International Journal of Thermal Technologies, Vol.2, No.3 (Sept. 2012) ISSN 2277 - 4114

Research Article

Experimental Analysis of Jet Ejector by Forced Draught

S. Gurulingam[a*], A. Kalaisselvane[a], N. Alagumurthy[a]

[a]Department of Mechanical Engineering, Pondicherry Engineering College

Accepted 31 Aug. 2012, Available online 1Sept 2012

Abstract

A jet compressor uses a jet of primary fluid to induce a peripheral secondary flow often against back pressure. Expansion of primary jet produces a partial vacuum near the secondary flow inlet creating a rapid re-pressurization of the mixed fluids followed by a diffuser to increase the pressure at the exit. Using the geometrical design parameters obtained by solving the governing equations, a CFD analysis is made using the FLUENT software to evaluate the optimum entrainment ratio that could be achieved for a given set of operating conditions, where the entrainment ratio (ER) is the ratio of the mass flow rate of the secondary fluid (propelled stream) to the primary fluid (motive fluid). In this paper a jet compressor's performance analysis is made using irreversibility characteristics .the various losses that occurs in different regions of jet compressor are quantified. Effort is made to increase the efficiency of jet compressor by reducing the losses based on minimization of entropy method. In order to match the ER that is achievable theoretically, an effort is made to force (charge) the propelled stream using a blower. So that the momentum difference between the motive and the propelled fluid is minimized. Experimental results obtained using the forced draft system is found to match the results obtained from the FLUENT analysis.

Keywords: *Ejector, Efficiency, Forced draught*

1. Introduction

Jet ejectors are the simplest devices among all compressors and vacuum pumps. They do not contain any moving parts, lubricants or seals; therefore, they are considered as highly reliable devices with low capital and maintenance costs. Furthermore, most jet ejectors use steam or compressed air as the motive fluid, which is easily found in chemical plants. Due to their simplicity and high reliability, they are widely used in chemical industrial processes; however, jet ejectors have a low efficiency. *(Keenan et al)*

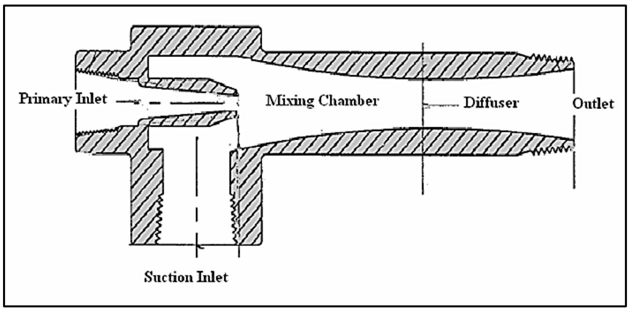

Fig. 1 Cross sectional view of a typical liquid jet pump

A high-pressure fluid with very low velocity at the primary inlet is accelerated to high velocity jet through a converging nozzle for the liquid jet pump or a converging-diverging supersonic nozzle for the gas ejector *(*Bonnington, *et al*, 1950*).* The supply pressure at the inlet is partly converted to be the jet momentum at the nozzle exit according to the Bernoulli equation. The high velocity, low static pressure primary jet induces a secondary flow from the suction port and accelerates it in the direction of the driving jet. The two streams then combine in the mixing section, and ideally the process is complete by the end of this section. A diffuser is usually installed at mixing chamber exit to lift the static pressure of mixed flow. (Sun *et al*, 1995)

2. Design aspects

The main part of designing work is to find out the cross sectional areas of the primary nozzle inlet, throat, outlet and also the secondary nozzle inlet and outlet, as well as the length of the constant area mixing chamber.

2.1 Design Aspects for Primary Nozzle

- Using the inlet conditions assumed like pressure, temperature, mass flow rate and mach number, we derived the parameters in the following way:

[*] Corresponding author's email: gurulingam525@gmail.com
Phone: +91- 9488223004

- Density of the inlet air is found out using the equation:
  $PV=mrt$ (2.1)

- Mach number is given by the equation

$$M = \frac{\text{local fluid velocity}}{\text{local sonic speed}} = \frac{V}{C} \quad (2.2)$$

Where

$$c = \sqrt{\gamma RT} \quad (2.3)$$

- Using Mach number the inlet velocity i.e. "V" is found out.
- Area of the inlet can be found from the formula:
  $$\dot{m} = \rho AV \quad (2.4)$$
- Corresponding diameter is also found from the area value
- Using the gas tables area ratio is taken corresponding to the inlet Mach number and thereby from the area ratio the area of the throat is calculated.
  $$\frac{A}{A^{\circ}}=constant, \quad (2.5)$$

for a specific mach number

2.2 Design Aspects for Secondary Nozzle

- Same procedure is followed here also, from inlet conditions assumed the diameter of the inlet and the throat is calculated.
- The exit pressure of the secondary nozzle is fixed from the pressure ratio corresponding to mach number=1

2.3 Designing of the Diverging Section (Primary Nozzle)

- Stagnation conditions are taken into account for finding out the Mach number at the exit.
  $$Pressure\ ratio: \frac{Po}{P} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma}{\gamma-1}} \quad (2.6)$$
  Stagnation temperature is calculated from the equation by using the inlet temperature conditions and mach number:
  $$Temperature\ ratio: \frac{To}{T} = \left(1 + \frac{\gamma - 1}{2} M^2\right) \quad (2.7)$$
- Later the outlet temperature is updated by substituting the new Mach number.
- Density of the air at the outlet is found out by using the exit pressure and temperature
  $$\frac{m}{V} = \frac{P}{RT} \quad (2.8)$$
- Thereby the cross sectional area of the outlet is derived

$$A = \frac{\dot{m}}{\rho V} \quad (2.9)$$

- Corresponding diameter is also calculated.

2.4 Design Aspects for the Mixing Section

- Applying momentum and energy equation in the mixing section the flow velocity and temperature are calculated.

$$Vmix = \frac{rV(s) + V(p)}{r + 1} \quad (2.10)$$

$$Tmix = \frac{rT(s) + T(p)}{r + 1} \quad (2.11)$$

- Mach no. Before shock wave M2=
  $$\frac{\eta mr M^* + Mp\sqrt{\frac{Tp}{T^{\circ}}}}{\sqrt{(r+1)\left(r + \frac{Tp}{T^{\circ}}\right)}} \quad (2.12)$$
  Ratio of actual mixture velocity to the velocity of sound in the mixture, i.e.

$$M^* = \sqrt{\frac{\left(\frac{\gamma + 1}{2}\right) * M^2}{1 + \frac{\gamma - 1}{2} * M^2}} \quad (2.13)$$

- Mach number after shock wave
  $$M_3 = \sqrt{\frac{\left(\frac{2}{\gamma - 1}\right) + M_2^2}{\frac{2\gamma}{\gamma - 1} * M_2^2 - 1}} \quad (2.14)$$

- Pressure before and after the shock wave is given by the pressure lift formula

$$\frac{P_3}{P_2} = \frac{1 + \gamma M_2^2}{1 + \gamma M_3^2} \quad (2.15)$$

- Length of the mixing section is given by

$$L = 10D_{ts} \quad (2.16)$$

2.5 Design Aspects for Diffuser Section

- Pressure lift ratio across the diffuser can be expressed by

$$\frac{P_{exit}}{P_3} = \{\eta_d \frac{\gamma - 1}{2} M_{3+1}^2\}^{\frac{\gamma}{\gamma-1}} \quad (2.17)$$

- For any mach number the area ratio is given by

$$\frac{A}{A^*} = \frac{1}{M}\left(\frac{2}{\gamma + 1} + \frac{\gamma - 1}{\gamma + 1} * M^2\right)^{\frac{\gamma+1}{2(\gamma-1)}} \quad (2.18)$$

- Since the area of the throat known and the Mach number after the shock wave that could find the outlet area of the diffuser.

3. Design and simulation

Different parameters which effects the design of an ejector is found out from different literature reviews. A C program is constructed which derives the design parameters form inlet boundary conditions. Many designs are created for different secondary mass flow rates and the results are compared. Output of the C- program will be in the form of a journal file. Journal file will get saved in the location that is specified in the program. The design software – 'GAMBIT' have got the option to run the journal file straight away. Once it is loaded the design is shown in a meshed form. The meshed model which is axis-symmetric is saved as a case file. The case file is loaded into 'FLUENT 6.3' by reading the case file. First grid is scaled to desired unit whether it is in mm or cm. Then grid is checked for any possible errors. (Emanuel *et al* , 1976) Solver properties are selected, there are two ways of solving the problem i.e. Pressure based and density based. Select a density based axis-symmetric solver. Energy, viscous properties are selected properly. Flowing fluid is selected as ideal gas, and also operating conditions are defined as standard. Boundary conditions are defined for different sides are follows Primary inlet as mass flow inlet, Secondary inlet as intake fan, Outer walls as walls, Central line as axis, Ejector outlet as pressure outlet. Once the boundary conditions are defined straight away the solving conditions are initialized. Solver is initialized from all zones. Number of iterations to be carried out is defined. Iterations are completed once the solution is converged. Different contours and vectors are plotted and analysis is done. Designing and meshing works are done in gambit and the mesh is exported to fluent software. Operating conditions and boundary conditions are specified and solver is initialized in all zones. Designs for 2bar, 3bar, 5bar is produced and put into simulation work. These designs are evaluated for different entrainment ratios (0.4, 0.6, 0.8, 1, 1.2, and 1.4). Case is iterated and different pressure and velocity contours are derived. Real time experimentation of an ejector is done and the same atmosphere is simulated in fluent also, observations are tabulated and results are compared.

3.1 simulated observations and results

3.1.1 Observations for 100000 Pascal (Primary Inlet Pressure)

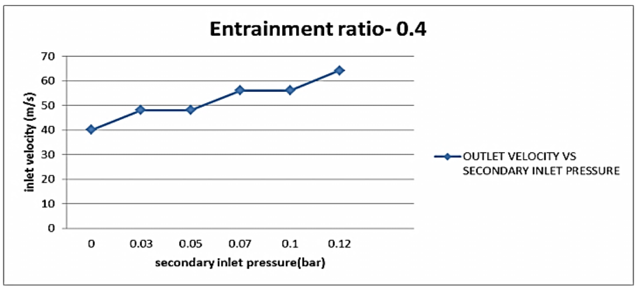

Figure 3.1 Graph between inlet velocity and secondary inlet pressure (Primary Inlet Pressure-100000 Pascal; entrainment ratio-0.4)

Velocity Contours:

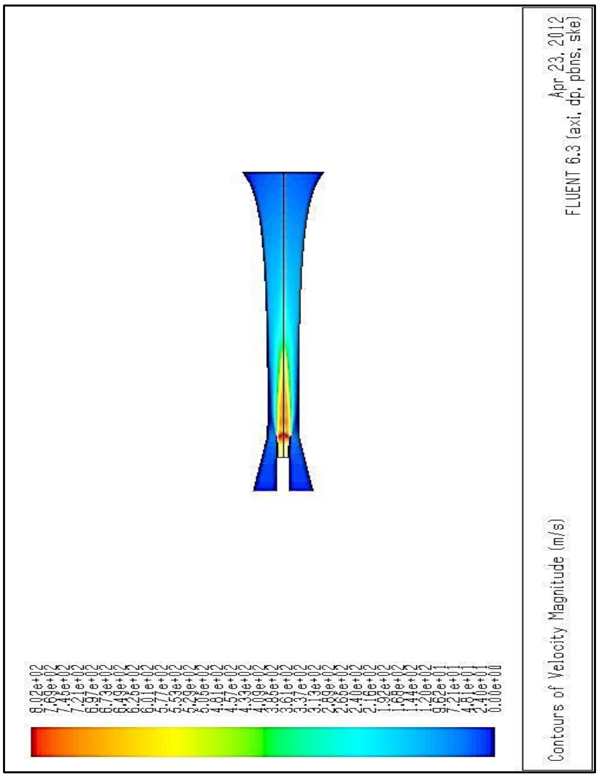

Fig 3.2 Simulated velocity contour (primary inlet pressure $1*10^5$Pascal; Secondary inlet pressure zero Pascal)

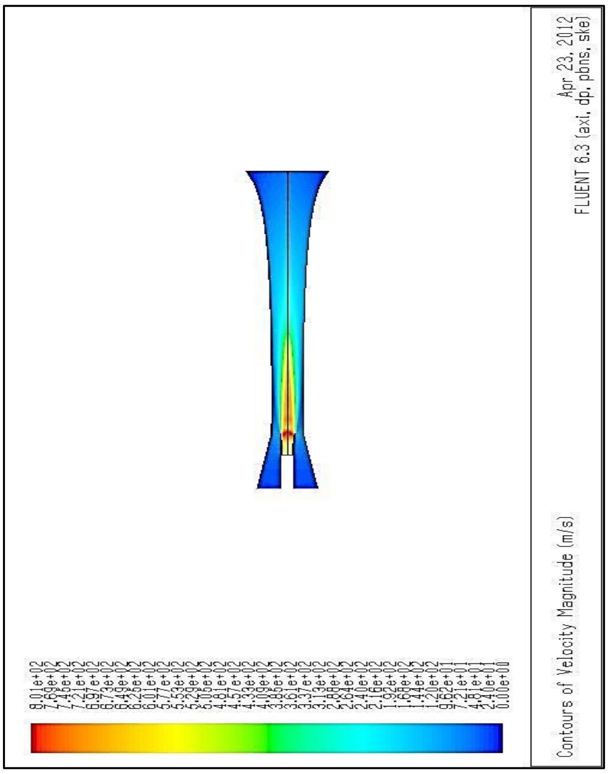

Fig 3.3 Simulated velocity contour (primary inlet pressure $1*10^5$Pascal; Secondary inlet pressure 5000 Pascal)

4. Results and discussion

4.1 Comparison between Experimental and Simulated Result

4.1.1 Simulated Values selected for comparison

Table 4.1 Simulated Values Taken For Comparison

	Intake fan pressure (bar)	Secondary inlet x-velocity (m/s)	Secondary outlet x-velocity (m/s)	Intake mass flow in primary (kg/s)	Intake mass flow in secondary inlet (kg/s)	Entrainment ratio
Natural	0	8.0155077	40.077538	0.04	0.025633286	0.64083215
Forced	0.03	16.027357	48.082073	0.04	0.035743471	0.893586775
	0.05	24.041029	56.095734	0.04	0.042654681	1.066367025
	0.07	24.044249	56.103249	0.04	0.049638131	1.240953275
	0.1	24.054789	64.146103	0.04	0.060063309	1.501582725
	0.11	32.07935	64.158699	0.04	0.063452012	1.5863003
	0.12	32.086319	72.194214	0.04	0.066787546	1.66968865

4.1.2 Experimental Observations and Results

Table 4.1.2 Experimental Values Taken For Comparison

Parameters	**Primary Flow Inlet**	**Secondary Flow Inlet**	
		Natural Draught	**Forced Draught**
H_a (m)	163.265	79.59m	175.51
Resultant Area (m2)	$7.73*10^{-4}$	$7.73*10^{-4}$	$7.73*10^{-4}$
Mass Flow Rate (Kg/sec)	0.0348	0.0243	0.03611

4.1.3 Error Occurred Between Experimentation and Simulation

Table 4.3 Error between Experimental and Simulated Results

	Entrainment ratio		Error
	Experimental	Simulated	(%)
Natural	0.6982	0.6408	8.22
Forced	1.0376	0.894	13.8

- In the CFD software, the average overall deviation for entrainment ratios between the simulation and experiment results are : {**Natural** : **8.22%** ; **Forced : 13.8%}**
- Since the occurred error value is found to be low, the simulated results are reliable.
- It shows to a conclusion that Performance can be increased by decreasing the velocity gradient, which is achieved by increasing the pressure of the secondary inlet.
- Secondary inlet velocity was found to be doubled in forced draught for the reading which matches the experimentation.{**Natural** : **8.0155077(m/s)** ; **Forced : 16.027357(m/s)}**
- As a result of forcing the secondary inlet Overall increase in outlet velocity is found to be **16.66%.** {from **40.077 m/s** to **48.082m/s**}
- From the simulation studies it was found that only a particular entrainment ratio gives best performance for an ejector which is designed for a particular capacity.

4.2 Experimental setup

Fig 4.1 Experimental setup showing ejector connected to the diesel engine outlet

Fig 4.2 Blower used for forced flow experimentation

5. Conclusions

The entire research was done to conduct studies on the performance improvement of jet ejectors. Simulation works were carried out on different boundary conditions. In a conventional ejector the secondary fluid will get inducted in because of the partial vacuum created inside the mixing section because of large gradient of velocity between the primary and the secondary inlet. Theoretical work proved that jet ejector performance is purely based upon how effectively the secondary fluid get mixed up with the primary fluid. The first and foremost motto of the study was to enhance proper intake of air in the secondary inlet, thereby increase the performance. Implementation of this idea was started by selecting various boundary conditions, designing ejector models for that particular capacity and simulating it in fluent software for natural and forced draught conditions. Once it has been done successfully in a virtual environment with the help of simulation it is observed that the secondary intake velocity gets enhanced, thereby enhancing the outlet velocity. So the proposed idea was found to be successful. In order to check the accuracy of simulation works a real time experimentation of a particular set of boundary condition was carried out and the results are compared. Error between the experimental and simulated values was found to be very low. With the help of all simulations, experimentations, data's collected and theoretical studies it is able to conclude that performance improvement of jet ejectors is possible by the enhancing the entrainment of secondary and primary fluid.

References

Keenan, J.H. and Neumann, E.P (1950), An Investigation of Ejector Design by Analysis and Experiment, *J. Applied Mechanics*, Trans ASME, 72, pp. 299-309

Bonnington, S. T. and King, A. L. (1976), Jet Pumps and Ejectors, *A State of the Art Review and Bibliography*, 2nd Edition, BHRA Fluid Engineering, Cranfield, Bedford,UK.

Sun, D.W. and Eames, I. W.(1995), Recent Developments in the Design Theories and Applications of Ejectors—A Review, *Journal of the Institute of Energy*,68, pp.65-79.

DeFrate, L. A. and Hoerl, A. E., (1959), Optimum Design of Ejectors Using Digital Computers,*Computer Techniques, Chemical Engineering Progress Symposium Series*, 55(21), pp. 43-51.

Emanuel, G. (1976), Optimum Performance for a Single-Stage Gaseous Ejector,*AIAA J.*, 14(9), pp. 1292-1296.

Keenan, J.H. and Neumann, E.P. (1942), A Simple Air Ejector,*Journal of Applied Mechanics*, 9(2), pp. A75-A81.

Neve, R. S., (1993), Computational Fluid Dynamics Analysis of Diffuser Performance in Gas-Powered Jet Pumps, *International Journal of Heat and Fluid Flow*, 14 (4), pp. 401-407.

Riffat, S. B., Gan, G. and Smith, S. (1996), Computational Fluid Dynamics Applied to Ejector Heat Pumps,*Applied Thermal Engineering*, 16(4), pp. 291-297.