Amirkabir University of Technology)
(Tehran Polytechnic)
Vol. 45, No.1, Summer 2013, pp. 5-7

Amirkabir Journal of Science & Research (Mechanical Engineering)
(AJSR - ME)

Numerical Modelling of Transonic Two Phase Flow with Shock in Converging-Diverging Nozzle

S. Hamidi[1], M. J. Kermani[2*] and H. Beheshti A[3]

1 Department of Mechanical & Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran
2 Department of Mechanical Engineering, Amirkabir University of Technology, Tehran, Iran
3 Department of Mechanical Engineering, Amirkabir University of Technology, Tehran, Iran

(Received 26 Feb 2011; Accepted 1 Dec 2012)

ABSTRACT

In this work, the numerical solution for compressible, unsteady, inviscid, two-phase and transonic liquid-vapor mixture flow is investigated using Ros's FDS time marching method. For space discretization, the fluid properties are extrapolated to the cell faces with the third order MUSCL algorithm of van Leer, and the time integration is done with the explicit two-step Lax-Wendroff method. In this study, the continuity, momentum and energy equations have been written in the fully conservative form and the properties of two-phase flow mixture in the quasi one-dimensional convergence-divergence nozzle have been investigated using the equilibrium thermodynamic model. The paper follows our earlier work, in which condensing transonic two-phase flow in a shock-free (no shock) converging-diverging nozzle was studied. The main goal of this article is to exhibit the two-phase flow with normal shock and to show the related physics (e.g. liquid phase evaporation via the shock) of the problem completely.

KEYWORDS

Roe's Method, Inviscid Flow, Compressible Flow, Normal Shock, Two Phase Flow, Equilibrium Thermodynamic, Quality.

* Corresponding Author, Email:mkermani@aut.ac.ir

1- INTRODUCTION

Condensation is a common phenomenon in many industrial thermo-fluid problems. Examples include flows in vapor nozzles, steam turbines, low temperature fuel cells, and many more. In such problems, flows contain vapor as one of the species that condenses due to the removal of latent heat from the vapor species.

A comprehensive review on the measurements of low pressure vapor nozzles (about 25 kPa or less) has been presented by Moore et al. [1]. Recently, Kermani et al. have performed some numerical computations of pure steam flow using the equilibrium thermodynamics [2]. In this work, the numerical solution for compressible, unsteady, inviscid, two-phase and transonic liquid-vapor mixture flow is investigated using Ros's FDS time marching method [3]. The solver is spatially third order and temporally second order accurate. The flow is assumed to obey the equilibrium thermodynamic model. For the two-phase flow in dry regions, the pressure (*P*), temperature (*T*), and velocity (u), are extrapolated to the cell faces by the MUSCL approach while in wet regions the steam quality (χ) has been used instead of pressure. The spurious numerical oscillations in the present high resolution computations are damped using the van Albada flux limiter [4]. The expansion shocks have also been avoided using the entropy correction formula given by Kermani and Plett [5]. A comparison of the literature shows good agreement.

The present paper is an extension of the previous work [2] that takes the working fluid as the two-phase and transonic liquid-vapor mixture with shock wave in the converging-diverging nozzle.

2- GOVERNING EQUATIONS

The governing equations for quasi one-dimensional, unsteady, inviscid and compressible flows are composed of the conservation laws of continuity, momentum and energy, and are shown in full conservative form. In the absence of body forces one can write [6]:

$$\frac{\partial Q}{\partial t}+\frac{\partial F}{\partial x}+H=0 \qquad (1)$$

$$Q=A\begin{bmatrix}\rho\\ \rho u\\ \rho e_t\end{bmatrix}, F=A\begin{bmatrix}\rho u\\ P+\rho u^2\\ \rho u h_t\end{bmatrix}, H=\begin{bmatrix}0\\ -P\dfrac{dA}{dx}\\ 0\end{bmatrix} \qquad (2)$$

Here *Q*, *F* and *H* are the conservative vector, the flux vector and the source term, respectively. A is the cross-sectional area of the nozzle, ρ is the density of the mixture, *u* is the velocity and $e_t$andh_t are respectively the total internal energy and total enthalpy of the mixture. In this study, the slip velocity between the gas and the liquid phases is ignored.

3- RESULTS

In the present computations, nozzle (A) from Moore et al. is considered [1]. In Fig. 1 the pressure distribution along this nozzle is compared with the experimental data. According to Fig. 1, there is a good agreement between our results and the experimental data. Figure 2 shows the profile of wetness fraction, (1-χ), along the $Moore_A$ nozzle for two back pressures 18 and 20 kPa. For these back pressures, shock waves appear in the diverging portions of the nozzle. For the present two-phase flow with a strong shock, the post shock condition has completely dried out. Figure 3 shows the profile of entropy along the $Moore_A$ nozzle with the normal shock. In the case with shock wave, the entropy of the mixture (s_m) increases across the shock.

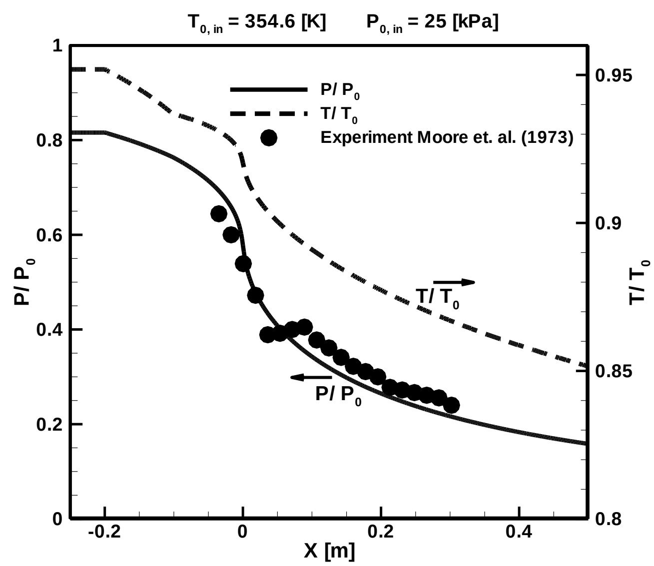

Figure 1: validations of the present solution; comparisons of pressure distribution along the $Moore_A$ nozzle.

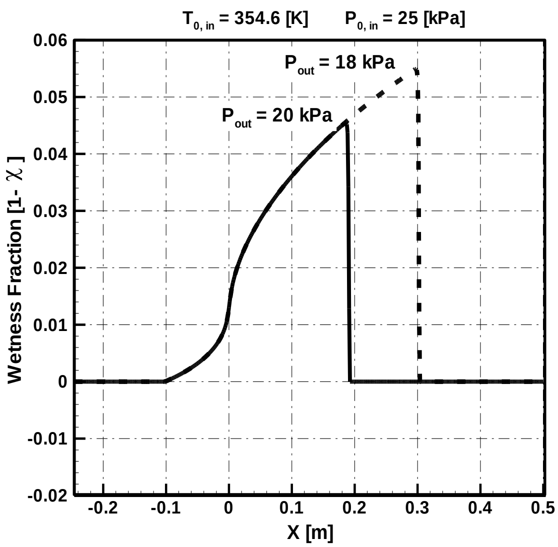

Figure 2: Wetness fraction profile along the $Moore_A$ nozzle for two different back pressure values.

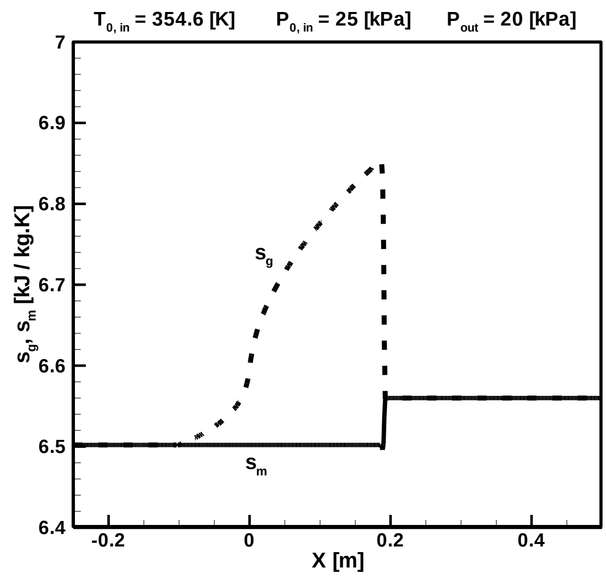

Figure 3: Entropy profiles along the $Moore_A$ nozzle with shock wave (subsonic outflow)

4- CONCLUSIONS

Comparisons of the numerical results with the experimental data show relatively good agreement. So the numerical method which is proposed in this paper is capable of simulating compressible flow problems with satisfactory precision. The following conclusions can be drawn:

a) The liquid phase rapidly evaporates because of the rapid pressure and temperature rise across the shock.

b) The entropy of the mixture (s_m) increases across the normal shock

5- REFERENCES

[1] M.J. Moore, P.T. Walters, R.I. Crane, and B.J. Davidson, 1973 “Predicting the Fog Drop Size in Wet Steam Turbines,” Inst. of Mechanical Engineers (UK), Wet Steam 4 Conference, University of Warwick, paper C37/73.

[2] M. J. Kermani, A.G.Gerber, and J.M. Stockie, “An Application of Roe's High Resolution Scheme to Transonic Two-Phase Flow through Nozzles,” Iranian Journal of Mechanical Engineering: Transaction of the ISME, E-84-04, vol. 7, no. 1, pp. 60-77. 2006.

[3] P.L. Roe, “Approximate Riemann Solvers, Parameter Vectors and Difference Schemes,” J. Comput. Phys., no. 43, pp. 357-372. 1981.

[4] G.D. van Albada, B. van Leer, and W. W. Roberts, “A Comparative Study of Computational Methods in Cosmic Gas Dynamics,” Astronomy and Astrophysics, vol. 108, no. 1, pp. 76-84. 1982

[5] M. J. Kermani, and E.G. Plett, “Modified Entropy Correction Formula for the Roe Scheme,” AIAA Paper, 2001-0083. 2001

[6] K.A. Hoffmann, and S.T. Chiang, “Computational Fluid Dynamics for Engineers,” Vol. II, Engineering Education Systems, Wichita, Kansas, USA. 1993.