Study of the Supersonic Ejector-Diffuser System with a Mixing Guide Vane at the Inlet of Secondary Stream

Fanshi Kong † • Vincent Lijia † • Heuy Dong Kim ‡ and Yingzi Jin **

Abstract

Ejector-diffuser system has long been used in many diverse fields of engineering applications and it has advantages over other fluid machinery, because of no moving parts and structural simplicity. This system makes use of high-pressure primary stream to entrain the low-pressure secondary stream through pure shear actions between two streams. In general, the flow field in the ejector-diffuser system is highly complicated due to turbulent mixing, compressibility effects and sometimes flow unsteadiness. A fatal drawback of the ejector system is in its low efficiency. Many works have been done to improve the performance of the ejector system, but not yet satisfactory, compared with that of other fluid machinery. In the present study, a mixing guide vane was installed at the inlet of the secondary stream for the purpose of the performance improvement of the ejector system. A CFD method has been applied to simulate the supersonic flows inside the ejector-diffuser system. The present results obtained were validated with existing experimental data. The mixing guide vane effects are discussed in terms of the entrainment ratio, total pressure loss as well as pressure recovery.

Key Words: Ejector-Diffuser System, Mixing Guide Vane, Shock Wave, Compressible Flow, Supersonic Flow

1. INTRODUCTION

Ejector-diffuser system makes use of high-pressure primary stream to entrain the low-pressure secondary stream through pure shear actions between two streams. The high-pressure primary stream, mainly discharged from a supersonic nozzle, drags the secondary stream into the diffuser, where the kinetic energy of the mixed stream is converted to pressure energy. This system has many advantages over other fluid machinery, such as no moving parts and structural simplicity. In addition of directly increasing the pressure without the input of mechanical energy, the ejector system can be used in such industrial applications as ejector refrigeration or seawater desalination [1].

A fatal drawback of the ejector system is in its low efficiency. For many years, researchers have tried to describe the phenomena of ejector flow in order to achieve a high performance of ejector. Hong et al. [2] tried to improve the efficiency of the ejector by reducing the pressure loss during mixing. Based on this idea, Chang & Chen [3, 4] have developed a new nozzle and found that entrainment ratio and critical back pressure can be higher. Recently, Computational Fluid Dynamics (CFD) method has been extensively used as a powerful technique for simulating and analyzing flow. Many works have been done to improve the performance of the ejector system, especially the geometry modification of the jet ejector, but not yet satisfactory, compared with that of other fluid machinery [5, 6].

Manohar [7] made an experiment work of a jet ejector with several mixing guide vanes installed at the inlet of the secondary stream. The experimental results showed that mixing guide vanes had a good effect in the pressure recovery of the ejector system [8]. In the present study, a CFD method has been applied to simulate the supersonic flows inside the ejector-diffuser system. A mixing guide vane was installed at the inlet of the secondary stream of the ejector-diffuser system, as schematically shown in Fig. 1. The present results obtained were validated with existing experimental data.

---

† Corresponding author, E-mail: kimhd@andong.ac.kr
‡ School of Mechanical Engineering, Andong National University
** Zhejiang Sci-Tech University

Fig. 1 Supersonic ejector-diffuser system
2. NUMERICAL SIMULATION

2.1 Computational flow model

The geometry of jet ejector physical model was built to be exactly the same with the experimental apparatus [7]. Similar model was used of an ejector without mixing guide vane in Somsak's study [9], which was a constant-area ejector and the working fluid was air. The diameter of mixing section is 6.08 mm (Dₖ); the diameter of mixing section is 97.79 mm (D₈). The computational flow model of supersonic ejector-diffuser system with the mixing guide vane is shown in Fig. 2.

![Fig. 2 Schematics of supersonic ejector-diffuser system](image)

Fig. 2 Schematics of supersonic ejector-diffuser system

![Fig. 3 Geometry of the mixing guide vane](image)

Fig. 3 Geometry of the mixing guide vane

The geometry of the mixing guide vane is illustrated in Fig. 3. It is not just a cylindrical pipe, and the diameter of the inlet and outlet of the mixing guide vane is different (D₁ > D₂). It was built as a circular truncated cone with a taper ratio of 0.011.

2.2 Computational domain

The computational mesh was generated in the grid-generating software ICEM. A structured mesh was employed in computations. Grids were densely clustered near the wall to capture the flow features in boundary layers.

A high quality mesh can provide accurate results and save the computational time. Several works to get grid-independent solutions have been done prior to the present study. The simulation result of different mesh sizes is summarized in Table 1. The deviation of results was defined as the difference of total mass flow rates between the CFD analysis and experiment results. The difference among 3 percentage deviations in Table 1 was less than 3%, therefore grid independence of solutions was also checked.

<table>
<thead>
<tr>
<th>Mesh Elements</th>
<th>Total mass flow rates (CFD)</th>
<th>Total mass flow rates (Exp)</th>
<th>Percentage Deviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>52,381</td>
<td>0.401 Kg/s</td>
<td>0.39 Kg/s</td>
<td>2.80 %</td>
</tr>
<tr>
<td>73,086</td>
<td>0.380 Kg/s</td>
<td>0.39 Kg/s</td>
<td>2.60 %</td>
</tr>
<tr>
<td>114,761</td>
<td>0.379 Kg/s</td>
<td></td>
<td>2.81 %</td>
</tr>
</tbody>
</table>

Ejector outlet was extended to stabilize the pressure outlet boundary conditions, so that more accurate results can be obtained. The boundary condition of primary stream was specified at the exit of supersonic nozzle.

2.3 CFD analysis

Commercial software ANSYS Fluent 13.0 was used for the Computational Fluid Dynamics simulations. Working fluids of both primary stream and secondary stream in the ejector-diffuser system were treated as ideal gas. Two-dimensional axial symmetric model, density-based solver, with the standard k-ω turbulent model was used in the computation. Governing equations were discretized spatially with a finite volume scheme. Third-order MUSCL was used in turbulent kinetic energy as well as spatial discretizations.

Total pressure boundary conditions were used at the primary stream inlet. The flow models were simulated for 5 different initial Mach numbers (Me) at the primary stream inlet from 1.20 to 1.66, as summarized in Table 2. The initial values can be calculated in these equations:

\[
\frac{P_0}{P_\infty} = \left(1 + \frac{\gamma - 1}{2} Me^2\right)^{\frac{\gamma}{\gamma - 1}} \\
Me = \frac{V}{\sqrt{R \cdot T}} \\
m = \rho V A
\]

where,

- \(P_0\): static pressure, Pa
- \(P_\infty\): total pressure, Pa
- \(\gamma\): specific heat ratio
- \(Me\): Mach number at nozzle exit
- \(V\): velocity, m/s
- \(R\): gas constant, J/Kg*K
- \(T\): temperature, K
- \(A\): area, m²
- \(m\): mass flow rate, Kg/s

<table>
<thead>
<tr>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
<th>Case 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>V (m/s)</td>
<td>411</td>
<td>449</td>
<td>490</td>
<td>528</td>
</tr>
<tr>
<td>Me</td>
<td>1.20</td>
<td>1.30</td>
<td>1.43</td>
<td>1.54</td>
</tr>
<tr>
<td>Total Pressure (KPa)</td>
<td>253</td>
<td>295</td>
<td>350</td>
<td>412</td>
</tr>
</tbody>
</table>

Mass flow rate boundary conditions were used at the secondary stream inlet. Pressure outlet boundary
conditions were specified at the outlet of ejector-diffuser system.

3. RESULTS AND DISCUSSION

From the results of the simulation analysis, higher pressure recovery increment and lower entrainment ratio decrement can be found in case 5 with a Mach number of 1.66 from Fig. 8 and Fig. 9. The following discussions are based on the CFD results of case 5.

Fig. 4 shows the contours of static pressure around the primary stream inlet for case 5. The difference of the static pressure distribution between two models is illustrated in Fig. 7 while $P_{s1}$ is described as the static pressure at nozzle exit. The model with mixing guide vane achieves a higher static pressure than another model after shock wave. That's one of evidences prove that the new model gets a higher pressure recovery after final promotion of diffuser. Fig. 5 shows the contours of Mach number around the inlet of ejector-diffuser system; Fig. 6 shows the contours of density around the inlet of ejector-diffuser system.

![Fig. 4 Contours of static pressure (case 5)]

![Fig. 5 Contours of Mach number (case 5)]

![Fig. 6 Contours of density (case 5)]

![Fig. 7 Distribution of static pressures along ejector axis (case 5)]

3.1 Entrainment ratio

Many kinds of flow characteristics can directly affect the performance of the ejector-diffuser system. Entrainment ratio ($R_{en}$) is one of the most important parameters, which can be influenced by geometry, back pressure as well as other operating conditions. As the productive capacity of primary stream, the entrainment ratio ($R_{en}$) can be represented as the following equation:

$$R_{en} = \frac{mass \ flow \ rate \ of \ secondary \ flow}{mass \ flow \ rate \ of \ primary \ flow} \quad -(4)$$

In this study, the entrainment ratio for different Mach numbers of primary stream is shown in Fig. 8. It is found that the entrainment ratio has a trend of decreasing when the Mach number was increased from 1.20 to 1.66.

The ejector-diffuser system without mixing guide vane shows better results of entrainment ratio in both experiment results and CFD analysis. The percentage decrease of entrainment ratio is about 6.75%. Under the mixing guide vane influence, the area of the mixing section and the productive capacity of primary stream were changed; the entrainment ratio were less than the model without mixing guide in all 5 cases.

3.2 Pressure recovery

Pressure recovery ($\Delta P$) can be defined as the difference between static pressure at the secondary stream inlet ($P_{s1}$) and static pressure at the outlet of ejector-diffuser system ($P_{out}$). The pressure recovery can express the operational capability of ejector-diffuser system visually. The Pressure recovery ($\Delta P$) can be represented as the following equation:

$$\Delta P = P_{s1} - P_{out}$$

$\quad -(5)$
Fig. 9 shows the pressure recovery of the ejector-diffuser system for different cases. The results show that the experiment results and CFD analysis have a similar trend of increasing, and the average percentage deviation of the model with mixing guide is 1.2 %; the model without mixing guide vane has an average percentage deviation of 3.8 %, but still reflects the variation of pressure recovery approximately.

The ejector-diffuser system with mixing guide vane shows better pressure recovery. The increment of pressure recovery between two models is about 14.9%. The diffuser in ejector-diffuser system plays a major role in the pressure recovery. The geometry of the diffuser in two cases is same while the mixing section is different. With a mixing guide vane, more vertical flow is introduced into the stream, which helps to improve the pressure recovery. Hence, addition of a mixing guide vane helps to mix the primary & secondary stream in the constant-area mixing section.

3.3 Total pressure loss

Total pressure loss ($P_{01} - P_{out}$) can be considered as the difference between total pressure at the nozzle exit ($P_{01}$) and total pressure at the outlet of ejector-diffuser system ($P_{out}$).

Fig. 10 shows the total pressure loss of the ejector-diffuser system for different Mach numbers of primary stream. The CFD analysis results show that both models have an almost same trend of increasing, and the average percentage deviation is less than 0.2 %. Following the Mach number increased from 1.20 to 1.66, the performance improvement in total pressure loss of the ejector system with a mixing guide vane is not obvious as pressure recovery, more accurate and precise results should be obtained in the further works.

4. CONCLUSIONS

Computational studies were carried to investigate the supersonic ejector-diffuser system with a mixing guide vane at the inlet of secondary stream. The CFD results have been validated with existing experimental data. Results obtained from the computational analysis were similar with experiment results with a percentage deviation of 2.6%.

The numerical simulation results with and without mixing guide vane have been compared. The mixing guide vane effects have been discussed in influence of the performance improvement of the ejector system. The mixing guide vane caused the entrainment ratio to reduce 6.75%. However, while the mixing guide vane does not influence the total pressure loss, it leads to a better pressure recovery of ejector-diffuser system. Further work is going on to optimize the mixing guide vane.
REFERENCES


