International Journal of Thermal Sciences 122 (2017) 257–265

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

journal homepage: www.elsevier.com/locate/ijts

Three-dimensional numerical investigations on rectangular cross-section ejector

K.M. Arun, Shaligram Tiwari, A. Mani*

Refrigeration and Air-conditioning Laboratory, Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai 600036, India

ARTICLE INFO

Article history:
Received 17 January 2017
Received in revised form
3 August 2017
Accepted 29 August 2017

Keywords:
Rectangular ejector
Compressible flow
Shock formation
Entrainment ratio
Turbulence model

ABSTRACT

The performance and flow structure in a three-dimensional rectangular ejector have been numerically investigated by using air as the working fluid. Commercial software ANSYS Fluent 15.5 has been used for numerical simulation of supersonic, compressible, turbulent flow ejectors. In order to confirm the appropriate turbulence model, the computed results are validated against indigenous experimental data. The study aims to optimize the convergent-divergent (C-D) nozzle position for each operating condition and to bring out the influence of reflected shock waves and boundary layer in the constant area mixing chamber on the performance of the ejector. Also, the variation of entrainment ratio (ER) for a range of primary, secondary and exit pressure has been investigated. Both the on-design and the off-design characteristics and local flow structure of a rectangular ejector have been considered.

1. Introduction

An ejector is a device that uses momentum of motive fluid to transport secondary fluid. In general, it consists of a driving nozzle, a constant area mixing section and a diffuser. The ejector system entrains secondary flow through shear action generated by the primary jet and the pressure difference between the two fluids. A mixing type shear layer is formed at the interface of primary and secondary fluids resulting into acceleration of secondary fluid by primary stream and its entrainment into the main flow. The performance of the ejector is expressed as 'entrainment ratio', defined as the ratio of secondary to primary mass flow rates. The efficiency of an ejector system is relatively low as compared to other fluid transport devices. However, its major advantage appears as a simple structure with no moving parts, being less expensive and transporting a fluid after compression without consuming electrical energy. That is the main reason why ejector system is widely used in refrigeration systems, crude oil distillation, petrochemical processes, hybrid vacuum systems and metal vacuum degassing.

In 1858, Henry Giffard invented condensing type ejector to find a solution to the problem of feeding liquid water to refill the reservoir of steam engine boilers. Since then, ejectors have been investigated for different applications. In 1910, Leblanc introduced a refrigeration cycle having an ejector to produce refrigeration effect by utilising low-grade energy [1]. Over the years, various researches have been done to design and evaluate ejectors in refrigeration field. The main motivation of the researchers is to entrain maximum secondary flow for a given primary flow rate and compress the entrained fluid to desired exit condition. Numerous experimental and numerical studies have been conducted to improve the performance of the circular cross-section ejector. Huang et al. [2] developed a solar ejector cooling system using R141b as the working fluid. Cizungu et al. [3] carried out a computer simulation of vapor jet refrigeration system using 1-D models based on mass, momentum and energy balances. They suggested that for any generator condition, the entrainment ratio is mainly dependent on the ejector geometry and the compression ratio. Selvaraju and Mani [4] developed a computer code based on the 1-D ejector theory to investigate performance of the ejector. The code was capable of evaluating the effect of specific heat of the working fluid and friction in the constant area mixing chamber. They also compared the performance of the ejector using environment-friendly refrigerants such as R134a, R152a, R290, R600a and R717. Bartosiewicz et al. [5] carried out numerical study to investigate the performance of supersonic ejectors using six different turbulence models in order to capture real flow characteristics that closely mimic the experimental observations. The validation that focused on shock location, shock strength and the average pressure

---

* Corresponding author.
E-mail address: mania@iitm.ac.in (A. Mani).

http://dx.doi.org/10.1016/j.ijthermalsci.2017.08.024
1290-0729/

Nomenclature

e	Total energy
G	Generation
k	Turbulent kinetic energy
M	Mach number
P	Pressure
q	Heat flux
S	Source term
T	Temperature
u	Velocity
x	Generalised Cartesian coordinate
Y	Dissipation

Greek

ρ	Density
τ	Viscous stress
ω	Specific dissipation rate
Γ	Effective diffusivity

Subscript

i	Direction along Cartesian coordinates x, y or z
o	Ejector outlet
p	Primary
s	Secondary

Abbreviations

AR	Aspect ratio
C-D	Convergent-divergent
CFD	Computational fluid dynamics
CFM	Cubic feet per minute
ER	Entrainment ratio
NEP	Distance between nozzle exit and inlet of constant area mixing chamber
RSM	Reynolds stress model
SST	Shear stress transport

recovery prediction, concluded that SST $k-\omega$ model predicts computational results showing best match with experiments. In 2007, Sriveerakul et al. [6] conducted a validation study to confirm suitability of using CFD and reported that it is a strong enough tool to predict the performance of the ejector as well as depict the flow and mixing processes in the ejector. They extended the study [7] to further investigate the flow physics in the ejector and confirmed the validity of concept of effective area used by Huang et al. [8]. Moreover, they observed the presence of two types of oblique shocks in the ejector, one immediately after the primary fluid stream leaves the C-D nozzle and second series of oblique shocks appear at the beginning of the diffuser section as a result of non-uniform mixed stream.

Selvaraju and Mani [9] conducted an experimental study on vapor jet refrigeration system using refrigerant R134a for a rated capacity of 0.5 kW. They concluded that there exists an optimum temperature of primary vapor at particular condensing and evaporating temperatures which yields maximum entrainment ratio and COP. Sankarlal and Mani [10] carried out an experimental study on ammonia ejector refrigeration system. They concluded that ER and COP of the ejector increase with increase in area ratio and expansion ratio and also increase with decrease in compression ratio.

Zhu et al. [11] investigated the effects of mixing section converging angle and primary nozzle exit position on performance of the ejector. In 2009, Lijo et al. [12] studied about transient flow through the vacuum ejector system using CFD method. The main motivation of the investigation was to analyse transient behaviour of the ejector when it is connected to a chamber being evacuated. They have plotted the flow field with respect to time and also observed a recirculation zone at the exit of the primary nozzle lower chamber pressure.

Chen *et al*, [13] conducted a numerical study on natural gas ejector to obtain the optimum geometry factors such as inclination angle of mixing chamber, ratio of diameter to length of the constant area mixing chamber, diverging angle of the diffuser, etc. They also verified their numerical results with a field experiment data. In 2012, Yang et al. [14] investigated the effect of different nozzle cross-sections on performance of ejector. They considered five different cross-sections of nozzle, such as circular, elliptical, square, rectangular and cross-shaped. It is reported that the square and cross-shaped nozzles give higher entrainment ratio than the circular nozzle. However, as far as effect of critical back pressure is concerned, it is found that the circular shape performs better than other nozzle shapes. In 2012, Openogrth et al. [15] investigated the performance of lobbed primary nozzle that has increased perimeter thereby more contact area between primary and secondary fluids. Effect of profiling the constant area mixing chamber on increase of ejector compression ratio has been included for nitrogen as working fluid. This investigation suggests that increasing number of lobes on the primary nozzle enhances entrainment of the secondary fluid up to a limit beyond which further increase in number of lobes will cause more friction loss due to increase in perimeter.

Liu et al. [16] developed a two-phase flow ejector expansion model and validated with experimental results. They concluded that motive nozzle efficiency decreases as ejector throat area decreases, and suction nozzle efficiency is affected by the ambient air temperature. Ruangtrakoon et al. [17] conducted a numerical study to investigate the effect of primary nozzle geometries on performance of the ejector of a steam jet refrigeration system and found that expansion angle of the primary jet stream has strong impact on the performance of the ejector. Yen et al. [18] also studied about ejector performance with varying area ratio by using different refrigerants as working fluid. Chen et al. [19] developed a 1-D model to predict the ejector performance and effectiveness of the model is verified against four sets of experimental data that include different working fluids and geometries such as circular and rectangular cross-section. Gagan et al. [20] conducted a comparison study to find out the best turbulence model with flow visualisation technique using particle image velocimetry (PIV) and concluded that standard $k-\omega$ model can predict the flow field more accurately than other models. In 2014, Mittal et al. [21] numerically investigated the transient behaviour of flow field at starting of the ejector and validated with experimental results. The ejector that has rectangular cross section throughout the length and one secondary inlet, uses a convergent primary nozzle instead of C-D nozzle. Results describe the way by which the flow field in the ejector achieves steady state after short transient effect. Such effects emanate from the complicated interactions of expanding primary jet and induced secondary stream through the shearing action.

Little et al. [22] validated CFD results of the large ejector of rectangular cross-section using shadowgraph flow visualisation technique. The results show that at the design condition *RNG* $k-\varepsilon$