ADVANCES in NATURAL and APPLIED SCIENCES

ISSN: 1995-0772
EISSN: 1998-1090
2016 September 10(13): pages 1-8

Published BY**AENSI Publication**
http://www.aensiweb.com/ANAS
Open Access Journal

Numerical Study of Steam Flow in a C-D Nozzlewith Experimental Verification

[1]Marwa Athab, [2]Arkan Al-Taie, [3]Hussein Mashi

[1]M.S.C Student, Mechanical engineering department, University of Technology. Baghdad, Iraq
[2]Prof. Dr. Mechanical engineering department, University of Technology. Baghdad, Iraq
[3]Dr. Mechanical engineering department, University of Technology. Baghdad, Iraq

Received 7 June 2016; Accepted 12 September 2016; Available 30 September 2016

Address For Correspondence:
M.S.C Student, Mechanical engineering department, University of Technology. Baghdad, Iraq

Copyright © 2016 by authors and American-Eurasian Network for Scientific Information **(AENSI Publication).**
This work is licensed under the Creative Commons Attribution International License (CC BY).http://creativecommons.org/licenses/by/4.0/

Open Access

ABSTRACT

Calculating flow propertyand shock wave location of saturated steam through a convergent-divergent nozzle in off-design conditions at different pressure ratios.The flow mixture is modeled using wet steam modelin conjunction with realizableK – ε. The modeling was done using ANSYS FLUNET 15 and solved for static pressure, static temperature, Mach number, liquid mass fraction of the flow for different nozzle inlet pressure at constant back pressure.The numerical investigation revealed that droplet formation and growth increases downstream the throat due to temperature decreases, while average liquid mass fraction in the nozzle exit decreases as the inlet steam temperature increases. Also the liquid mass fraction is decreases after the shock wave because of temperature increases.The wet steam model in conjunction with realizableK – ε predicted the flow properties with acceptable accuracy as compared with the experimental measurements.

KEYWORDS:C-D nozzle, steam, shock wave, wet steam, liquid mass fraction.

INTRODUCTION

Analysis of steam flow through nozzles is the subject of many researches due to its wide applications and the complex nature of the flow. In the last years the tools, methods, and models used in the analysis has become more capable and more accurate which allowed the researchers to do in depth accurate and efficient investigations.

Tianm C and Yijia L[1] noticed that the realizable k–ε turbulence model gives the best results for predicting the shockwave position and separation point.A shockwave study in a convergent-divergent nozzleby using numerical methodemployed in CFD code, FLUENTwere introduced by Padmanathan [2].The study concluded that there was an increase in static pressure, density and static temperature across the shock. A CFD code FLUENT ANSYS have been used to investigate the influence of the thermal parameters on changes of the condensation onset by examining the region of nucleation and growth of steam droplets along the flow directionwere considered byVilela [3].The study implemented that as the steam temperature increased the average liquid mass fraction in the nozzle exit decreased. The liquid mass fraction increased with increasing the pressure difference to the point where the steam is already in the form of saturated steam.Control-volume-based computational fluid dynamics (CFD) code Fluent was used to simulate steam flow in a nozzle of steam injector were introduced by Poorasdion [4]. Hegazy [5] studied the condensation process of steam flow through nozzleby using numerical method. The values of pressure ratio affected the position of the shock wave where increasing the pressure ratio would move the shock wave into the nozzle exit.

ToCite ThisArticle: Marwa H. Athab, Arkan Al-Taie, Hussein W.Mashi, Numerical Study of Steam Flow in a C-D Nozzlewith Experimental Verification University of Technology-Iraq.**Advances in Natural and Applied Sciences**. 10(13); Pages: 1-8

The main aim of the study is to calculating flow property and shock wave location of saturated steam through a convergent-divergent nozzle in off-design conditions at different pressure ratios.

2. Governing Equations:

To describe the wet steam model, the mass, momentum and energy conservation equations are used in the following form[6]:

The equation for conservation of mass is:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\bar{v}) = 0 \quad (1)$$

Conservation of momentum equation is:

$$\frac{\partial}{\partial t}(\rho\bar{v}) + \nabla.(\rho\bar{v}\,\bar{v}) = -\nabla p + \nabla.(\bar{\tau}) + \bar{F} \quad (2)$$

Where (p) is the static pressure, $\bar{\tau}$ is the stress tensor and $\bar{F}$ is the body forces.

Conservation of energy equation can be written as follows:

$$\frac{\partial}{\partial t}(\rho E) + \nabla.\left(\bar{v}(\rho E + p)\right) = \nabla.q + \nabla.(\bar{\tau}\bar{v}) \quad (3)$$

Where S_h is the source term describing the energy exchange between the phases.

In addition to two transport equations are needed.The first transport equation governs the mass fraction of the condensed liquid phase (β):

$$\frac{\partial \rho\beta}{\partial t} + \nabla \cdot (\rho\vec{v}\beta) = \Gamma \quad (4)$$

Where Γ is the mass generation rate due to condensation and evaporation (kg per unit volume per second).The second transport equation models the evolution of the number density of the droplets per unit volume:

$$\frac{\partial \rho\eta}{\partial t} + \nabla \cdot (\rho\vec{v}\eta) = \rho I \quad (5)$$

Where I is the nucleation rate (number of new droplets per unit volume per second).

To determine the number of droplets per unit volume,and the average droplet volume, in the following expression is used:

$$\eta = \frac{\beta}{(1-\beta)V_d(\frac{\rho_l}{\rho_v})} \quad (6)$$

Where: (ρ_l) is the liquid density and the average droplet volume is defined as

$$V_d = \frac{4}{3}\pi r_d^3 \quad (7)$$

Where r_d is the droplet radius.

3. Numerical Simulation:

The nozzle dimensions are taken from the experimentalset-up by[7] is modeled as a two-dimensional geometry by (CFD) code Fluent 15. The nozzle was discretized by quadrilateral cells. The governing equations (Equations (1) to (3)) are discretized by second order upwind scheme and solved by density-based solver, while for turbulence model the realizable$K-\varepsilon$is selected. The computations are stopped When Residues and the mass imbalance fall below 10^{-5}.Boundary conditions of the inlet and outlet faces are selected to be pressure inlet boundary with total pressure (1.5) bar and total temperature (384)K respectively.

4. Results:

4.1 Results Validation:

In order to verify the numerical model of the present work, the flow in the nozzle presented in Fig 1[7]was modeled and the flow characteristics were computed and the pressure ratio distribution compared with the experimental result. The results for the numerical and experimental pressure ratio distribution showed a good agreement as shown in Fig 2.

4.2 Steam Properties through the nozzle:

4.2.1. Static Pressure:

The numerical results in Fig 3 show it can be noticed that the static pressure is decreasing across the diverging part of the nozzle and continues to decrease through and downstream the throat till a sudden raise in the static pressure occur and it continues to rise until reaching the exit area. The sudden rise is due to the formation of a shockwave as was detected in the experimental investigation.

4.2.2. Mach No:

From Fig 4 it can be seen that Mach number is increasing through the convergent section and continues to increase to supersonic through and downstream of the throat, then a sudden change occurs as Mach number turn

to subsonic region, and continues to decrease as it reaches the exit of the nozzle. The sudden change is due to the formation of a shockwave between (x/l=0.06 and x/l=0.1).

4.2.3. Static Temperature:

Fig 6 shows the static temperature distribution along the nozzle.It shows that the temperature drops in the nozzle and rises again downstream.This is due to the expansion where the static temperature decreases as the flow approach the throat area where it reaches the lowest value at the throat. Then the formation of water droplets after the throat which causes the release of a heat, the static temperature increases gradually and stabilizes after the shockwave.

4.2.4. Liquid Mass Fraction:

Figs 8 and 9indicated the changes of liquid mass fraction along nozzle axis at the same inlet conditions. InFig.8, the liquid mass fraction reaches its maximum value downstream of the nozzle throat before shock occurrence. As a result of steam expansion and decreases of its temperature (which can be seen in Figs 6 and 7), the droplets are formed just downstream of the nozzle throat. From the figure it also can be noticed that the liquid mass fraction is reduced after the shock wave because of the increased temperature as can be seen in Figs (6 and 7).

5. Effect of the inlet pressure on the shock wave location and liquid mass fraction:

The inletpressure (Po) has a direct effect on the flow behavior inside steam nozzle where increasing the inlet pressure will decrease the pressure ratio (pressure ratio is the ratio between the back pressure at the nozzle exit (Pb)and the stagnation pressure at the nozzle inlet (Po)). The pressure ratio values affected the position of shock wave. Decreasing the pressure ratio moves the shock wave into direction of the nozzle exit. The effect of the pressure ratio on the pressure distribution is presented by Fig. 10.At the high pressure ratio (PR=0.66), the shock wave starts to appear near the nozzle throat between (x/l=0.06 and x/l=0.1). Decreasing the pressure ratio (PR=0.39 and PR=0.33) causes gradual moving of the shock toward the nozzle exit. The effect of the pressure ratio on the Mach number distribution along the nozzle axis is shown in Fig. 11.Fig 12represents effect of the pressure ratio on the liquid mass fraction, wherethe increaseof inlet pressure increases the liquid mass fraction.

Conclusions:

The following can be deduced from the numerical investigation:

1. Droplet formation and growth increases downstream the throat due to temperature decreases.
2. As the steam temperature increased, the average liquid mass fraction in the nozzle exit decreased.
3. The liquid mass fraction is decreased after the shock wave because of the temperature increases.
4. AS the inlet pressure increases the liquid mass fraction increases downstream the throat.
5. AS the inlet pressure increases the shock wave moves toward nozzle exit.

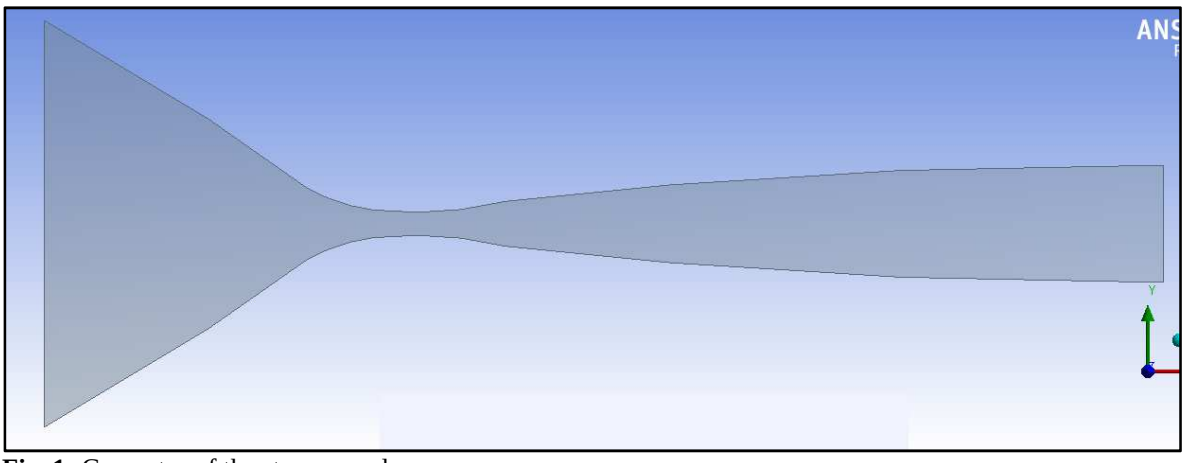

Fig. 1: Geometry of the steam nozzle

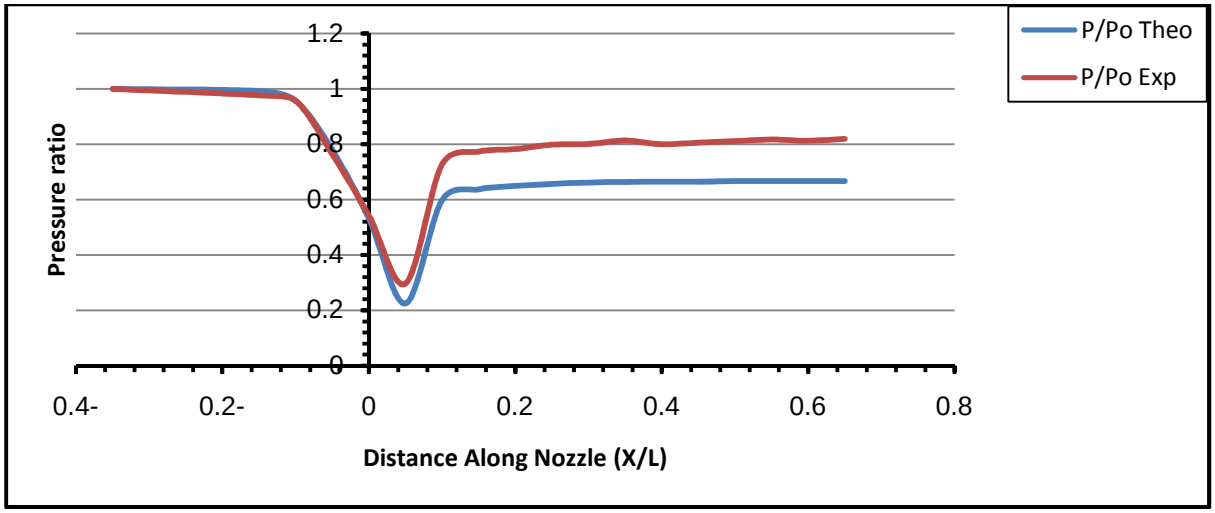

Fig. 2: Comparison between the numerical and experimental pressure ratio

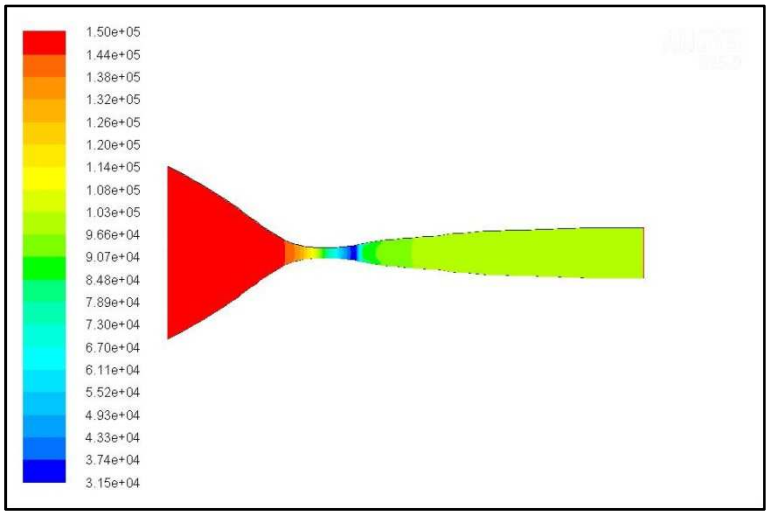

Fig. 3: Contour of Static Pressure

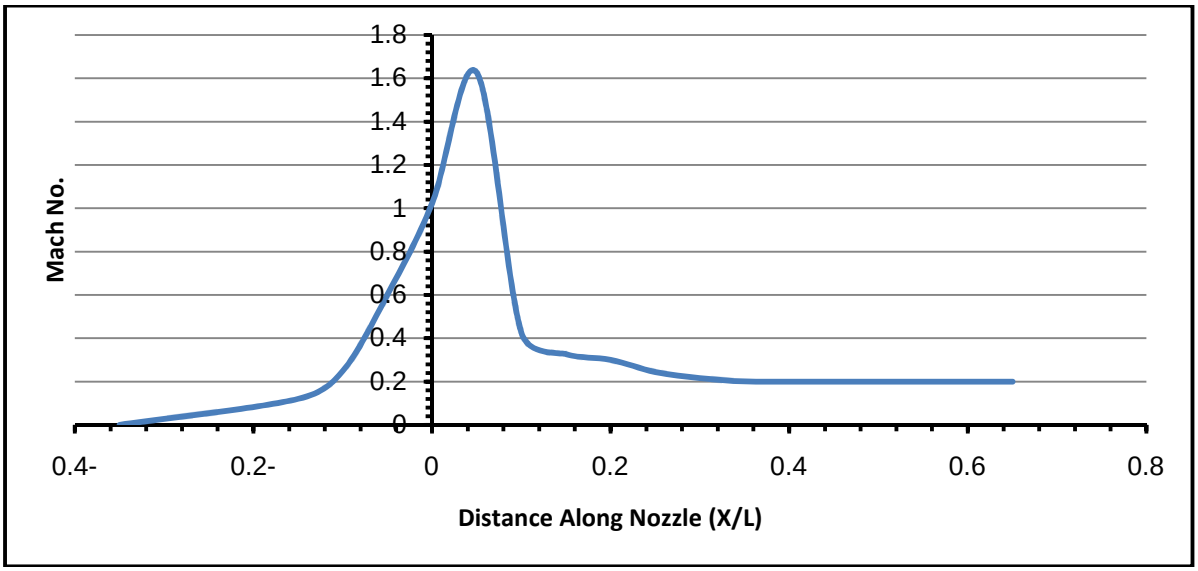

Fig. 4: Mach no. distribution

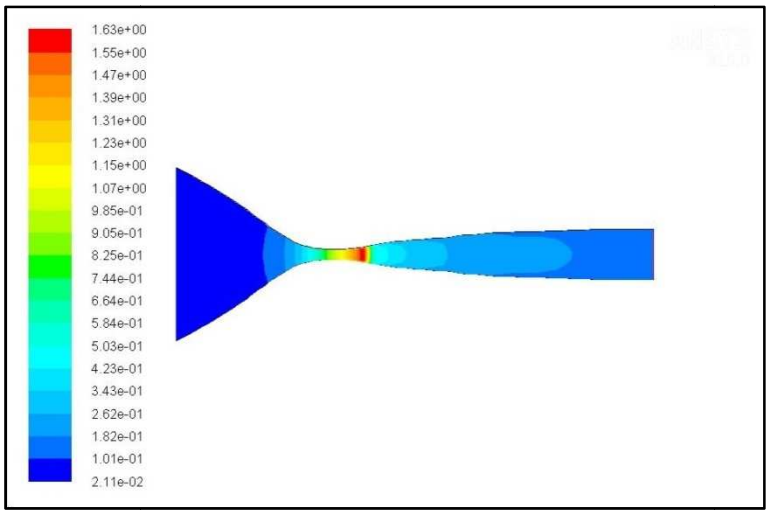

Fig.5: Mach no. contour

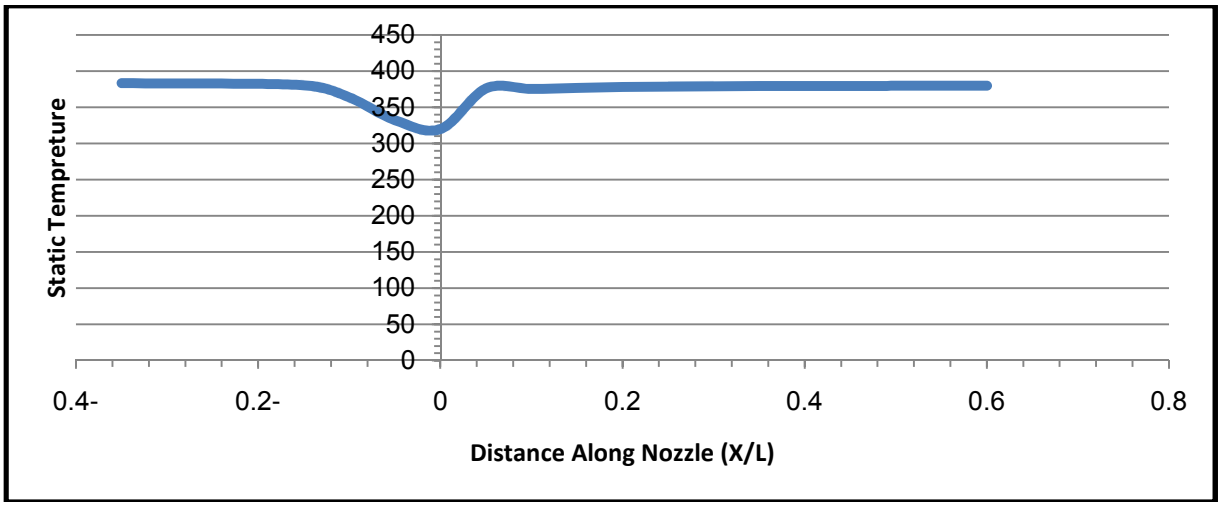

Fig. 6: Static temperature distribution

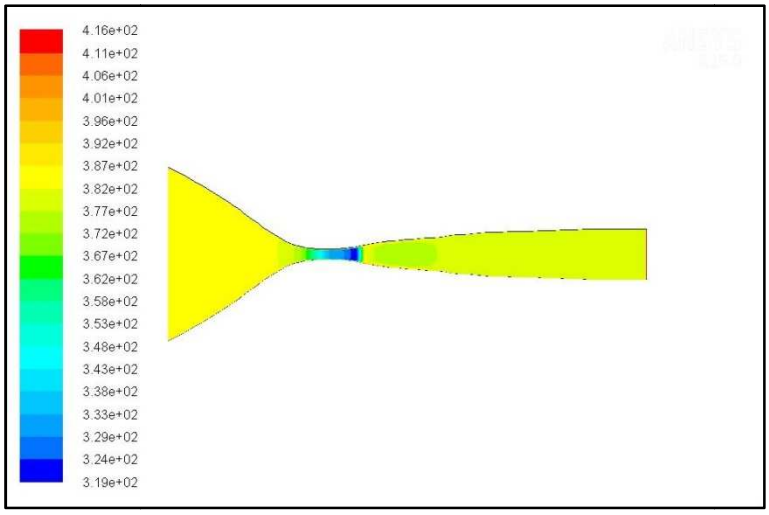

Fig.7: Contour of static temperature

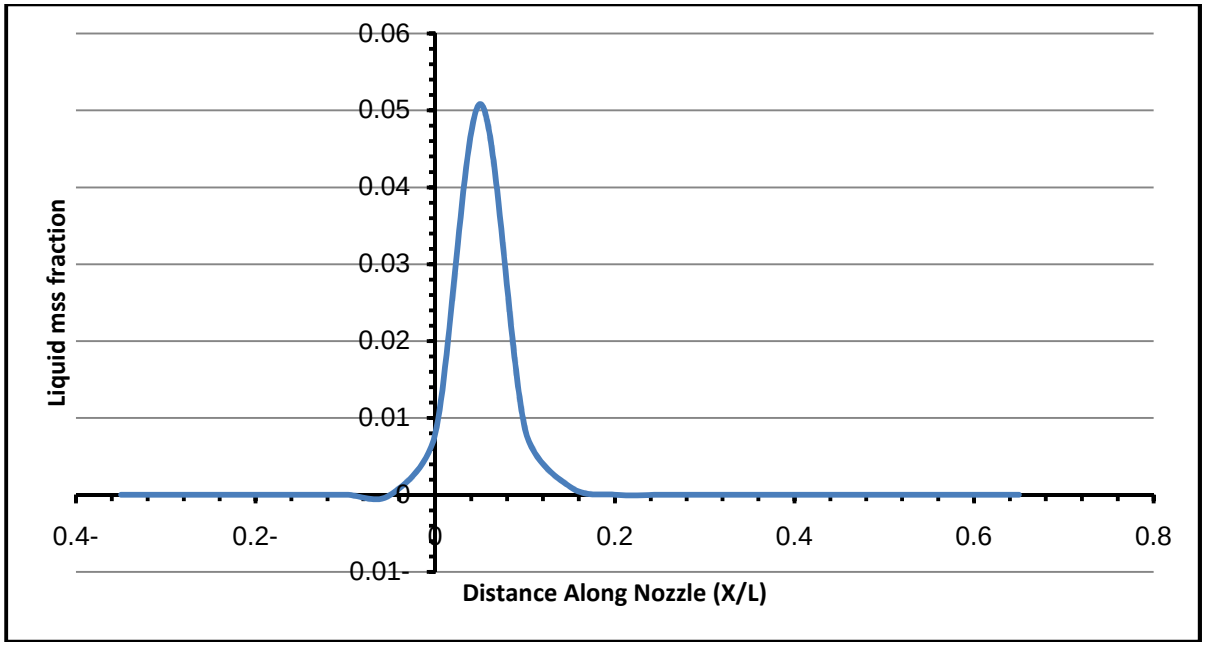

Fig. 8: Liquid mass fraction distribution

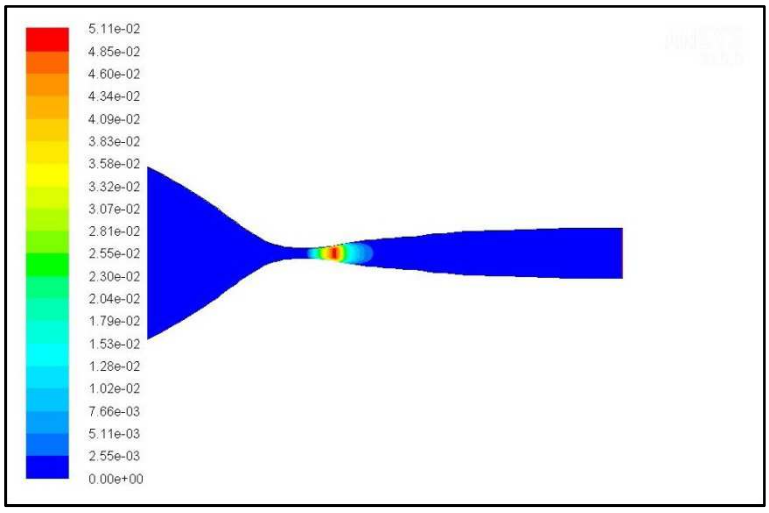

Fig. 9: Contour of liquid mass fraction

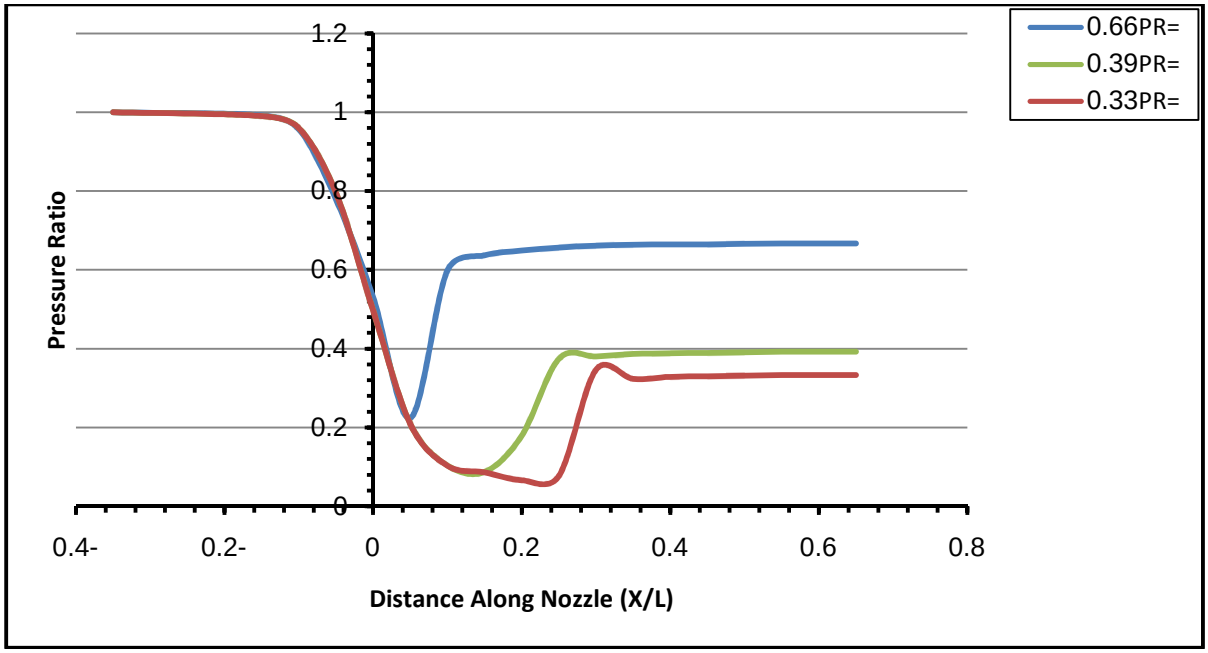

Fig. 10: Effect of pressure ratio on the pressure distribution along nozzle axis

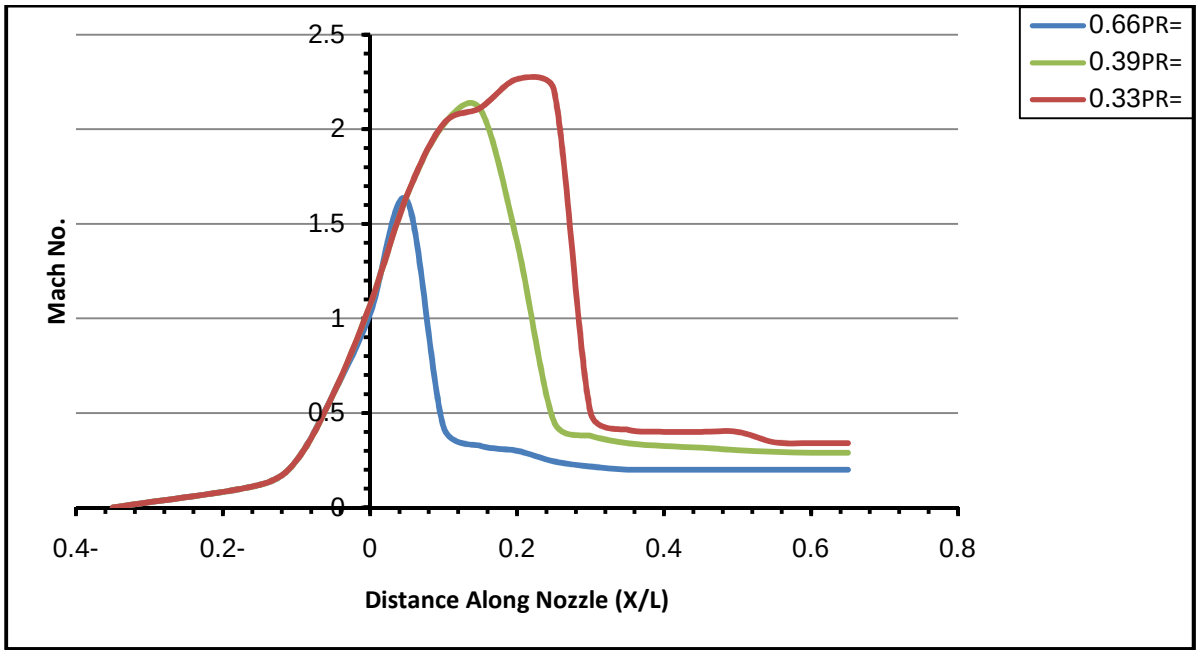

Fig. 11: Effect of pressure ratio on the Mach no. distribution along nozzle axis

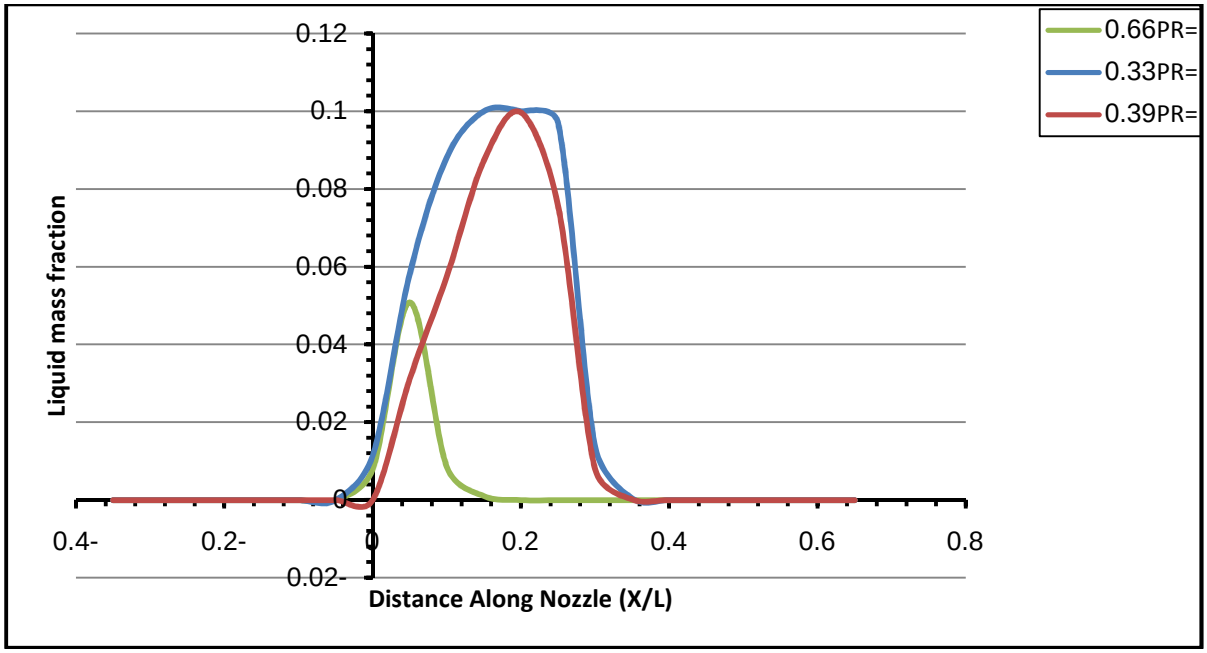

Fig. 12: Effect of pressure ratio on the liquid mass fraction distribution along nozzle axis

Nomenclature:
Latin Symbols:

r_d	Droplet radius
V_d	Average Droplet Volume

Greek symbols:

β	liquid – phase mass fraction
Γ	Mass generation rate.
η	Number of liquid droplets per unit volume.
$\bar{v}$	Mean speed of molecules
ρ	Density

REFERENCES

1. Tianm, C and L. Yijia,2013. Turbulence Models of Separated Flow in Shock Wave Thrust Vector Nozzle. Engineering Applications of Computational Fluid Mechanics, 7: 2.
2. Padmanathan, p and S. Vaidyanatha, 2012. Computational Analysis of Shockwave in Convergent Divergent Nozzle. International Journal of Engineering Research and Applications. ISSN: 2248-9622, 2: 2.
3. Vilela, R.D., C.R. Andrade and E.L. Zaparoli, 2010. CFD STUDY OF STEAM CONDENSING FLOW IN A LAVAL NOZZLE. 13rd Brazilian Congress of Thermal Sciences and Engineering, Brazil.
4. Poorasdion, S., M. Alishiri and M. Saadatmand, 2013. CFD INVESTIGATION ON THE STEAM CONDENSING PHENOMENA INSIDE NOZZLE OF STEAM EJECTOR, The International Desalination Association World Congress on Desalination and Water Reuse. IDAWC/TIAN13-250, Tianjin, China.
5. Hegazy, A.S., W.A. El-Askary, A.A. El-Terhy and M.S. Farag, Numerical Study on condensation Process of Steam Flow in Nozzles. Recent Advances on Mechanics, Materials, Mechanical Engineering and Chemical Engineering, ISBN: 978-1-61804-295-8.Eygpt.
6. Yong, Y and S. Shengqiang, 2009. Numerical simulation on non-equilibrium spontaneous condensation in supersonic steam flow. International Communications in Heat and Mass Transfer, 36: 902-907.
7. Marwa, H.A.,2016. Flow Study And Shock Wave Prediction In a Convergent-Divergent Nozzle With Experimental Verification. M.SC. Thesis , University of Technology, Iraq.