International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 09 Issue: 01 | Jan 2022 www.irjet.net p-ISSN: 2395-0072

Comparison of Multiphase Flow Model and Single-Phase Flow Model of Steam Jet Ejector

Abdullah Rumane[1], Shilpa Mondkar[2]

[1]Student, Dept. of Mechanical Engineering, Pillai College of Engineering, Maharashtra, India
[2]Asst. Professor, Dept. of Mechanical Engineering, Pillai College of Engineering, Maharashtra, India

---

Abstract - *Jet ejectors are the simplest devices between all compressors and vacuum pumps. They do not contain any moving parts, lubricants or seals, and are therefore considered to be the most reliable devices with low supporting costs. In addition, most jet ejectors use steam or compressed air as a motive fluid, which is readily available in chemical plants. They are widely used in the chemical industry processes; however, jet ejectors have low efficiency. Computational Fluid Dynamics (CFD) analysis of single phase and multiphase flow was performed in a steam jet ejector taking steam and water as the two fluids. Commercial fluid code ANSYS FLUENT was used to perform analysis of both single phase and multiphase flow models. Pressure and velocity behavior of the two models showed that the multiphase flow model represented the actual behavioral conditions of a conventional steam jet ejector.*

Key Words: **Steam Jet Ejector, Mach Number, Nozzle, Motive Fluid, Mass Transfer.**

1.INTRODUCTION

Jet ejectors are broadly used in the chemical industries because of their simplicity. In many cases, they offer a great option for vacuum production in processes. They are found in variety of sizes. Because of their simplicity, conventional jet ejectors are well designed in a given situation that are highly forgiving of errors with a limited volume and performance.

1.1 Operating Principle of Jet Ejector

As shown in the Figure 1, the jet ejector design has four major sections:

1. Nozzle 2. Suction Chamber 3. Throat 4. Diffuser

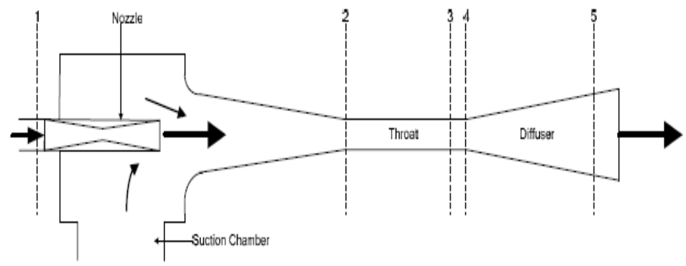

Figure 1. Jet Ejector Design

The operating principle of ejectors is described below:

1. At Point 1, subsonic motive fluid enters then nozzle. In the converging section of nozzle, the stream velocity increases and the pressure reduces. The stream reaches its sonic velocity at the nozzle throat. The increase in cross sectional area at the diverging section of the nozzle decreases the shock wave pressure and its velocity increases to supersonic velocity.
2. At Point 2, the entrained fluid enters the ejector where there is increase in velocity & reduction in pressure.
3. The motive fluid and the entrained fluid mix within the suction chamber and the converging section of diffuser or they both mix together in throat section.
4. In the throat section, there is generation of shock wave. Reduction in the mixture velocity to a subsonic condition and the back pressure resistance of the condenser results in shock wave at Point 3.
5. As the mixture flows into the diverging section of diffuser, the kinetic energy of the mixture is transformed into pressure energy.

There can be different purposes for ejector construction such as:

1. For greater penetration into the second liquid.
2. Producing a large mix between two liquids.
3. Pumping fluid from low pressure region to high pressure region [7].

1.2 Multiphase Flow

A phase can be described as one of the states of matter like solid, liquid or gas. Multiphase flow is the contemporaneous flow of several phases, with two phase flow being the simplest case. Multiphase flow is the associative flow of two or more distinct phases with common interfaces in, say, a channel. Each phase, representing a volume fraction (or mass fraction) of solid, liquid, or gaseous matter, has its individual properties, velocity, and temperature.

Characteristics of Multiphase flow: -

All multiphase flow problems have features that are characteristically different from those found in single-phase problems.

1. In the case of steam and liquid water, the density of the two phases differs by a factor of about 1000. Therefore the influence of gravitational body force on multiphase flows is of much greater importance than in the case of single-phase flows.
2. The speed of the sound changes dramatically for materials undergoing a phase change and can be orders of magnitude different. This significantly influences a flow through an orifice.
3. The corresponding concentration of different phases is usually a dependent parameter of great importance in multi phaser flows, while it is a parameter of no consequence in single-phase flows.
4. The phase change means flow-induced pressure drops can cause further phase change (e.g.,, water can evaporate through an orifice), increasing the relative volume of the gaseous, compressible medium and increasing efflux velocities, unlike single-phase incompressible flow where decreasing of an orifice would decrease efflux velocities.
5. The geographical distribution of the various phases in the flow channel strongly affects the flow behavior.
6. There are large number of fluctuations in multiphase flow.

Multiphase flow is much more complicated than single-phase flow due to the variation of flow patterns. Fluid distribution changes greatly in different flow regimes, which significantly affects pressure gradients. Since steam and water both are mixing in the mixing chamber, it is vitally important to understand the internal behavior of the phase change occurring inside the jet ejector.

2. THE CFD MODELING

With the improvement and recent development of CFD codes, a full set of fluid dynamic and multiphase flow equations can be solved numerically. The current study used commercial CFD code, ANSYS FLUENT 2021 R1 ACADEMIC version. The equations are solved by converting the complex partial differential equations into simple algebraic equations.

Mesh was created in this study for solving the mass, momentum, and energy equations. The mixture composition and phase pressures were defined at the inlet boundary of the ejector. The Relizable k-e model with standard wall functions were used due to their proven reliability in solving mixture problems.

2.1 Geometry Details

In order to perform CFD analysis, the fluid domain was developed using ANSYS DESIGN MODELER [DM]. As shown in Figure.1, the domain neglects solid geometrical regions and considers only the region containing fluid. Geometry from Huang et al. (1999) was used in the CFD model [3]. Synonymous geometries have been taken for the validation of both, i.e., single phase flow and that of multiphase flow.

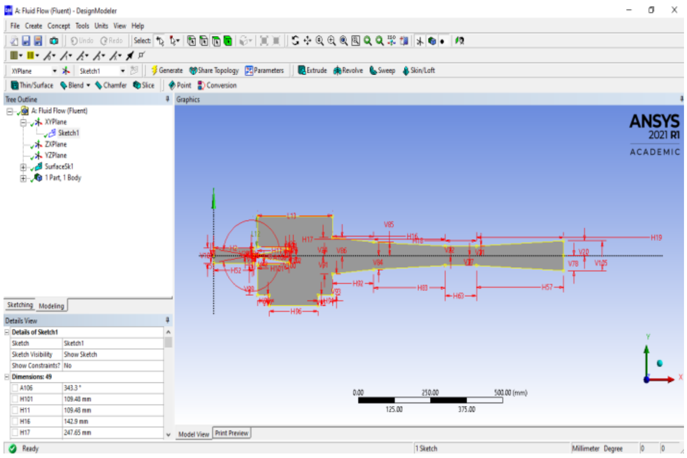

Figure 2. Geometry of Ejector Fluid Domain

Refinement of mesh was done at the critical areas of the ejector as shown in Figure 3 including All Triangle Method & Edge Sizing, so as to capture the physics of the problem in detail. The statistics of nodes and element count were 13551 and 25568 respectively.

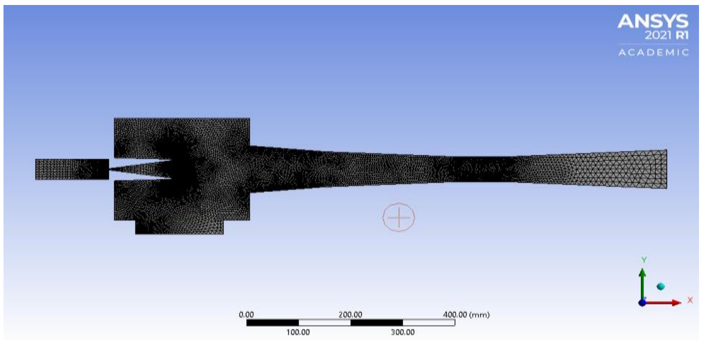

Figure 3. Meshing

2.2 Modeling Assumptions

Due to relative simplicity of the flow geometry, absence of strong body forces and relatively high flow rates, standard wall functions have been selected and are assumed to effectively model the near-wall viscosity affected regions for the turbulent flows. No slip boundary conditions are assumed at the wall of ejector. The effect of wall roughness on the flow and shear stress has not been investigated. The inlet fluids are taken as Steam at 413K temperature and 4 Bar Pressure & Water at 303K and atmospheric pressure.

2.3 Solution Strategy and Convergence for Single phase flow model

Ansys Solver was used to proceed further into investigating the flow regime of Single-Phase flow model. Inlets and outlets were provided.

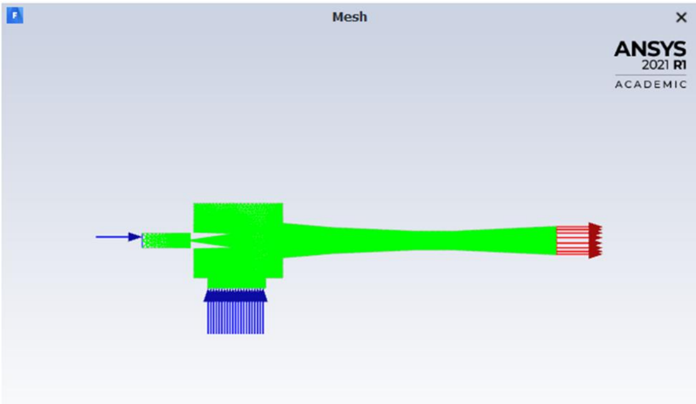

Figure 4. Inlets and outlet of Ejector

Energy equation was taken into consideration along with Realizable k-e model with standard wall function.

Figure 5. Model Selection

Time step was taken as 100 and time step size was taken as 0.001s and the solution was calculated.

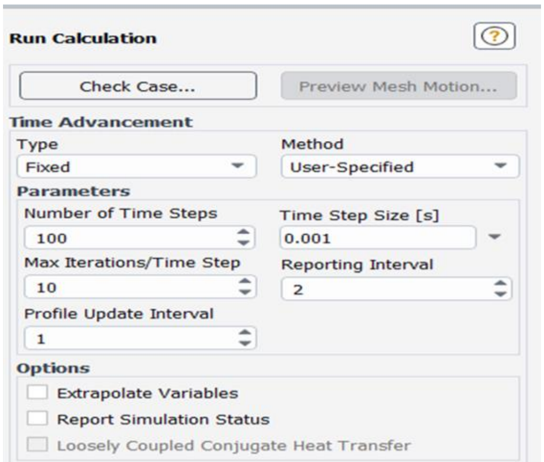

Figure 6. Calculation Steps

Solution was converged in 639 iterations.

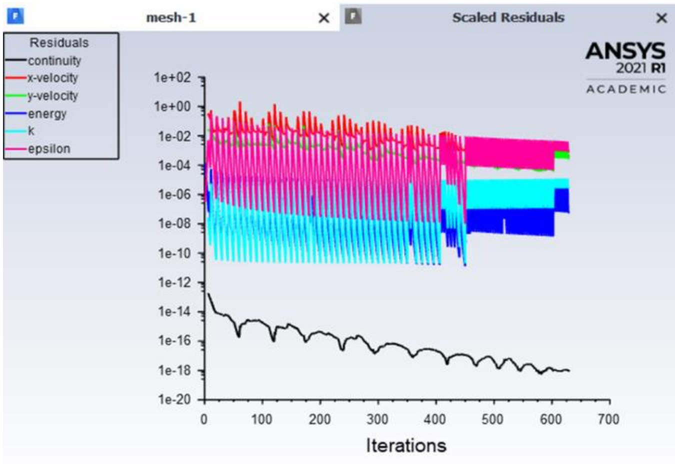

Figure 7. Solution Converging and Scaled Residuals

2.4 Solution Strategy and Convergence for Multi-phase flow model

Ansys Solver was used to proceed further into investigating the flow regime of Single-Phase flow model. Inlets and outlets we provided and VOF Model was selected as a Multiphase flow model. Courant No. was set at 0.25. The two phases include steam and water.

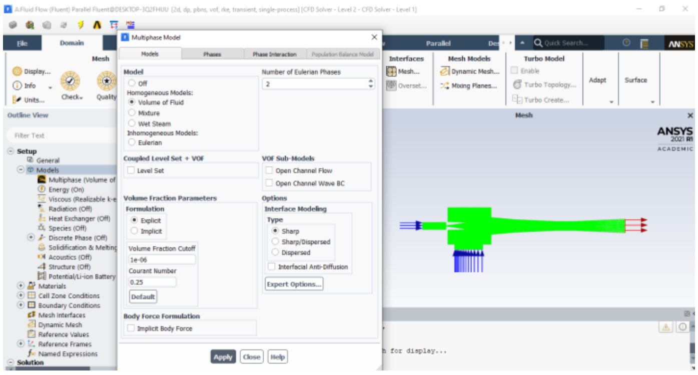

Figure 8. Multiphase model – VOF with two Eulerian phases Steam & Water

In solution methods, pressure-velocity coupling simple scheme was set. Momentum, Turbulent Dissipation Rate and Energy was kept at Second Order Upwind. For Volume Fraction, Geo-Reconstruct VOF scheme model was used.

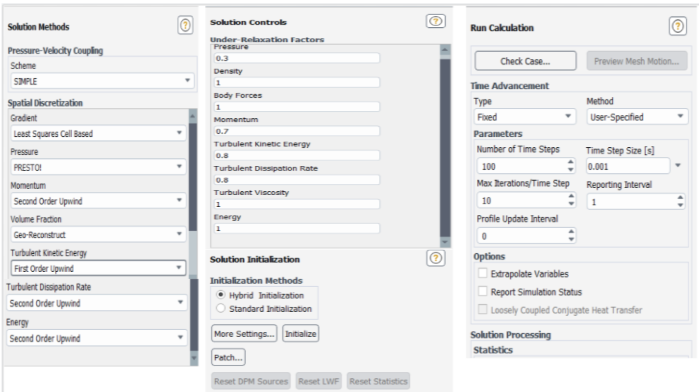

Figure 9. Solution Initialization in Ansys

3. COMPARISON & DISCUSSION OF CFD RESULTS OF MULTIPHASE FLOW MODEL WITH SINGLE PHASE FLOW MODEL

A. Pressure Contour: -

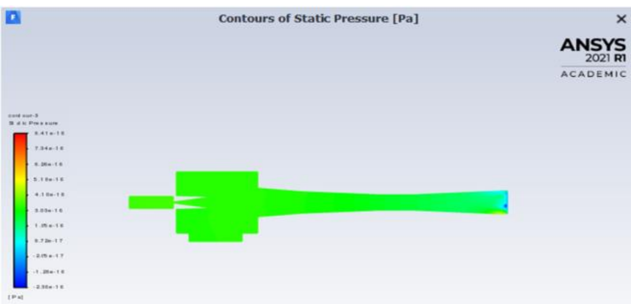

Figure 10. Pressure Contour of Single-Phase Model

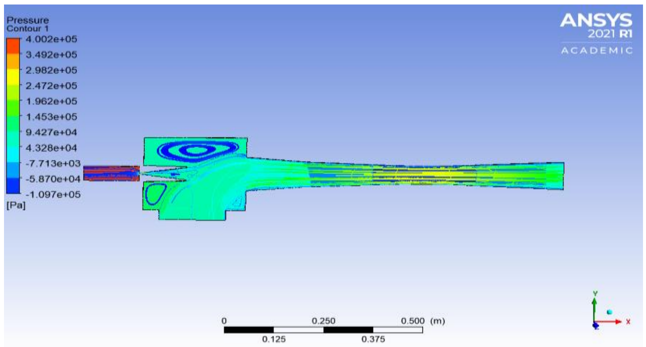

Figure 11. Pressure Contour of Multiphase Model

The pressure contour of multiphase model shows the variation of pressure of the two fluids whereas the single-phase model treats both the fluids at a constant pressure.

B. Velocity Contour: -

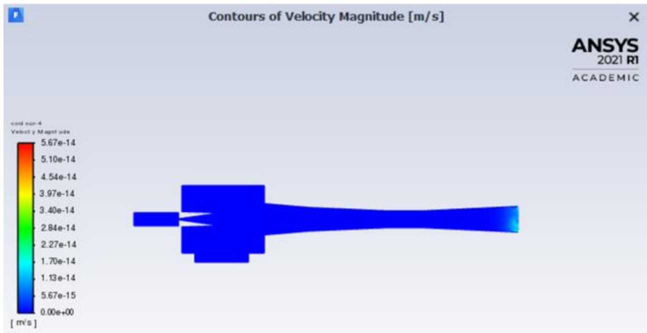

Figure 12. Velocity Contour of Single-Phase Model

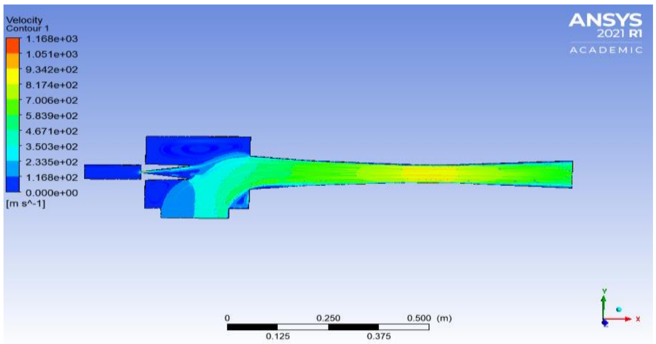

Figure 13. Velocity Contour of Multiphase Model

The velocity contour of single-phase model shows a slight increase in velocity at the exit and give no information about the flow field inside the ejector. Whereas the multiphase flow model shows the proper variation of velocity and also gives an insight about the formation of eddies in the upper region of mixing chamber.

C. Temperature Contour: -

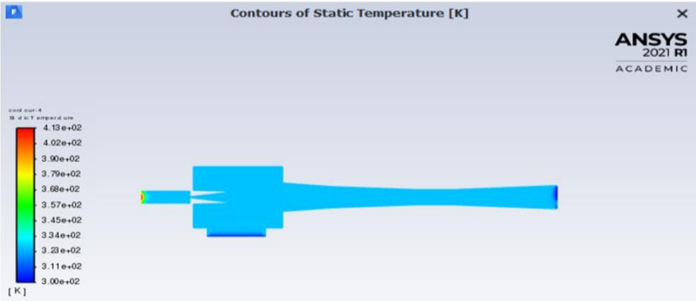

Figure 14. Temperature Contour of Single-Phase Model

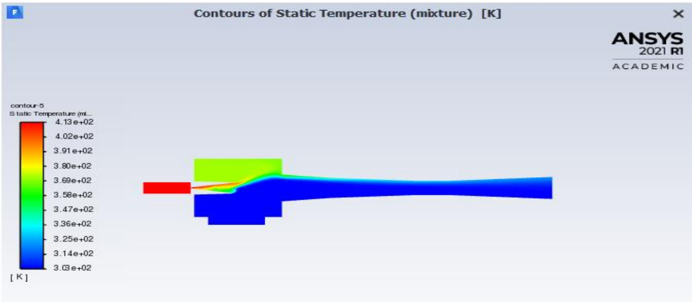

Figure 15. Temperature Contour of Multi-Phase Model

The temperature contour of the single-phase model is constant whereas that of multiphase model shows the development of shock wave just at the exit of nozzle. The temperature goes on reducing from the inlet of the steam from about 413K to 324K at the outlet of steam jet ejector.

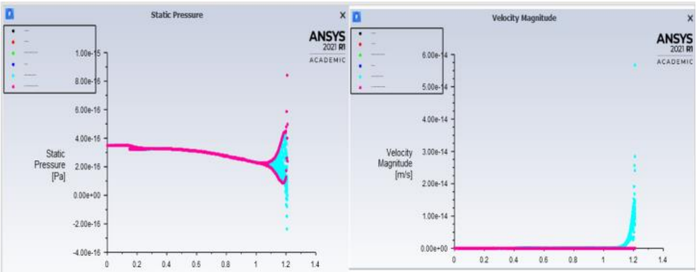

Figure 16. Graph of Static Pressure and Velocity for Single Phase Model

The graph of static pressure shows a decrease in pressure till the exit and sudden rise in pressure at the outlet of steam jet ejector. Whereas the velocity graph shows a constant velocity throughout the flow regime with a sudden rise in velocity at exit.

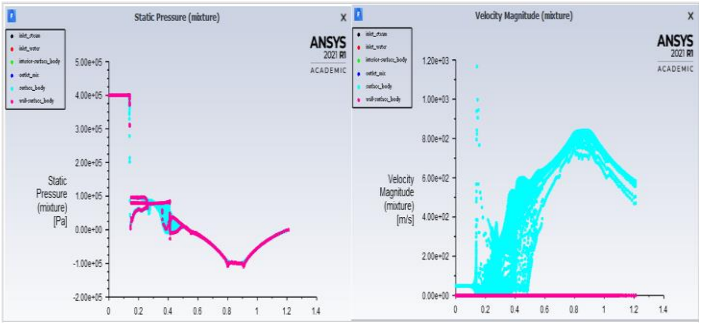

Figure 17. Graph of Static Pressure and Velocity for Multi-phase Model

The graph of the pressure and velocity of multiphase model resembles to that of the ideal steam jet ejector shown in the figure below. Both the graphs are clearly indicative about the inverse relationship of pressure and velocity.

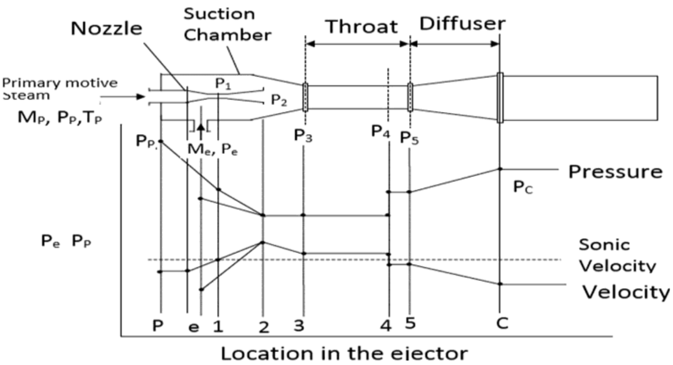

Figure 18. Conventional Steam Jet Ejector with Pressure and Velocity Graphs

It is evident that in the multiphase flow model of the steam jet ejector, there is a formation of eddies which is not seen in the single-phase flow model. Whenever there is a sudden introduction of steam and water in the mixing chamber, phase change occurs which can be seen in the multiphase model.

4. SUMMARY & CONCLUSION

In this study, a 2-dimensional CFD model of steam ejector was built to investigate the effects of multiphase flow regime over single-phase flow regime. It is thus observed that the use of multiphase flow models gives insight into the various flow parameters that might be getting affected by the flow regime. It is very useful in modifying, thus optimizing the ejector and getting to know the inside picture of what is happening in the jet ejector. It is thus concluded that the multiphase flow model of steam jet ejector is in close agreement with that of Conventional Ideal jet ejector.

Study of bubbles formation and the formation of eddies due to sudden phase change can help in effectively designing an ejector and thus enhancing the overall output of the steam jet ejector.

REFERENCES

[1] Arth R. Patel, Jayesh Khunt, Performance optimization of steam jet ejector using cfd, IJIRST –International Journal for Innovative Research in Science & Technology| Volume 2 | Issue 1 | June 2015

[2] T.Aravind, Dr P Ravinder Reddy, S S Baserkoed, Thermal Analysis of Steam Ejector Using CFD, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 3, Issue 12, December 2014

[3] Natthawut Ruangtrakoon, Tongchana Thongtip, Satha Aphornratana, Thanarath Sriveerakul, CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle, International Journal of Thermal Sciences 63 (2013) 133e145.

[4] Szabolcs Varga, Armando C. Oliveira, Bogdan Diaconu ,2008, Numerical assessment of steam ejector efficiencies using CFD, international journal of refrigeration 32 (2 0 0 9) 1203 – 1211.

[5] Xianchang Li, Ting Wang and Benjamin Day (2010), “Numerical analysis of the performance of a thermal ejector in a steam evaporator”, Applied Thermal Engineering, v30, pp. 2708-2717,2010

[6] Natthawut Ruangtrakoon, Tongchana Thongtip, Satha Aphornratana, Thanarath Sriveerakul ,CFD simulation on the effect of primary nozzle geometries for a steam ejector in refrigeration cycle, International Journal of Thermal Sciences, Issue 63, 2013, ISSN 1290-0729

[7] Abdullah Rumane, Shilpa Mondkar, A Review on Jet Ejector System, International Research Journal of Engineering and Technology (IRJET), Volume 8, issue 4, April 2021