Applied Thermal Engineering 93 (2016) 939–946

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

A novel ejector with a bypass to enhance the performance

Weixiong Chen, Huiqiang Chen, Chaoyin Shi, Kangkang Xue, Daotong Chong *, Junjie Yan

State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China

HIGHLIGHTS

- A novel ejector with bypass is proposed to enhance the ejector performance.
- There is an optimal bypass inlet position to obtain the maximum entrainment ratio.
- The ejector performance with bypass installed in the optimal position is analyzed.

ARTICLE INFO

Article history:
Received 13 July 2015
Accepted 7 October 2015
Available online 23 October 2015

Keywords:
Novel ejector
Bypass inlet
Enhanced performance
Entrainment ratio

ABSTRACT

It is crucial to enhance the ejector capacity to widen its application in industry. A novel ejector with a bypass installed in the low pressure district is proposed to enhance the ejector capacity. The results show that there is an optimal position for the maximum entrainment ratio, which corresponds to the lowest pressure around the second shock position. When the ejector operates at design conditions (the primary/induced/back pressures are 0.3/0.02/0.08 MPa), the optimal length ratio is about 1.1. Then, the ejector performance with the optimal bypass is investigated under different operational parameters. It is found that the novel ejector has relatively high entrainment ratio and low critical back pressure compared to the ejector without bypass under the design condition. When the primary pressure is bigger than a certain value, the entrainment ratio would always improve as the primary or induced pressures increase.

1. Introduction

Ejectors are regarded as a promising technology, because they could be powered by the low-grade thermal energy, such as the waste pressure or waste heat from industrial processes and solar collectors [1]. Today, ejectors are widely applied in industrial fields to address the challenge of the low grade energy utilization, for example, refrigeration systems [1] and pressure boosters in the oil and gas industries [2,3]. Ejectors are usually used as a pressure booster device. The ejectors have many advantages compared to compressors, because they do not have moving parts, could operate in a reliable and economical manner. In summary, ejectors represent a type of economically feasible and environmentally friendly technology. However, the primary defect of an ejector is its relatively poor performance, which limits its applications. Therefore, the current renewed interest is focused on promoting the ejector performance.

Generally, the ejector could be designed and analyzed by two different mixing theoretical models: one is the constant-area mixing model, which assumes that mixing process of motive and induced streams occur at the mixing tube with constant area. The other model is the constant-pressure mixing model, which assumes that mixing process is at the same pressure [4,5]. The constant-pressure mixing model is more widely used due to its higher performance over the constant-area mixing model [6–9]. By now, the theoretical model set up by Huang et al. [6] usually applied as a basis model, which could accurately predict performance during the critical operation. Besides, several researchers introduced the Prandtl–Meyer fan flow [7] and Fanno flow [8] to modify the 1D model, these assumptions make the model more close to the actual flow inside the ejector. More recently, Chen et al. [9] established a theoretical model to represent the overall operation modes (critical/sub-critical mode). The above models could provide and analyze the ejector performance under different operational parameters and structural parameters. However, some length parameters which may significantly influence the ejector performance, including the primary nozzle exit position (NXP) and mixing tube length, are not involved in these theoretical models because of the one-dimensional hypothesis.

To improve the ejector performance, many different methods have been adopted, such as ejector geometrical parameter optimization and adjustable ejector structure. Generally known, the ejector performance is closely related with its structural factors, so many investigators do their best to obtain the optimal ejector structural

* Corresponding author. Tel.: +86 29 82665359; fax: +86 29 82665359.
E-mail address: dtchong@mail.xjtu.edu.cn (D. Chong).

http://dx.doi.org/10.1016/j.applthermaleng.2015.10.067
1359-4311/

parameters for the maximum entrainment ratio. Zhu et al. [10] studied the effect of NXP and mixing section convergence angle, they stated that the entrainment ratio could be relatively improved by 40% while NXP varies from 6 mm to −12 mm, and the increment is 26.6% while the convergence angle increases from 0° to 2.07°. Chen et al. [11] also analyzed the variation between NXP and ejector performance, the results showed that the entrainment ratio increased from 11.9% to 20.1%, the pressure ratio increased from 56.3% to 58.0%, when NXP varied from 0 to 3.6 mm. Not only does NXP have a significant influence on the ejector performance, the geometrical factor of mixing tube also has important role. Pianthong et al. [12] pointed out that the entrainment ratio was almost independent of the mixing tube length, but the critical back pressure was sensitive to the mixing tube length. Varga et al. [13] concluded the same conclusion. However, the results from Chen et al. [11] pointed out that the mixing tube length had great effect on both the critical back pressure and the entrainment ratio.

Ejector design is usually based on a desired working fluid and ejector performance, which is sensitive and depends on the operation conditions. Indeed, the well-designed ejector could only operate effectively when the operation conditions approach the design conditions. The traditional method to widen the ejector operation scope is based on the geometrical optimization. Recently, the adjustable-geometry ejector, considered as a potential method, was proposed to deal with this challenge [14]. It means that a spindle was located in the primary nozzle, and its location could be moved along with the axis direction. Varga's research team [15–17] performed a lot of research work about the movable spindle ejector numerically and experimentally. They analyzed the ejector performance using R152a and R600a as refrigerants, and their numerical results concluded that the spindle position had a great effect on entrainment ratio, especially for the low condenser pressure. The maximum increase of R600a ejector performance was high up to 177% when saturation generator and evaporator temperatures 80 °C and 10 °C, while the condenser pressure low up to 350 kPa [16]. Their experimental results stated that there was an 70% increment of COP in comparison to that of a fixed-geometry R600a ejector, when the spindle position was 4 mm, the generator, evaporator and condenser saturation temperatures were 83 °C, 9 °C and 20 °C, respectively [17]. Chen et al. [18] carried out the experimental study on adjustable R134a ejector, and they indicated that the pressure recovery could be increased through a reasonable rise of the primary pressure.

In a word, it is clear that the geometrical factors have an important role on entrainment ratio, but the optimal structural size varies with the operation parameters, and there are still no commonly optimal values for different conditions. The ejector only operates with high entrainment ratio in a narrow operation conditions, but the adjustable-ejector could deal with this problem, and its weakness is that the induced mass flow rate also decreases under most of the working conditions although its entrainment ratio increases. In the past few years, the CFD method has been widely adopted to simulate the flow field and obtain the optimal geometrical factors. Then, researchers could capture the first and second series of shock trains phenomena [19]. They proposed the concept "critical section" to evaluate the entrainment ratio [11,19,20], "the second shock position" to judge the critical back pressure [11,19,21], and obtained a good result. Authors also analyzed the flow field carefully, and always found that there is low pressure area around diffuser inlet, which results from the second series of shock trains. Therefore, the authors propose to increase the entrainment ratio by adding a bypass inlet.

In the present studies, the ejector with bypass would be investigated, the numerical method verified by the experiment would be adopted to analyze the performance. Meanwhile, the effect of inlet location and operation factors is performed.

2. Ejector model and its verification

2.1. Modeling approach

When the CFD method is adopted to simulate the ejector internal flow field, the calculation process could be decomposed into two steps: the first step is to establish the flow domain to represent the flow of the actual situation, and the next is to set up mathematical model to solve. In the present study, Gambit 2.4, commercial software, is applied to deal with the first step. Fluent 6.3 is used to handle with the mathematical process.

In our study, the ejector is simulated in a 3D domain, the calculation regions are shown in Fig. 1, and a half of the ejector is chosen to represent the whole flow regions. The grid independence has been checked. The grid elements are about 2,521,980, which is sufficient to ensure independence and accuracy of the calculated results.

Fluent is used to deal with the 3D steady-state turbulent flow, the fluid flow processes could be described using the mass, momentum and energy conservation equations [22]. The convective terms are discretized by 2nd order upwind scheme. The RNG k–ε model is used to represent the supersonic flow inside the ejector, and has a better performance predictor compared with the other turbulence models [10,11]. The SIMPLEC method is taken to deal with the pressure field. Aiming to resolve the flow characteristic adjacent to the ejector wall, the standard wall function is selected.

The working fluid used in the present study is air, which is treated as ideal gas. Its density is adopted to obey the ideal gas law, while the other properties are given fixed values. The primary/induced flow inlets are defined as "pressure inlet" conditions, while the mixing flow outlet is set as the "pressure outlet" condition. The whole wall surfaces are treated as the adiabatic walls.

Moreover, the ejector performance could be solved as mentioned above, while the convergence is obtained for each calculation process. In the present study, the residue of continuity equation is less than 10^{-4}, as well as turbulent kinetic energy and turbulent energy dissipation rate. The residue of momentum equations is smaller than 10^{-5}, so is the energy equations. Besides, the mass flow rate at the diffuser outlet is almost constant.

2.2. Validation by experimental results

An experiment rig has been built up to study ejector performance in indoor laboratory, as shown in Fig. 2. The experiment system mainly includes compressor, dryer, gas storage tanks, electric valve, ejector, data acquisition system and the measuring system. The pressurized air is continuously provided by air compressor, which flows into the high-pressure vessel and the low-pressure vessel. The air stream from the high pressure vessel, named as primary flow,

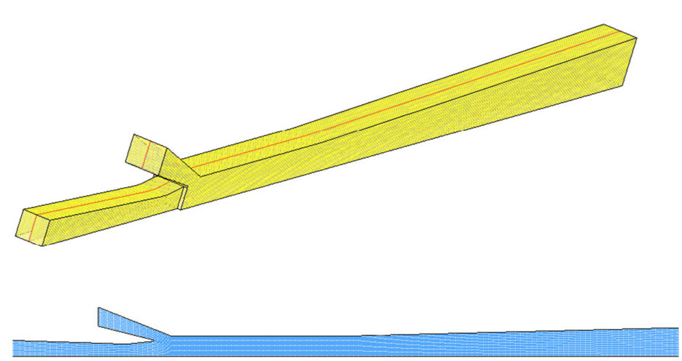

Fig. 1. Calculation domain and grid structure of ejector. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)