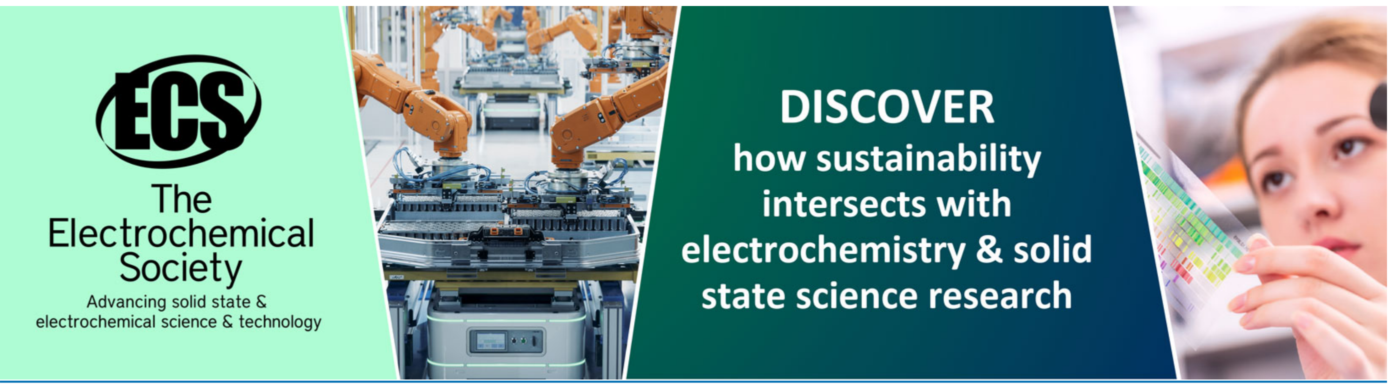
PAPER • **OPEN ACCESS**

Study on steam pressure characteristics in various types of nozzles

To cite this article: Firman and Muhammad Anshar 2018 *J. Phys.: Conf. Ser.* **979** 012084



Study on steam pressure characteristics in various types of nozzles

Firman, Muhammad Anshar

Mechanical Engineering Department, Politeknik Negeri Ujung Pandang, Makassar, Indonesia 90245

E-mail: firman@poliupg.ac.id

Abstract. Steam Jet Refrigeration (SJR) is one of the most widely applied technologies in the industry. The SJR system was utilizes residual steam from the steam generator and then flowed through the nozzle to a tank that was containing liquid. The nozzle converts the pressure energy into kinetic energy. Thus, it can evaporate the liquid briefly and release it to the condenser. The chilled water, was produced from the condenser, can be used to cool the product through a heat transfer process. This research aims to study the characteristics of vapor pressure in different types of nozzles using a simulation. The Simulation was performed using ANSYS FLUENT software for nozzle types such as convergent, convrgent-parallel, and convergent-divergent. The results of this study was presented the visualization of pressure in nozzles and was been validated with experiment data.

1. Introduction

Steam injector refrigeration system can replace vapor compression of refrigeration system. In general, the refrigerant thermodynamic characteristics of the two systems are the same. The fact indicates that only water that has been used so far and available resources to drive injectors. Some research has been done on the use of fluorocarbon in vapor compression systems. These systems use water as a refrigerant in vapor compression cycle and every compression obtained from compression jet principles, known as refrigeration water-steam or steam-injector refrigeration (SJR) system.

SJR system utilizes remaining steam of boilers that is then channeled through nozzles into a tank (flash tanks) containing water. The nozzles convert pressure energy into kinetic energy so that it can quickly evaporate water. Cold water resulting from condensers (chilled water) is used to cool products through heat transfer. Generally, evaporating 1% of water in the tank can reduce 6^{o}C of water temperature [1].

SJR system can be operated at the boiler temperature of 120^{o}C up to 140^{o}C and evaporator temperature of 5^{o}C up to 15^{o}C [2]. The temperature of evaporators depends on the pressure of evaporators. Low pressure or vacuum condition of evaporators is highly dependent on the design of injectors. According to [3], one of the factors that affect vacuum pressure of evaporators is the geometry between water surface and the suction side of injectors. It is therefore required that an injector be designed properly for the applications of refrigeration steam jet systems. The design of optimum angle of steam injector inclination is 2^o for convergent type and 3^o for divergent type with a throat length of 137 mm [4]. However, geometry factor causes the efficiency or performance (COP) of steam jet system refrigeration is still very low.

A quick evaporation of 1 kg of water results in a decline in temperatures of 5, 7°C [1]. If such a condition persists, the temperature of the water will get lower and lower. This should be allowed to take place to achieve a desired water temperature. The shortage of SJR refrigeration is associated with the fact that water freezes at the temperature of 0°C; if water freezes, the circulation in the system will not occur. Accordingly, the SJR refrigeration applications at low temperatures is very limited.

The performance of SJR system relies heavily on the efficiency of nozzles. Meanwhile, the efficiency of nozzles are influenced by geometry and fluid pressure in the injector. Sahni [5] reported that a the COP of steam jet refrigeration system is influenced by the geometry of nozzles and pressure drop.

According to Mitchley [3], one of the influential factors in the design of injector is geometry between water surface and the suction side of injector. On the output side the parameter that affects the efficiency of injector is pressure drop [5]. Other factors to consider are critical pressure and shock waves on the injector [6]. The ratio between intake pressure and critical pressure also affects the efficiency of injector and depends on geometry [7]. Vadalia [8] states that the optimization of nozzle geometry (throoat diameter) is very influential towards the performance of steam jet injector. Petel [9] conducted research on the geometry optimization of injector in order to obtain the best efficiency by computational fluid dynamics (CFD) analysis and by the reduction of the pressure drop through the geometry injector.

2. Materials and methods

In this experiment there are 4 types of nozzle analyzed theoretically by simulations of convergent, parallel-convergent, 1° inclination-angle convergent-divergent, and 3° inclination-angle convergent-divergent nozzles. However, only the simulation of 1° inclination-angle convergent-divergent nozzles which were validated by the data of experiments.

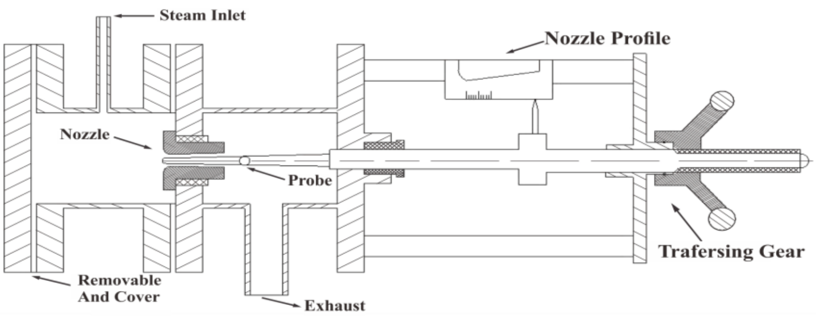


Figure 1. Experiment scheme.

The experiments were begun by placing a 1° inclination-angle convergent-divergent on certain positions, as shown at figure 1, then followed by setting the intake pressure of nozzle as much as 5.4 bars, which was maintained constant. Furthermore, the profile arm of nozzles was arranged to read the pressure at each point along a range of 5 mm. The exit pressure obtained from the nozzles in the experiment was 2 bars. The data of experiment were used as the boundary condition in the simulation as shown in figure 2.

To view the condition of fluid inside the nozzle visually with the simulations, ANSYS 16.0 with an system analysis feature FLUENT was used. The simulation steps were, firstly, drawing CAD nozzle by using solid works. Secondly, the CAD nozzle was imported to the ANSYS by using the geometric features in the ANSYS. Thirdly, the geometry of nozzle was imported to the mesh. Fourthly, the mesh was imported to the setup then to set solver based on pressure and steady state of problems in time types; then the energy equation was activated and the inlet and outlet pressures in the boundary conditions were set up. Finally, iterative solver and mixed initialization were selected and the program was run.

Figure 2. Boundary Condition.

3. Results and discussion

The use of ANSYS 16.0 CFD with the feature of simulation system FLUENT in ANSYS 16.0 was applied in to order to simulate the flows of fluid in the form of nozzle geometry in the experiment. In the experiment the inlet and outlet sides of the nozzles were the parameter derived from the results of laboratory testing, which were 5.5 bars and 2 bars on the inlet side and outlet side respectively. From figure 12, it can be seen that pressure graph on each wall of the nozzle with pressure changes at each position.

To see the characteristics of each varied nozzle, various step models were validated in accordance with the data of experiment. The contour parameters shown in figure 3, 5, 7 and 9 were found after inputting the inlet and outlet pressures on the 1° inclination-angle convergent-divergent, 3° inclination-angle convergent-divergent, convergent, and divergent-parallel nozzle models by using the method of fluent simulation. To compare the results of laboratory experiments, the results of simulations using ASNYS 16.0 can be seen below.

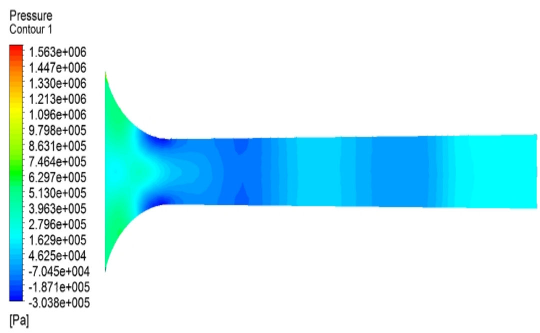


Figure 3. Contour of 1° inclination angle convergent-divergent Nozzle.

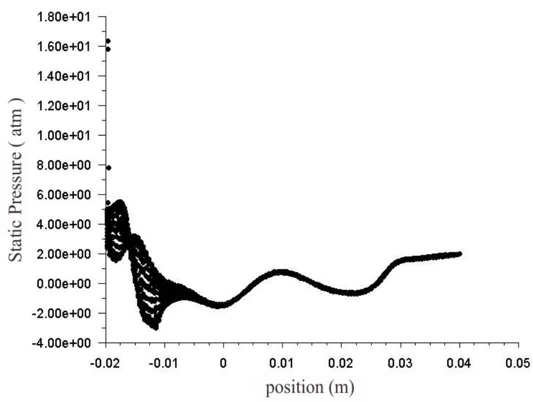


Figure 4. Pressure Graph of 1° inclination angle convergent-divergent Nozzle.

On the 1° inclination-angle convergent-divergent the pressure will decrease at when its position was after 30 mm as shown in figure 4. As shown in figure 3, the pressures began to go down after shocks took place. This is in line with the results of the study of Pansari [10] that the decrease moved from the shock location headed the exit side.

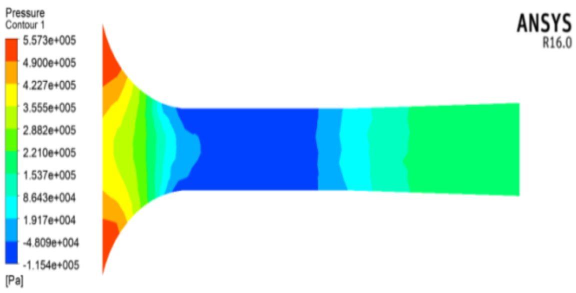


Figure 5. Contour Pressure of 3° inclination angle convergent-divergent Nozzle.

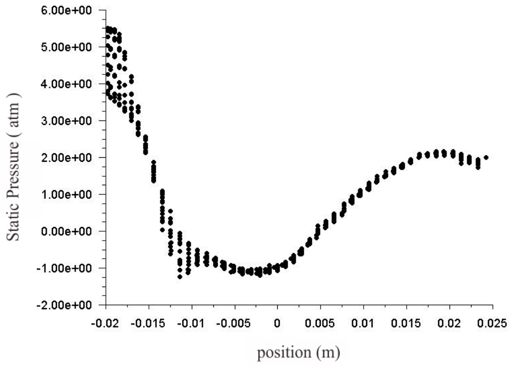


Figure 6. Pressure Graph of 3° inclination angle convergent-divergent Nozzle.

On the contrary, the 3° inclination-angle convergent-divergent underwent pressure decline when its position reached 20 mm; the differences were also due to the angle of inclination. The influence of inclination angle is accorded to the research results of Surya [11] claiming that inclination angle affected the performance of nozzles.

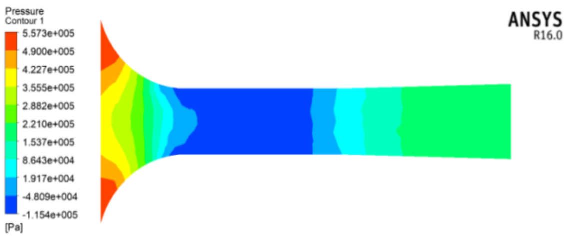


Figure 7. Contour pressure of convergent nozzle.

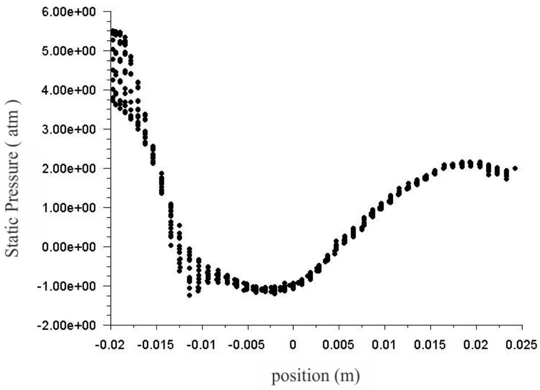


Figure 8. Pressure graph of convergent nozzle.

The characteristics of pressure on convergent nozzles were almost the same with those of convergent-parallel nozzle where the decrease of pressure occurred when they reached the same position. The decrease of pressure happened because the flow of fluid in the nozzles of both types underwent a sudden change of pressure on the exit sides of such nozzles. These conditions were in line with the research results of Rao [12] that analyzed flows in convergent nozzles using the CFD for the ratio of different nozzles.

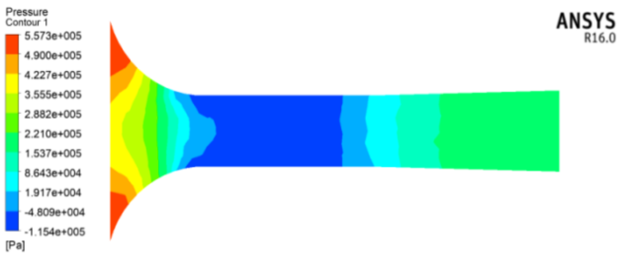


Figure 9. Contour pressure of convergent-parallel nozzle.

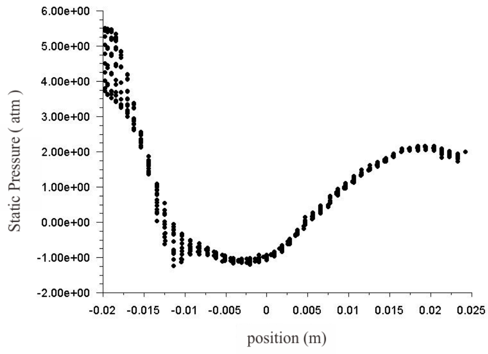


Figure 10. Pressure graph of convergent-parallel nozzle.

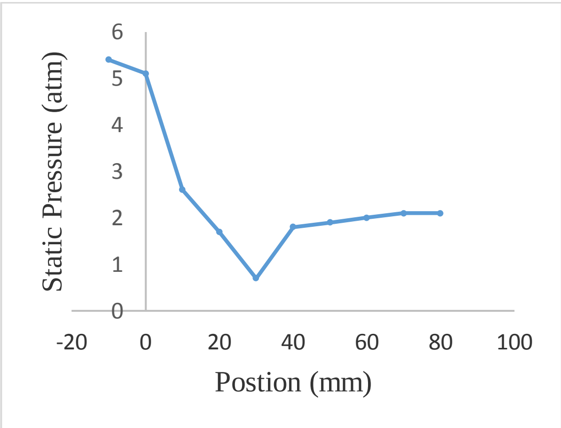


Figure 11. Graph of experiment results.

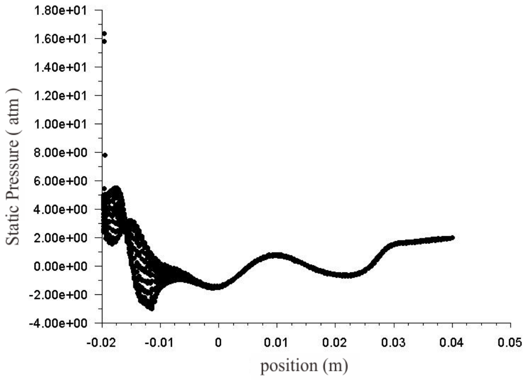


Figure 12. Graph of simulation results.

The graph in figure 11, the results of the experiment, demonstrated the suitability of characteristics with those of the simulation in figure 12. The contours of pressure as in figure 3 up to figure 12 show the variation of pressure along the walls of nozzles; the pressures increased at inlet sides but decreased at outlet sides. The analysis results were in line with the view of Satyanarayana [13] that CFD analysis on convergent-divergent nozzle with rectangular, square, and round surfaces showed the result of the computation was very suitable with the experimental results. Sudhakar [14] has also made such a modeling with theoretical analysis by using CFD with parameters of pressure, temperature, and flow speed that resulted in a suitability of flow characteristics in convergent-divergent nozzles. Similarly, the suitability was demonstrated by the modeling of fluid flow in convergent-divergent nozzles assuming that the Quasi-one of dimensional isentropis flows using CFD [15].

4. Conclusion

Based on the results of experiments by simulations, it was found that various pressures occurred along the nozzles. The characteristic analysis on pressures in every type of nozzle showed different results; and this was subject to the difference of geometry. It was concluded that analysis using FLUENT indicated conformity with the results of the experiments in this study.

Acknowledgement

Author would like thanks Ministry of Research, Technology, and Higher Education for the financial support for Applied Product Research Fiscal Year 2017.

References

[1] Shet U S P, T Sundararajan, J M Malkerjuna 2014 *Refrigeration Cycles* (Mandroa: Indian Institute of Technology)
[2] Pianthong K, W Seehanam, M Behnia, T Sriveerakul, S Aphonratana 2007 Investigation and Improvement of Ejector Refrigeration system Using Computational Fluid Dynamics Technique *Energy conversion &Manajement* **48** 2556-2564
[3] Mitchley S R 1998 Vacum Boiling of Water in a Steam Jet Refrigeration System *Dissertation* (Johannesburg: Faculty of Engineering University of The Witwatersand)
[4] Saengmanee C, Pianthong K 2010 Design of a Steam ejector by co-operating the ESDU design method and CFD Simulation *The First TSME International Conference on Mechanical Engineering* 20-22 October Ubon Ratchathani
[5] Sahni R 2015 Ejector Expansion Refrigeration Systems Reaserch Inventy *International Journal of Engineering and science* **5** (2) 25-29
[6] Elbel S, Hanjak P 2008 Ejector Refrigeration: An Overview of Historical and Present Development with an Emphasis on Air- Conditioning Applications *International refrigeration and Air Conditioning Conference at Purdue* July 14-17 (2350) 1-9
[7] Chunnannond K, S Aphornratana 2004 Ejectors: Application in refrigeration technology *Renewable and Sustainable Energy, Reviews* **8** 129-155
[8] Vadalia D R 2017 Performance Optimization of Steam Jet Ejector Using CFD A Review *International Research Journal of Engineering and technology (IRJET)* **4** (2)
[9] Petel A R and Khunt J 2013 Performance Optimization of Steam Jet Ejector using CFD *International Journal Innovative Research in Science and Technology* **2** (1)
[10] Pansari K and Jilani S A K 2013 *Numerical Investigation of the Performance of Convergent Divergent Nozzle* **3** (3) 2001-2006
[11] Surya S D, Vesu T A, Raghavan K S, Chavali M 2017 CFD Simulation of Ejector in Steam Jet Refrigeration *Journal of Applied Mechanical Engineering* **6** 263
[12] Rao G R, Ramakanth U S, Lakshman A 2013 Flow Analysis in a Convergent-Divergent Nozzle Using CFD *International Journal of Research in Mechanical Engineering* **1**
[13] Satyanarayana G, Varun C, Naidu S S 2013 CFD Analysis of Convergent-Divergent Nozzle. *Acta Technica Corviniensis- Bulletin of Engineering, Tome VI (Year 2013) Vascicule* **3**
[14] Sudakhar B V V N, Sekhar B P C, Sekhar B P, Mohan P N, Ahmad M D S 2016 Modelling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics *International Research Journal of Engineering and Technology (IJRET*) **3** (8)
[15] Lakshmi K S, Vakatesh K 2016 Modelling and Simulation of Supersonic Nozzle using Computational Fluid Dynamics *Technology Management and Research* **3** (9)