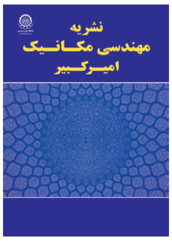

Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 52(11) (2021) 747-750
DOI: 10.22060/mej.2019.15146.6041

Design and Analysis of Gas Ejector in High Altitude Test Facility

E. Mohammadi[1], N. Fouladi[2*], A. Madadi[1]

[1] Department of Aerospace Engineering Department, Amirkabir University of Technology, Tehran, Iran
[2] SpaceTransportation Research Institute, Iranian Space Research Center, Tehran, Iran

ABSTRACT: Usually, ground testing of space engines is performed in a high altitude test facility. The facility is equipped with a supersonic diffuser that expels automatically engine gases to the atmospheric environment and maintains a vacuum pressure around its nozzle and motor. Normally; in the case of lower motor pressure, the supersonic flow in the diffuser could not be established. In this situation, an auxiliary ejector is employed frequently at the end section of the diffuser. In the present study, a new algorithm for designing the supersonic ejector has been proposed. Unlike conventional methods, this algorithm can be used with different primary and secondary fluids. The main design parameters are determined by the algorithm, while the secondary parameters are selected from the experimental test results of the existing references. A safe margin is considered for the safe operation of the ejector, which is predicted through the numerical simulations. Also, numerical simulation is used to verify the present design algorithm. Finally, using the proposed algorithm, an ejector is designed to reduce the minimum starting pressure of a second throat exhaust diffuser. An integrated simulation of the diffuser-ejector is performed and the appropriateness of the designed ejector is confirmed.

Review History:

Received: 2018/10/17
Revised: 2019/03/18
Accepted: 2019/05/05
Available Online: 2019/05/15

Keywords:

High altitude test facility
Supersonic ejector
Design algorithm
Numerical simulation

1- Introduction

Space engines that are designed to operate at high altitude levels frequently employ high expansion ratio nozzles to achieve best performance and maximum thrust. When such engines are tested at sea level, the accurate measurement of engine performance is not possible due to the massive flow separation at the divergent section of the nozzle. Therefore; High Altitude Test Facility (HATF) is required to simulate altitude conditions in the sea level environment. This system consists of a diffuser with or without an auxiliary ejector [1]. Fig. 1 schematically shows the central core of diffuser-ejector test facility. The system consists of supersonic diffuser, spray cooler and ejector. This study mainly focuses on ejector section.

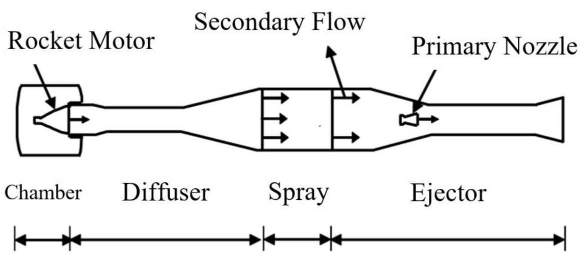

Fig. 1. Schematic of an altitude test facility [2]

The first and simplest analysis of an ejector has been proposed by Keenan and Neumann [3]. They used the simplest form of ejector, a constant-area ejector without a diffuser. They calculated the performance of ejector using the one-dimensional continuity, momentum and energy equations. In spite of some simplifications, their theoretical results were consistent and compared well with the experimental results. Munday and Bagster [4] suggested that two primary and secondary fluids flow maintain separated in some of their paths without mixing. Primary fluid by expanding through its path creates a converging path for secondary flow and makes it reach sound velocity at a given section. The mixing of two fluids begins after that section. Huang et al. [5] calculated the cross section area of ejector using Munday and Bagster method. They postulated that mixing in constant pressure takes place in constant area section of ejector. Emanuel [6] proposed a simple analytical model for optimizing the performance of an ejector. This model requires the entrainment ratio to be less than unity. The novel approach here was to express the ejector model in term of enthalpy and therefore can be used to different primary and secondary fluids.

The main purpose of the present research is to develop a new algorithm for designing a supersonic ejector in high altitude test facility application. Using this method gives us the ability to design ejectors for reducing the minimum starting pressure of the diffuser in ground testing of low combustion pressures engines.

2- Methodology

The main assumption in ejector modeling is the mixing of fluids, which takes place in constant pressure and in constant area region of the ejector. High pressure primary fluid expands with supersonic speed out of the primary nozzle and creates a

*Corresponding author's email: n.fouladi@isrc.ac.ir

very low pressure region in that region. This makes secondary fluid to be entrained to the mixing chamber. Expanded primary flow entrains secondary fluid in a converging path through mixing chamber. At some section along the ejector, the speed of secondary fluid rises to sonic velocity and the choking condition is reached. There is no mixing up to here, so gas dynamic relations are used separately for primary and secondary fluids. After this cross-section two flow streams are mixed completely in constant pressure. After that there is one single mixed fluid with new thermodynamic properties. Then a normal shock is assumed to be induced in flow. Further compression of flow to atmosphere pressure is achieved through divergent section of the ejector. Non-isentropic processes inside flow are modeled through considering different efficiency factors. Accurate efficiency factors must be measured by experiments. Considering some required changes in Huang et al. [5] algorithm, it is necessary to develop and reconstruct the governing formulation to different primary (usually water vapor) and secondary (engine exit gases) fluids. In this research, a new algorithm is developed to extend the Huang algorithm for designing the supersonic ejector for HATF applications. The detailed explanations of the proposed algorithm are given in the full length paper.

3- Results and Discussion

To validate the design algorithm, it is necessary to have the exact geometric features of an existing ejector. Unfortunately, such information is not available in existing resources. Therefore, the numerical simulation technique is used to verify the design algorithm. For confirmation of the utilized numerical simulation method, the comparison with experimental results is accomplished. To do this, the experimental work of Sriveerakul et al. [7] is considered. Static pressure measurement for this ejector was carried out on the ejector wall from the nozzle outlet to the diffuser output (see Fig. 2). As seen in the Fig. 2, the most correspondence between numerical results and experimental results is obtained for the kω-SST turbulence model. Therefore, this turbulence model is used for all of the numerical analysis in the present research.

At this step, the design of an ejector was performed to reduce the minimum starting pressure of an available diffuser to ground testing of an engine with lower combustion pressure. According to numerical simulations, the available diffuser minimum starting pressure is 23 bar. However, the engine combustion pressure is 11.5 bar. Therefore, the current task is to design an ejector that integrated diffuser-ejector assembly can be started at the combustion pressure of 11.5 bar.

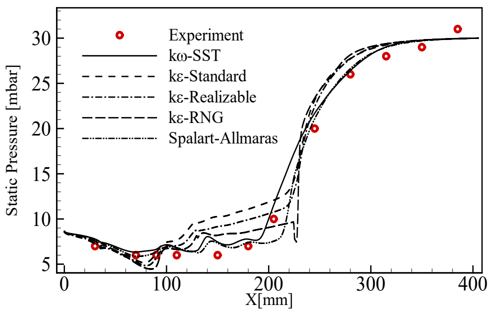

Fig. 2. Comparison of static pressure on the ejector wall for some turbulence models and experiment data

Fig. 3 shows contours of Mach number for integrated assembly in which the engine is OFF and ejector is ON. Since the engine is OFF and its flow-rate is zero, the ejector exit flow expands to the ejector wall and isolates the diffuser internal region from atmospheric environment. In practice, this technique is frequently used to pre-evacuation of the diffuser before starting the engine. With pre-evacuation, the diffuser reaches to starting condition at smaller starting time (see Fouladi et al. 2018 [8]).

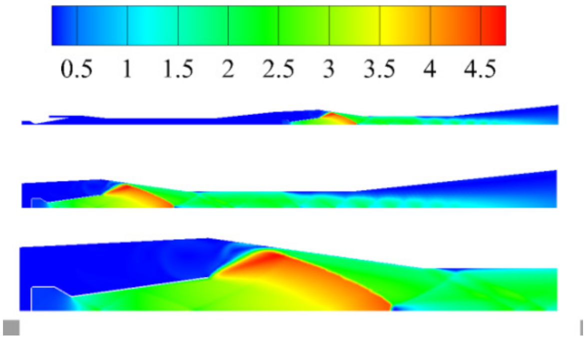

Fig. 3. Mach contour for engine OFF and ejector ON

Fig. 4 shows contours of Mach number for integrated assembly in which ejector is OFF and motor combustion pressure is set to 11.5 bar. Low Mach number at nozzle exit plane is originated from enormous flow separation at the divergent section of the nozzle and subsequently the diffuser is not started.

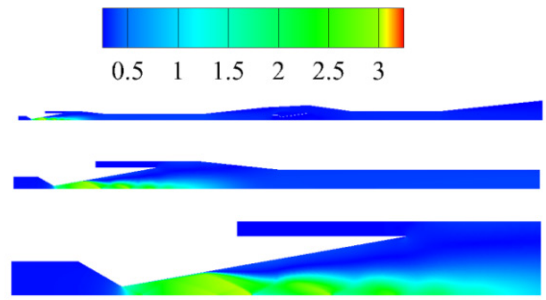

Fig. 4. Mach contour for engine ON and ejector OFF

Fig. 5 shows contours of Mach number for integrated assembly in which both engine and ejector are in operating mode. The engine combustion pressure is 11.5 bar. Expanding engine jet up to the diffuser wall is a good sign to a started diffuser. As seen, both ejector and diffuser operate well in their started modes.

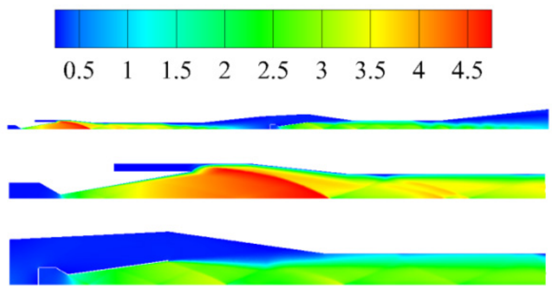

Fig. 5. Mach contour for both engine and ejector ON

4- Conclusion

In the present work, a design algorithm was proposed to design a gas ejector suitable for high altitude test facilities. One of the features that distinguish this algorithm from similar one is its extended formulation for different primary and secondary fluids. A numerical simulation technique was used to validate design process. Using the developed code and one already available diffuser, an ejector was designed to reduce diffuser minimum starting pressure to half value. The verification of current design algorithm was accomplished by numerical simulation of integrated diffuser-ejector assembly.

5- References

[1] P. Sreekireddy, T.K.K. Reddy, V. Dadi, P. Bhramara, CFD Simulation of Steam Ejector System in High Altitude Test (HAT) Facility, in: ASME 2012 Gas Turbine India Conference, American Society of Mechanical Engineers, 2012, pp. 149-157.

[2] R. Manikanda Kumaran, T. Sundararajan, D. Raja Manohar, Simulations of high altitude tests for large area ratio rocket motors, AIAA journal, 51(2) (2012) 433-443.

[3] J. Keenan, E. Neumann, A simple air ejector, ASME J. Appl. Mech, 9(2) (1942) A75-A81.

[4] J. Munday, D. Bagster, The choking phenomena in ejector with particular reference to steam jet refrigeration, in: Thermal Fluids Conference of Inst, of Eng. Australia, 1976, pp. 84-88.

[5] B. Huang, J. Chang, C. Wang, V. Petrenko, A 1-D analysis of ejector performance, International journal of refrigeration, 22(5) (1999) 354-364.

[6] G. Emanuel, Optimum performance for a single-stage gaseous ejector, AIAA Journal, 14(9) (1976) 1292-1296.

[7] T. Sriveerakul, S. Aphornratana, K. Chunnanond, Performance prediction of steam ejector using computational fluid dynamics: Part 1. Validation of the CFD results, International Journal of Thermal Sciences, 46(8) (2007) 812-822.

[8] N. Fouladi, A. Mohamadi, H. Rezaei, Numerical investigation of pre-evacuation influences of second throat exhaust diffuser, Fluid Mechanics and Aerodynamics, 5(2) (2017) 55–69.