 \Delta G_c} \tag{5}$$

where ΔG_c *i*s the bulk Gibbs free energy change of vapor phase depending on the physical parameters of water vapor (IAPWS-IF97).

Based on the droplet heat transfer energy balance, the droplet radius growth or decrease can be expressed:

$$\frac{\mathrm{d}r}{\mathrm{d}t} = \frac{k_c}{r\rho_d(1+c\mathrm{Kn})}\left(\frac{T_d-T_c}{h_c-h_d}\right) \tag{6}$$

where Kn is the Knudsen number, k_c – the thermal conductivity of the vapor, $c = 3.81$ – the an empirical factor, and T_d – the droplet temperature, given:

$$T_d = T_s(p) - T_{sc}\frac{r^*}{r} \tag{7}$$

Turbulent and numerical model

The SST k-ω turbulence model are adopted for turbulence closed, which is the same with our previous works. The turbulence model is applied homogeneously to all phases, the details of SST k-ω turbulence model could be found [10].

In this work, the CFD software ANSYS CFX 15.0 is used for solving the governing equations. Both NEC and EC model have already been implemented within the commercial implicit solver ANSYS CFX. In NEC model, the single phase flow field is provided for initialization in order to reduce the calculation time.

The primary fluid pressure of the ejector studied in this paper is 1.4M Pa and the temperature is 200 °C. The secondary fluid is saturation steam and its pressure is 0.06 Mpa. the back pressure of the ejector outlet is 0.1 MPa. A professional mesh generation software ICEM-CFD is used to generate full-hexahedral grid, in order to improve the analysis efficiency and reduce the calculation time, the periodic geometry and boundary condition are used, after grid depended learning, the total number of cells is about 250000.

Numerical validation for NEC model

The 3-D experimental data in nozzle carried out in [11] has been used for numerical validation, the nozzle width between nozzle side walls is 152 mm, the centerline pressure distribution of side wall has been given in [11] for test A~E. Because the No. E is working with wet steam and the inlet humidity is not known, the tests of No. B is chosen. The nucleation bulk tension factor (NBTF) has a great influence on the nucleation rate in eqs. (3) and (8). The surface tension coefficient σ can be written:

$$\sigma = \sigma_{\text{bulk}} \cdot NBTF$$

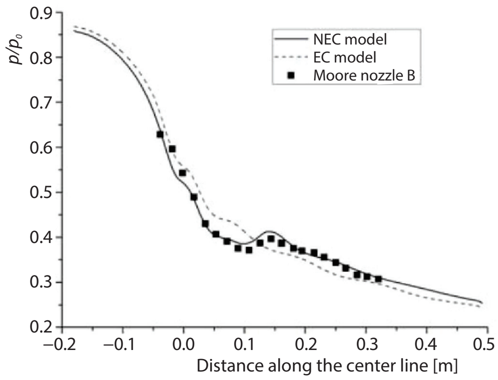

Figure 1. Comparison between the numerical and experimental results for distribution of pressure ratio along the centerline of the Moore's nozzle

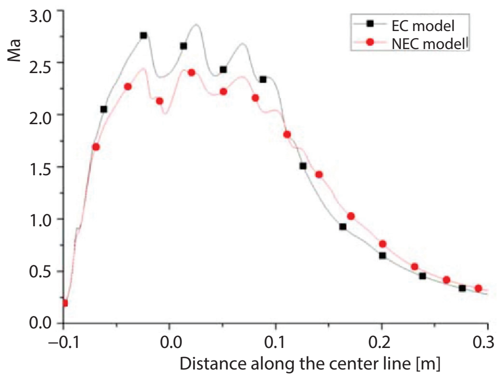

Figure 2. Mach number distribution along the center line of two models

So it could be seen that the NBTF is very sensitive to the calculation results of nucleation rate, in this paper, 0.85 is selected for the nozzle flow field, and the default value 1.0 is selected for the ejector flow field calculation. Comparisons of calculation results and experiment data are shown in fig. 1. It is shown that the static pressure ratio distributions along the center line are in good agreement with the experimental data, the pressure jumps due to condensation are also well captured. Figure 3 gives the results of different condensation model, it could be seen that the pressure distribution of two model both agree with experiment data well in front of throat, but for the *sub-cooled* effect of real high speed steam flow, the vapor will expand as superheat steam state until it crosses the Wilson zone.

Results and discussion

Figure 2 shows the Mach number distribution of two model. It could be seen that, there is a shock train (about three obvious oblique shock waves) in mix-chamber, the locations of the shock wave are almost the same of two model, but its maximum number are different, for the EC model the maximum mach number is 2.85, while for the NEC model, the maximum mach number is 2.4, the difference in the number of Maher is probably caused by two reasons: the temperature distribution is different, especially in condensation zones, another reason is that, there is a slightly different in velocity of two models, because the expansive degree in condensation zone is different. Figure 4 also shows that the maximum values of Mach number occur on the axis of the ejector flow field, the cross points of oblique shock waves could be seen clearly, moreover, it could be seen that main shock wave is formed as a weak *oblique shock* with the development of the flow along the axis, for both NEC model and EC model. Unlike other scholars' research [12] results that there is a strong *normal shock* in mixing chamber before the diffuser outlet, in fact, the strength of the shock is determined by the geometrical structure and working parameters, if the ejector is working at design conditions, the chock flow is easily formed in mixing chamber throat, but in this paper, the calculation condition is not at the design point, the primary pressure is lower than the design pressure 0.2 MPa.

Figure 3 gives the static pressure distribution along the center line of two model, the static pressure is the most important parameter in ejector field, because it determines the distribution of shock in the ejector and the entrainment performance. It could be seen that, the difference of pressure distribution between the two models at the front of the first shock is not obvious, and it also shows that, the shock distribution is similar of the two models, the pressure jump is mainly because primary fluid is compressed when it expanding into the mixing chamber, and the steam flows in mixing chamber with high and low supersonic flow, formed like *diamond mesh*. The first large pressure oscillation of the NEC model is larger than EC model, but after the first shock, the static pressure of the NEC model is lower than EC model at the same location, moreover, the first shock wave location of the NEC model is slightly more than the EC model about 10 cm, this is mainly caused by the effect of supersaturation of the NEC model, the expansion of the working fluid in primary nozzle is higher than that of EC model. For the effect of supersaturation, the temperature distribution of NEC model is lower than EC model at the same location, as shown in fig. 4. It also could be seen that the location of static temperature jump of NEC model is more than that of EC model, in the authors' view, this phenomenon is caused by supersaturated liquid droplet nucleation effect, which cannot be simulated through EC model, as the droplet generated in high speed steam field, the droplet flows to outlet with high speed, but when the temperature is higher than liquid saturation temperature, the droplet will not immediately evaporate, this is thermodynamic non-equilibrium state, which makes the temperature jump is behind the location of the EC model, and also makes the wetness distribution along the center line in mixing chamber much longer than EC model.

As the NEC model could consider the non-equilibrium property of high speed steam, the vapor subcooling degree is obtained as

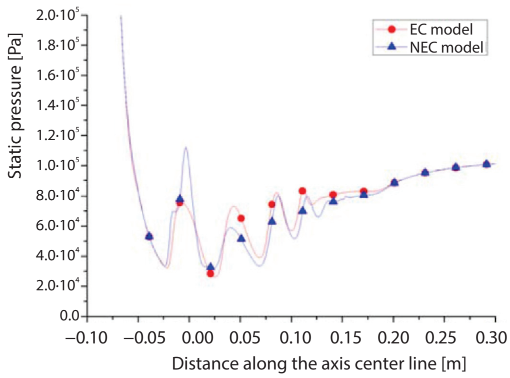

Figure 3. Static pressure distribution along the center line of two models

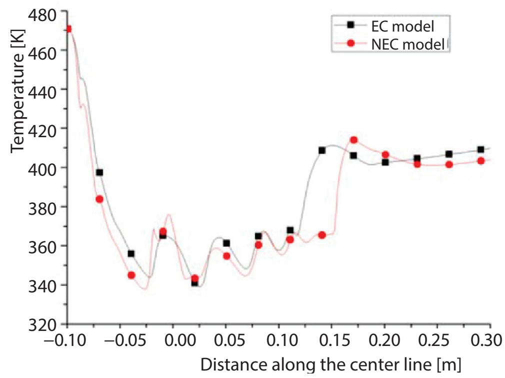

Figure 4. Static temperature distribution along the center line of two models

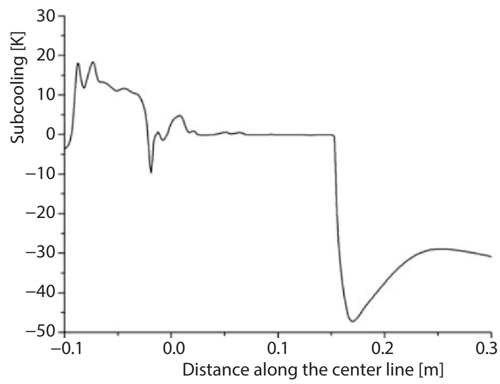

Figure 5. Subcooling degree distribution along the center line

shown in fig. 5, the maximum subcooling degree in ejector is about 18 K, the subcooling phenomenon occurs at the back of the primary nozzle throat zone, and it begin to go down in the diffuser zone of the primary nozzle, it can be seen from fig. 5 that at the location about $X = 0.02$ (after the inlet of the mixing chamber), the subcooling degree curve has extreme points, this may be caused by steam mixing, while the secondary steam enthalpy is lower, which makes the steam in mixing chamber subcooling degree increasing.

Conclusion

The non-equilibrium condensation model had been used to calculate the flow field of supersonic ejector, compared to the EC model under the same boundary condition. To ensure the accuracy of NEC model calculation, the experiment data reported by Moore were adopted, and the static pressure ratio distributions of nozzle A to D are in good agreement with the experimental values. The non-equilibrium effect could been considered using NEC model, the present model based on the combination of classical nucleation theory and Navier-Stokes equation could predict the condensation nucleation in the supersonic steam ejector. The numerical results of Moore's nozzle shows that the NEC model agrees with experimental data better than EC model, especially for the pressure jump due to condensation, but the NBTF is different for each nozzle. When using the NEC model, the ER value of the ejector is higher than that of EC model under the same boundary conditions, it seems that as the droplets generated in supersonic zones, the volume fraction of NEC model is much lower than that of EC mode, which enhance the ejector performance. The volume fraction distribution of two models are remarkably different, the wetness zone along the axial direction of NEC model is longer than that of the EC model.

Nomenclature

ΔG_c – bulk Gibbs free energy change, [–]
H – total enthalpy, [Jkg^{-1}]
J – nucleation rate, [–]
K – Boltzmann's constant (=1.3807 · 10^{-23} J/K), [–]
Kn – Knudsen number, [–]
L – latent heat, [–]
p – pressure, [Nm^{-2}]
q_c – condensation coefficient, [–]
R – gas specific constant, [–]
r – droplet radius, [m]
T – temperature, [K]
u – velocity, [ms^{-1}]

Greek symbols

Γ_t – effective thermal conductivity, [Wm^{-1}K^{-1})]
γ – vapor specific heat ratio, [–]
μ_t – effective dynamic viscosity, [kgm^{-1}s^{-1}]
ρ – density, [kgm^{-3}]
σ – surface tension, [Nm^{-1}]
χ – spatial dimension, [m]

Subscripts

i, j – tensor notation, [–]
l – liquid phase, [–]
s – saturation state, [–]
v – vapor phase, [–]

References

[1] Keenan, J. H., *et al.*. An Investigation of Ejector Design by Analysis and Experiment, *Journal Appl. Mech. Trans. ASME, 72* (1950), Jan., pp. 299-309
[2] Huang, B. J., *et al.*. Ejector Performance Characteristics and Design Analysis of Jet Refrigeration System, *Journal Eng. Power Trans. ASME, 107* (1985), 3, pp. 792-802
[3] Sriveerakul, T., *et al.*. Performance Prediction of Steam Ejector Using Computational Fluid Dynamics – Part 2: Flow Structure of a Steam Ejector Influenced by Oper-Ating Pressures and Geometries, *Int. J. Thermal Sci.*, *46* (2007), 8, pp. 823–833
[4] Zheng, H. T., *et al.*. Computational Fluid Dynamics Simulation of the Supersonic Steam Ejector – Part 1: Comparative Study of Different Equations of State, *Proc. IMechE Part C: Journal Mechanical Engineering Science, 226* (2012), 3, pp. 709-714
[5] Cai, L., *et al.*. Computational Fluid Dynamics Simulation of the Supersonic Steam Ejector – Part 2: Optimal Design of Geometry and the Effect of Operating Conditions on the Ejector, *Proc. IMechE Part C: Journal Mechanical Engineering Science. 226* (2012), 3, pp. 709-714

[6] Bakhtar, F., *et al.*. On the Performance of a Cascade of Turbine Rotor Tip Section Blading in Wet Steam. Part 2: Surface Pressure Distributions, *Journal Mech. Eng. Sci*. *211* (2005), 7, pp. 531-540
[7] Gyarmathy, G.. Bases of a Theory for Wet Steam Turbines, Ph. D. thesis, Federal Technical University of Zurich, Zurich, Switzerland, 1962
[8] Young, J. B., Yan, K. K., The Inertial Deposition of Fog Droplets on Steam Turbine Blades, *Journal Turbomach*. *110* (1988), 2, pp. 155-162
[9] Yu, X. G., *et al.*, Numerical Investigation of Condensing Flow in the Last Stage of Low-Pressure Nuclear Steam Turbine, *Nuclear Engineering and Design*, *275* (2014), Aug., pp. 197-204
[10] Dykas, S., *et al.*. Predictions of Losses in the Flow through the Last Stage of Low-Pressure Steam Turbine, *Int. J. Numer. Methods Fluids*, *53* (2006), 6, pp. 933-945
[11] Moore, M. J., *et al.*, Predicting the Fog-Drop Size in Wet-Steam Turbines, *Institute of Mechanical Engineering*, (1974), Jan., pp. 101-109
[12] Sharifi, N., *et al.*, Numerical Assessment of Steam Nucleation on Thermodynamic Performance of Steam Ejectors, *Applied Thermal Engineering*, *52* (2013), 2, pp. 449-459

Paper submitted: July 23, 2018
Paper revised: September 23, 2018
Paper accepted: December 25, 2018