International Journal of Thermal Sciences 78 (2014) 9–15

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Comparative study of turbulence models in application to gas ejectors

Jerzy Gagan [a], Kamil Smierciew [a], Dariusz Butrymowicz [a,*], Jaroslaw Karwacki [b]

[a] Bialystok University of Technology, Wiejska 45C, Bialystok 15-351, Poland
[b] The Szewalski Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, Gdansk 80-231, Poland

A R T I C L E I N F O

Article history:
Received 13 February 2013
Received in revised form
15 November 2013
Accepted 15 November 2013
Available online 18 December 2013

Keywords:
Refrigeration
Ejector
PIV technique
CFD modelling
Solar air-conditioning

A B S T R A C T

Gas and vapour single phase ejectors are commonly applied in variety of thermal systems for power generation as well as refrigeration. The general difficulties in design of the ejector system are lack of the reliable models of the ejectors. The most useful tool for prediction of operation of the ejector is CFD which requires selection of the turbulence model. The paper presents the flow visualisation investigations with application of PIV technique along with CFD modelling results based on which recommendation of the $k-\varepsilon$ standard turbulence model is formulated.

1. Introduction

Gas and vapour single phase ejectors are commonly applied in variety of thermal systems for power generation as well as refrigeration. A special interest is paid to the application of ejector refrigeration systems that are motivated by heat generated in solar collectors.

Many theoretical and experimental studies have been carried out in order to understand not only the fundamentals in terms on fluid dynamics and heat transfer but also ejector operation behaviour in case of various geometries and operation parameters. A good overview of literature in this field may be found in article of Sun [1] and El-Dessouky [2]. Most of theoretical studies are still semi-empirical or lumped parameters models. Keenan et al. [3] make a first step in the one-dimensional analysis by their model of a constant area mixing flow for air. This model was later developed by Munday and Bagster [4] and Huang et al. [5].

However, these models of constant-area or constant mixing pressure are not able to correct reproduce the flow locally along the ejector. The first attempt of application of computational fluid dynamics (CFD) techniques in ejector simulation were made in 1990s. For now the CFD becomes commonly applied tool to investigate and predict ejectors global operation for various operating conditions, understanding the complex local flow physics and recently becomes a natural part of designing or optimising procedure.

The studies on the ejector design and operation usually deal with the geometrical design of ejectors [6], the performance characterisation [7–10] or the analysis of some flow phenomena such as shock structure [10], shock interaction with turbulent boundary layer [11], mixing process [12,13] and condensation [7] for various ejector applications and various working fluids. CFD investigations are particularly limited by the computer capacity and the studies found in the literature on ejectors generally consider 2D or 2D axisymmetric computational domains. Only a few papers deal with 3D simulation, e.g. Ref. [14].

Bartosiewicz et al. [17] examined six turbulence models of $k-\varepsilon$, RNG $k-\varepsilon$, realizable $k-\varepsilon$, RSM and two types of $k-\omega$ model. All these studies correctly reproduced a general profile of internal flows, whereas the accuracy of computed internal shock wave was not satisfactory. Bouhanguel et al. [14] also examined the several RANS turbulence models in supersonic air ejector. Authors confirmed that the results depended on the of turbulence model selection [15,16]. Moreover, it seems that none of the RANS turbulence models tested is able to accurately capture the shock reflection pattern in the nozzle exit region. The study of turbulence models was also carried out by El-Behery and Hammed [18] in case of axi-symmetric diffuser. They have shown that the standard $k-\omega$, SST $k-\omega$ and v2-f models clearly performed better than other models when an adverse pressure gradient was present. The RSM model shows an

* Corresponding author. Tel. +48 58 6995 299.
E-mail addresses: d.butrym@pb.edu.pl, butrym@imp.gda.pl (D. Butrymowicz).

1290-0729/$ – see front matter © 2013 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.ijthermalsci.2013.11.009

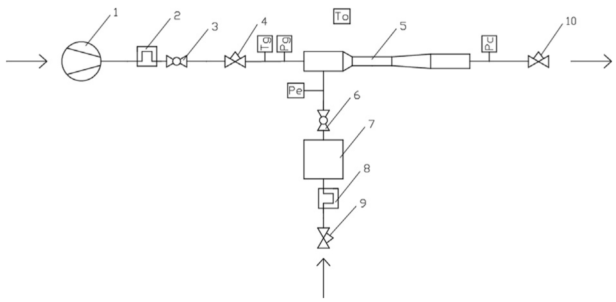

Fig. 1. Schematic of test ejector system: 1 – air compressor; 2, 8 – mass flowmeters; 3, 6 – cut-off valves; 4, 9 10 – control valves; 5 – tested ejector; 7 – seeding vessel.

acceptable agreement with the velocity and turbulent kinetic energy profiles but it fails to predict the location of separation and attachment points. The standard k–ε and the low-Re k–ε models give very poor results. Dvorak and Vit [13] compared the numerical results of static pressure at wall, velocity profile and turbulence intensity for most popular turbulence models with experimental results obtain by hot wire anemometry. On the basis of these results they suggested that in the case of analysed parameters the most promising turbulence model is realizable k–ε.

The comparison of numerical results with flow visualisation techniques seems to be a good approach in order to find the best numerical method for a solution of a flow in supersonic ejector. Kolar and Dvorak [19] verify the k–ω SST turbulence model by comparison with experimental colour Schlieren picture. They have found good agreement of shock wave prediction and boundary layer separation, but shear stress between stream is over-predicted. Ristić et al. [20] investigated the flow separation of flow in a supersonic nozzle. They compared the numerical results with experimental flow visualisation by the shadow method, Schlieren picture and the holographic interferometry. Five grids of different cells number and two turbulence models, namely k–ω and Spalart–Allmaras, were used in numerical calculations. Authors have discussed the results, however without recommendation on the selection of the turbulence model. Schlieren technique was also used to study the instability at the entrance part of the mixing chamber [21,22]. Results of flow visualisation in supersonic ejector were presented by Bouhanguel et al. [23]. These visualisation techniques use the laser sheet method and enables the investigation of specific phenomena such as shock structure, flow instabilities, and mixing process. Three zones of flow field area were studied: downstream region of the primary nozzle exit, mixing chamber (middle region), end of the mixing chamber and entrance to the diffuser. The results of this visualisation were used for comparison of the flow field to validation of the turbulence models by the same authors in Ref. [14]. Bouhanguel et al. [24,25] presented also the first results of velocity measurements using PIV technique obtained on a supersonic air ejector. The authors compared the PIV and CFD velocity field of tested ejector and also axial velocity obtained for various motive pressure founding good agreement between results. This study shows that the results are consistent with the theory of supersonic flow in ejectors, mainly for the shock train theory.

Since there are not clear recommendations on the selection of the turbulence model that are based on the experimental investigations then modelling of the supersonic ejectors is still an

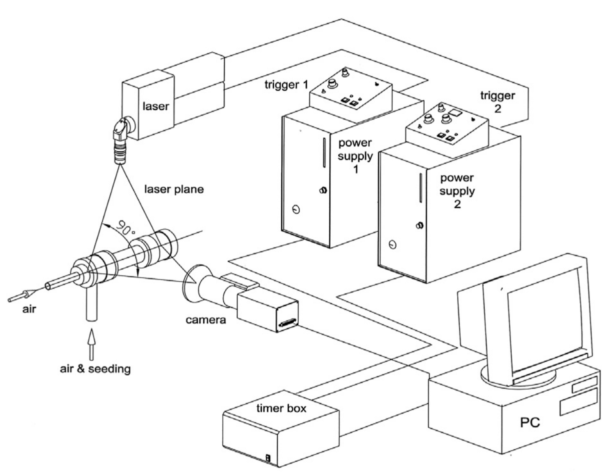

Fig. 2. Schematic of the test bench.

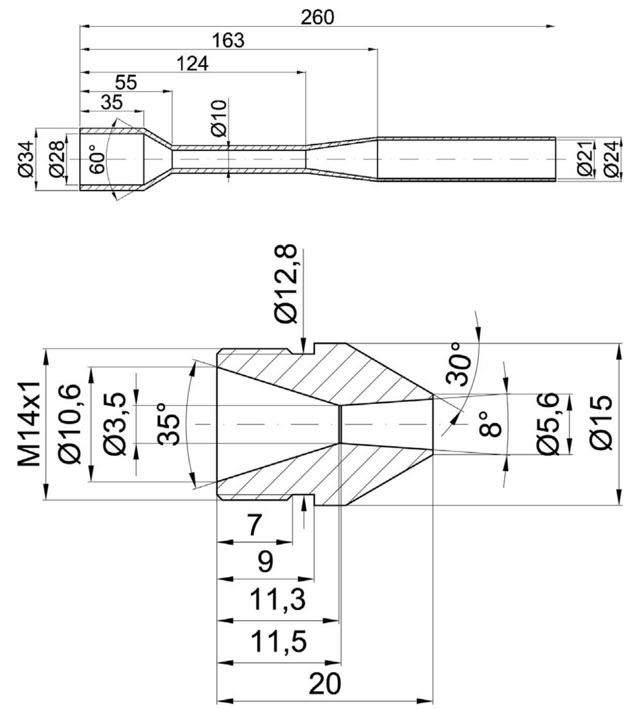

Fig. 3. Geometry of tested ejector and motive nozzle.

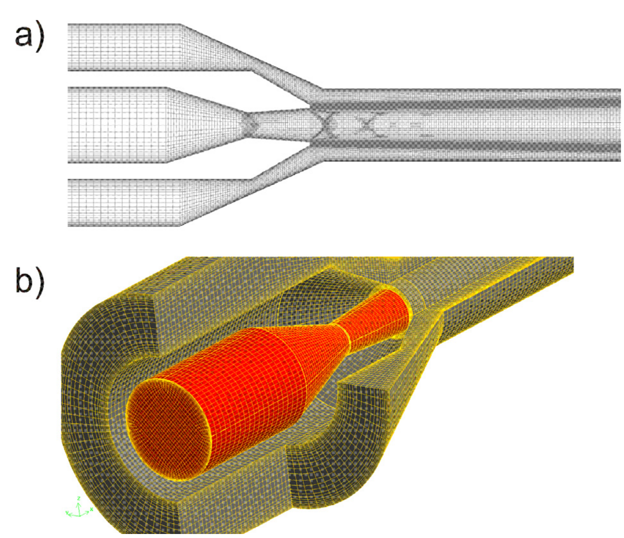

Fig. 4. Calculation grids of tested ejector: a) 2D model; b) 3D model.