ISSN (Print) : 0974-6846
Indian Journal of Science and Technology, *Vol 9(7), DOI: 10.17485/ijst/2016/v9i7/87730, February 2016* ISSN (Online) : 0974-5645

Reducing and Recycling Gases Sent to the Flare by Ejector

Fereydon Mehrmandi [1*] and Behrooz Roozbehani[2]

[1]Department of Energy Conversion, Dezful Branch, Islamic Azad University, Dezful, Iran; nima4141@yahoo.com
[2]Department of Process Engineering, Abadan Oil School, Oil Industry University, Abadan, Iran; roozbehani@put.ac.ir

Abstract

Background/Objectives: The purpose of this study is to investigate the reduction and recycling of sent gases to the flare by ejector. Methods/Statistical analysis: A balance and integrated model of CFD is provided so that the validity of this study suits for ultrasonic ejector. We investigatedthe phenomenon and behaviors of the ejector accurately with details. The symmetrical geometry of ejector was selected to achieve simplicity and greater accuracy in the calculations. Accuracy of calculation is the convergence of solution by fluent software. Findings: The practical ejectors work under a variety of boundary and initial operating regimes. In mentioned condition, there is the possibility of its replacement with the compressor in a flare gases recovering system. The efficiency of CFD was obviously clear in calculations of various ideal gases. Moreover, it is possible to design an algorithm by mathematical tools in order to minimize the computational efforts and time. This work also leads to reduction of modeling time of actual gas with minimum loss of function.Applications/ Improvements: The convergence of calculations indicates the significance of results and their consistence with theories and ejector operation.

Keywords: Ejector,Flare, Fluent,Ultrasonic

1. Introduction

Ejectors are one of the most important used devices in industry. The devices have two major tasks. One is to create vacuum and exhaust the gases and another is to mix fluids. One of the above tasks or both of them can be considered in designing and using of ejectors. The early works on ejectors can be found elsewhere[1-4].

The various advancements in flare gas reduction at early stages can be understood from various studies[5-9].

In 2010, a study was conducted about feasibility, calculation and recovering of sent gases to flare of Tabriz refinery with liquid ring compressors[10]. South Pars Gas Complex company investigated all issues and effective strategies to reduce the level of sent gases to flare and all effective strategies were intended in developing the services of the plan[11].

In recent years, reducing sent gases to flare or its recovering has been considered very important due to the increasing of international community and governmental efforts to protect the environment on the basis of the Kyoto and Montreal protocols that limit and control the emissions the greenhouse and pollutant gases.

Developing the system of reducing and recovering sent gases to flare, in addition to providing comprehensive program to prevent flaring valuable gases and pollution caused by it, also provides the most appropriate ways of using recovered gases. This system not only leads to complying with the requirements of new international standards, but also causes significant profitability.

The present study has investigated the reduction and recovering of sent gases to flare by ejector. Generally, the plan evaluates and determines the ways offlaring reduction (including the identification of problems in the functioning of the process, identifying problems in design and technical and economic feasibility study of

** Author for correspondence*

ways to reduce sent gases to flare) by field study of designing condition, unit'sperformance, equipment performance and by characterization of flare, flaring level, reasons and resources of flare gas producing and classifying the resources based on the various types of flaring. Since measuring the flow of sent gases to flare is essential to reducing management, in this study measurement of gas flow sent to flare has been intentioned as one of the most important steps; in addition to investigation of practical strategies of reducing and managing flaring system and the most important methods were selected, established and tested by technical and economic investigation of all of possible ways to reduce flaring and by rate measurement systems.

2. Explanation of the Problem

The desired geometry has two pressure inlets and a pressure outlet. The geometry was selected as a symmetric model with symmetry to the central axis in order to reduce the calculation difficulties.

The flow range has growing boundary layers on corresponding geometry walls. The flow analysis was performed by investigating each input and output boundaries of flow that the present sample has two inlet and an outlet areas.

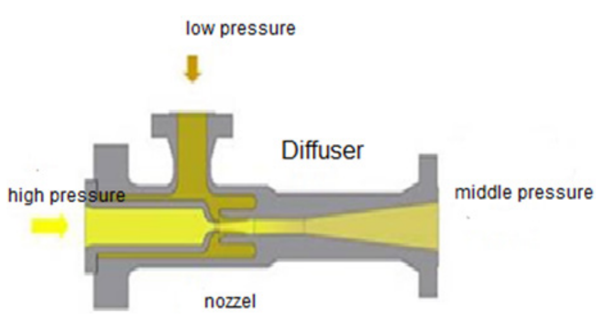

Figure 1. Nozzles, diffusers and placement of pressure.

Therefore, the flow passages within elements should be counted with considering inlet and outlet borders and imposing real conditions,Figure 1 shows the nozzles, diffusers and pressure placement. Figure 2 show input and output of pressure and the axis. As the flow passes within boundary layers, the boundary layer is significantly grown, that led to friction losses. In this analysis, it was supposed that the fluid inlet condition to early nozzle of ejector is the natural gas with average pressure. Analysis of computational fluid dynamics (numerical method) seeks to develop the design and model of the ejector in order to gain the design of the ejector.

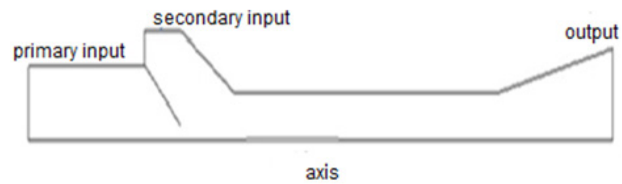

Figure 2. Inputs and outputs of pressure and axis.

3. The Basic Governing Equations

The low of mass conservation equations is as follow:

$$\frac{\partial M_{cv}}{\partial t} = \sum_{in} \dot{m} - \sum_{out} \dot{m} \tag{1}$$

where, M cv is the amount of mass in control volume in any moment and m is mass Debby in input and output of control volume.

If the velocity components at the point of (x, y) are described as u and v respectively, by the mass conservation relation (the above equation) we will have:

By dividing the above equation on the size of control volume of xΔyΔ we have:

$$\frac{\partial}{\partial t}(\rho \Delta x \Delta y) = \rho u \Delta y + \rho v \Delta x - \left[\rho u + \frac{\partial(\rho u)}{\partial x}\Delta x\right]\Delta y - \left[\rho v + \frac{\partial(\rho v)}{\partial y}\Delta y\right]\Delta x \tag{2}$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \tag{3}$$

$$\frac{\partial \rho}{\partial t} + u\frac{\partial \rho}{\partial x} + \frac{\partial \rho}{\partial y} + \frac{\partial \rho}{\partial z} + \rho\left[\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right] = 0 \tag{4}$$

$$\frac{}{Dt} + \ \nabla. \ = 0 \tag{5}$$

The equation of motion (momentum or transfer) is as follow:

$$\frac{\partial}{\partial t}(M.Vn)_{cv} = \sum F_n + \sum_{in} \dot{m} V_n - \sum_{out} \dot{m} V_n \tag{6}$$

where, n is the direction where the analysis is done. V_n and F_n are velocity and fluid forces component in the event of n

Nervier – stokes equation along x , incompressible flow and viscosity μare fixed as follow[12] :

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+X \qquad (7)$$

$$\rho\frac{DV}{Dt}=-\nabla P+\mu\nabla^2 V+F \qquad (8)$$

F is volume force vector per volume unit of (x, y, and z).

Shear stress is equal to zero and only the normal stresses (which are fluid pressure) are considered non – zero (isotropic stresses or stresses are non – respect to certain direction). The size of these stresses are equal to pressure size and with negative sign.

Motion equation in the form of Cartesian (General Equation form) is as follow[13,14]:

$$\rho\frac{DU_i}{Dt}=\rho\left(\frac{\partial ui}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=-\frac{\partial P}{\partial x_i}+B_i \qquad (9)$$

The above equation in three orders of Cartesian is as follow:

$$\rho\frac{Du}{Dt}=\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+B \quad (10)$$

$$\rho\frac{Dv}{Dt}=\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+B_y \quad (11)$$

$$\rho\frac{Dw}{Dt}=\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+B_z \quad (12)$$

The Euler equations for in- viscid flow (withoutfriction) is as follow:

$$\frac{D\vec{V}}{Dt}=\frac{\partial\vec{V}}{\partial t}+\left(\vec{V}.\vec{\nabla}\right)\vec{V}\rightarrow\rho\left[\frac{\partial\vec{V}}{\partial t}+\vec{\nabla}\left(\frac{V^2}{2}\right)-\vec{V}\times\left(\vec{\nabla}\times\vec{V}\right)\right]=-\vec{\nabla}P+\vec{B} \qquad (13)$$

by doing an integral from equation (13) between two points and along a flow line, $\vec{ds}$ is considered as a longitudinal element along a flow line and $\vec{ds}$ is calculated by dot product of both sides of general equation (the above equation).

$$\rho\left[\frac{\partial\vec{V}}{\partial t}.\vec{ds}+\vec{\nabla}\left(\frac{V^2}{2}\right).\vec{ds}-\vec{V}\times\left(\vec{\nabla}\times\vec{V}\right).\vec{ds}\right]=-(\vec{\nabla}P+\rho\vec{\nabla}\psi).\vec{ds} \qquad (14)$$

Since the $\vec{V}$ vector is parallel to $\vec{ds}$ vector, therefore the vector $\vec{V}\times\left(\vec{\nabla}\times\vec{V}\right)$ is perpendicular to vector $\vec{ds}$ and as a result, the point product of these two vectors is equal to zero.

$$\vec{V}\times\left(\vec{\nabla}\times\vec{V}\right).\vec{ds}=0 \qquad (15)$$

Therefore, after doing integral from both sides of equation (14) along a flow line for in-viscidflow (withoutfriction) we will have:

$$\int_1^2\frac{\partial\vec{V}}{\partial t}.\vec{ds}+\frac{V_2^2-V_1^2}{2}+\int_1^2\frac{dp}{\rho}+\psi_2-\psi_1=0 \qquad (16)$$

Ψ is gravitational potential equal to gz. Z is measured height from an arbitrary reference and ρ gz is gravitational force and $\hat{Z}$ is unit vector perpendicular to the center of the Earth.

$$\frac{V_2^2-V_1^2}{2}+\int_1^2\frac{dp}{\rho}+g(z_2-z_1)=0 \qquad (17)$$

That is generally called Bernoulli equation or Euler equation. Since the amount of density id always constant for an incompressible flow, therefore the general equation (Eulerequation) can be achieved by modifying the integral of above equation.

$$\frac{V_2^2-V_1^2}{2}+\frac{P_2-P_1}{\rho}+g(z_2-z_1)=0 \qquad (18)$$

The function of the ejector is based on Euler principle. According to the Euler principle, the amount of energy in a study flow and withoutfriction is fixed and equal to the total amount of kinetic, potential and pressure energies.

$$\frac{V^2}{2}+gz+\int\frac{dP}{\rho}=C \qquad (19)$$

According to the law of energy conservation, this amount of energy is always constant in the absence of friction.If the speed decreased due to the cross– section changing, the amount of energy is converted to the pressure energy and conversely, the pressure is reduced by increasing of speed. The increasing and decreasing of fluid speed is possible in equipment in which the cross section of fluid is changeable. The geometry of these devices is in converging or diverging form and their task is to convert the fluid enthalpy to kinetic energy and vice versa. Depending on the speed of the fluid at the entrance of the facility is more or less than the speed of sound, the facility causesan increase or decreasein velocity by its geometry shape.

The boundary condition for primary and secondary inlets has been specified in the form of pressure stagnation and temperature stagnation and is specified in the form of static pressure to spreader output[15]. The output boundary was selected in the form of pressure condition, rather than velocity boundary condition, in order to prevent from backflow problems. Since the modeling geometry has cylindrical shape and is symmetric to the axis, therefore the central line can be named as axis. The used fluid was *Methane* (CH_4).The boundary condition in two inputs was specified in the form of inlet pressure and in output, it was specified in the form of outlet pressure. In this type of geometry, the ejector has two surrounding walls. The wall considered as a fixed constant and without Unconditional shear slip. The Rough coefficient of it is zero. The wall is made up of aluminum with a thickness of zero. The amount of heat is zero. The first output pressure is between 600,000 to 1.8 million Pascal. The pressure in the second input is less than the pressure at the first input that is usually selected equal to half of the initial inlet pressure. The amount of pressure in the output should be between the first and second input pressure and usually the selected amount is higher than initial input pressure.

4. Program Control

The software used for the current work is the business FLUENT which is widely utilized in turbo machines industry analysis such as compressors flow, hydraulic turbines, gas turbines, pumps, valves, rocket motor, pneumatic control units, nozzles, holes and ducts. The present geometry has been meshed using GAMBIT. The boundary layer ability to engage and network is produced and then the FLUENT enters and solves the problem. The two-dimensional geometry with the growth of boundary layer is shown in Figure 3.

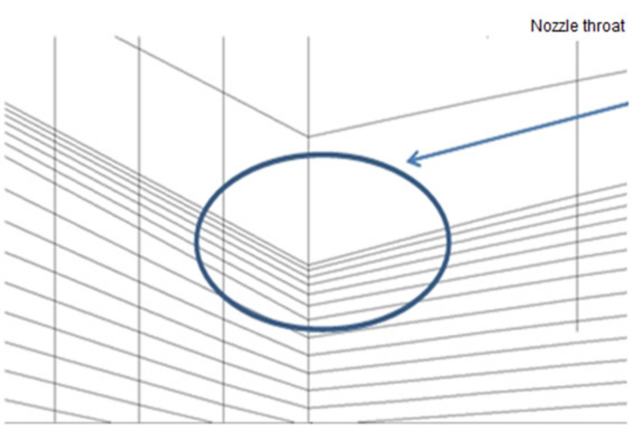

Figure 3. The growth of the boundary layer near the wall of the nozzle outlet.

Accurate modeling in the boundary layer is essential to achieve adverse pressure gradients created with certainty[16]. Therefore, a sufficient number of grid lines in this area with the initial size of 0.10 and growth rate of 1.01 have incorporated their boundary layers. Fix-sized functions are used for meshing the ejector. The ejector's geometry has been provided using the proportion of getting layered. This model is a symmetrical two-dimensional model that contains only the lower half of the ejector.

The ultrasonic flow requires the use of density based on a solver of the Couple kind. In the present study, the Couple type of a solver is used. The second type of upwind discrete is also used for the equations of motion.

5. Results

5.1 Pressure Profile

Contour of the total pressure is shown in Figure 4. This contour represents the pressure reduction in the primary nozzle downstream due the mixing of the two fluids and the change in the ejector nozzle input diagonal. However, there is a decrease in pressure caused by the energy transferred between the secondary current recessions there. Pressure recession along the axis of ejector is transferred because of the increased density of the primary flow. This phenomenon is explained by the high pressure fluid motion increasing. By increasing the flow rate the mainstream gets higher and thus mixing nozzle is moves downstream. Figure 4 is shows clearly that the pressure of output flow is higher than the pressure of secondary flow, which is the desired result.

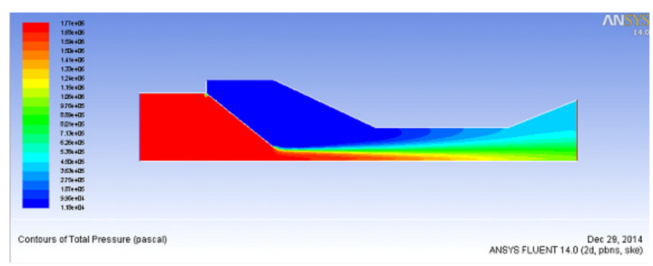

Figure 4. Total pressure contour.

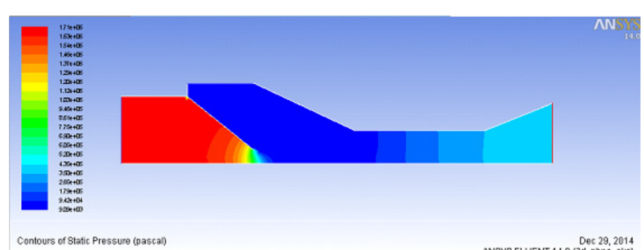

Figure 5. The static pressure contour.

Static pressure contour is shown in Figure 5. The static pressure decrease at the nozzle outlet is more significant than the pressure recession in input. It is due to the high speed of the nozzle outlet in tune with higher static pressure.

5.2 Velocity View

The velocity profile intended for the base case values is shown in Figure 6. The secondary fluid flow rate is low at the beginning, but its speed increases when mixed with a small amount of the fluid from the input.

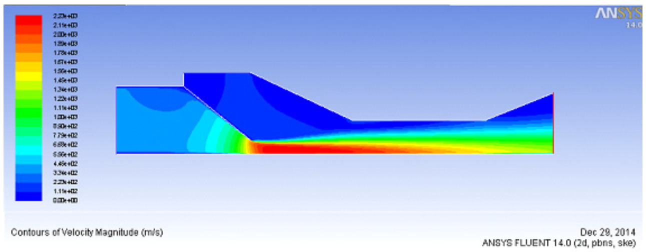

Figure 6. Velocity Contour.

These shock waves are caused by non-uniform mixing of the two streams. The flow in pipes is already fixed at supersonic speed and the diffuser helps achieving the static pressure at the end.

Figure 7 shows the contour speed of the nozzle throat. Then, the contour velocity vector in inlet2 is shown in Figure 8.

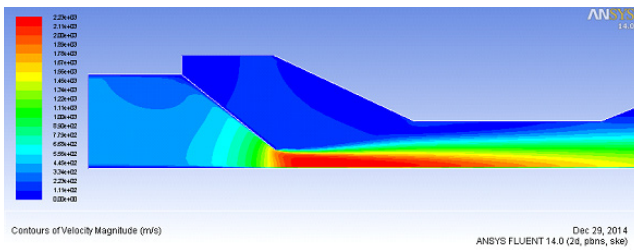

Figure 7. Velocity contours at the nozzle throat.

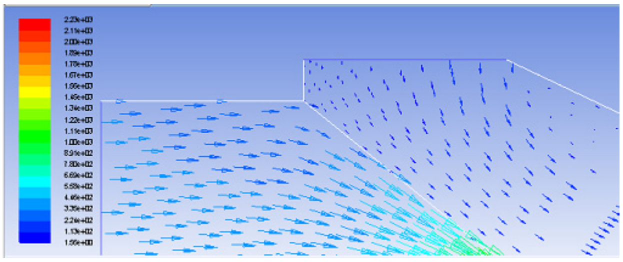

Figure 8. The contour of the velocity vector at inlet2.

The velocity vector contour in the mix can be seen in Figure 9. In Figure 10, the velocity vectors can be seen.

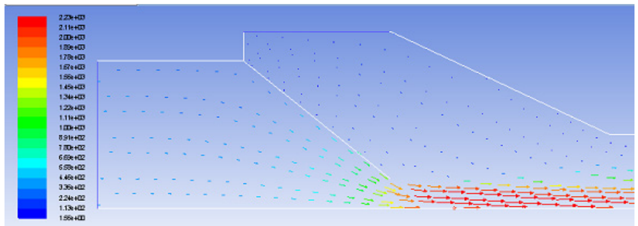

Figure 9. The contour of velocity vector in the mixture area.

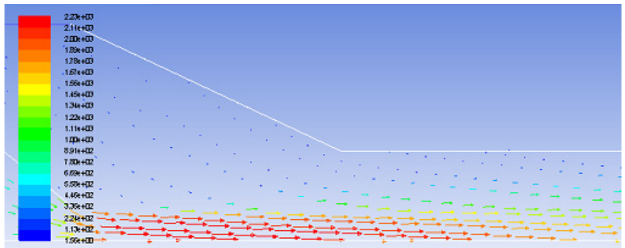

Figure 10. The velocity vectors.

5.3 Mach number Profile

A series of shock waves and the spread waves can be seen in a diamond shape in the primary downstream. Mach number on the back the primary nozzles is higher than at the nozzle outlet, and the primary reason for this phenomenon is the higher pressure behind the nozzle outlet. Mach number profile is shown in Figure 11.

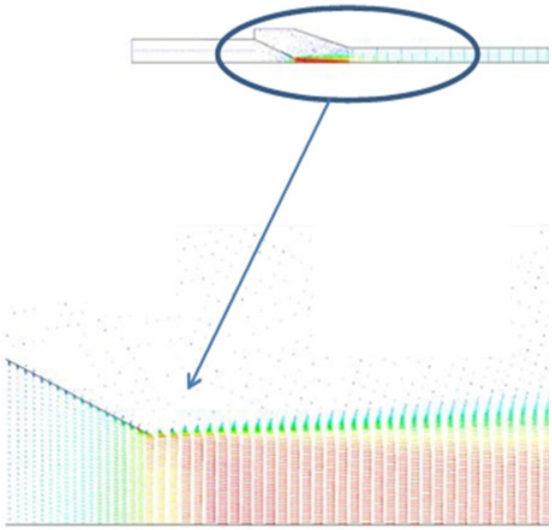

Figure 11. Mach number Profile.

5.4 Temperature Profile

Constant temperature profile for the base case is shown in Figure 12. The temperature at the initial flow Kelvin and the secondary flow is 318 degrees Kelvin and the mixing of the two streams keepsthe temperature of 318 degrees Kelvin. Temperature contours are fixed at this state.

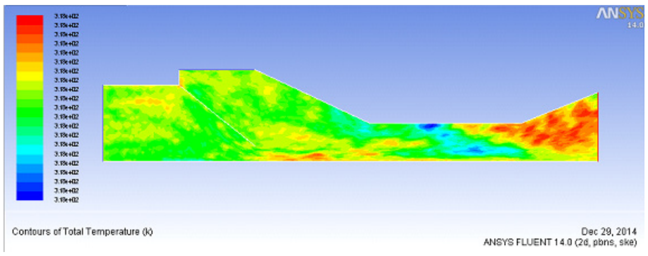

Figure 12. General profile of constant temperature.

5.5 Turbulence Intensity

Turbulence intensity for the base case is shown in Figure 13. Turbulence intensity ranges from the minimum to the maximum of its ability. The region with the greatest intensity of turbulence is after the nozzle outlet between primary and secondary flow. The turbulence intensity eventually disappears when entering the mixing chamber. This viscous interaction between two flows is the existence of shear layers of the primary current influencing the secondary flow, which creates small twisting vortex interaction.

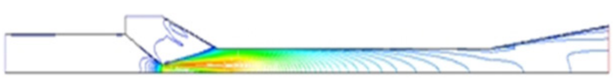

Figure 13. The contour of the turbulence intensity.

A view of the contour of the turbulence intensity with regard to all levels is shown in Figure 14.

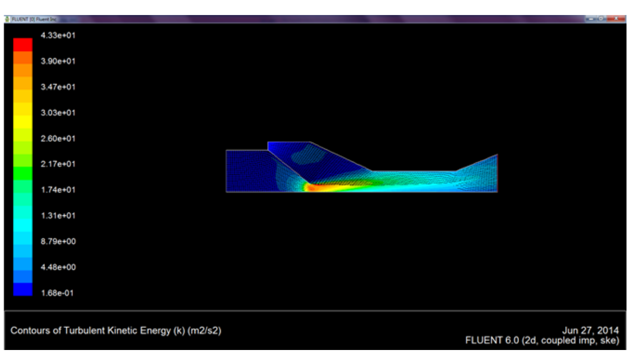

Figure 14. Contour of turbulence intensity with regard to all levels.

6. Conclusion

In this paper, the reduction and recovering of gas delivered to the flare carried by ejector was investigated. A balanced and integrated CFD model was presented that the validity of the results is suitable for use with the ultrasonic ejector. These results cover a wide range of operating conditions. In this research, phenomena and behavior in the ejector is more accurately studied. The ejector's symmetric geometry is selected for the convenience and accuracy in calculations, the accuracy of the calculations is the integration of the fluent software solutions. The convergence of calculations represents significant results and is compatible with the theory and practice of the ejector. The applicatory ejectors have the ability to work under a variety of initial and boundary conditions in which the density is high enough, and in such case there is a chance to replace it with a recover Flare gas compressor system. Efficiency of CFD is clear in the calculation of ideal gas. Mathematical tools can also be used to design an algorithm using computational effort and minimize calculations time. It also reduces modeling time for the actual gas, in which case we are faced with minimal reduction in performance.

7. References

1. Stefan E, Harnjak P. Ejector refrigeration: an overview of historical and present developments with emphasis on air-conditioning applications. International Refrigeration and Air Conditioning Conference; U S A; 2008.p. 1–8.
2. Mehr-Homji CB. The historical evolution of turbo machinery. Proceedings of the 29th Turbo machinery Symposium; 2000. p.281–322.
3. Eisner R, Pieper PJ. The world's greatest debtor nation. North American Review of Economics and Finance. 1990; 1(1):9–32.
4. Lambooy T, Varner MA, Argyrou A. The corporate responsibility to remedy (3rd Pillar Ruggie Framework)-Analysis of the corporate responses in three major oil spill cases: Shell-Nigeria, BP–US (the Gulf), Chevron–Ecuador; University of Oslo: Norway; 2011.
5. Refinery R. Flare minimization plan. Chevron: USA; 2009.
6. Sinaki SY, Tarighaleslami AH, Jafarigol F. Study on external costs of flare gases using Asia development bank method;Islamic Azad University: Tehran;2011. p. 1–6.
7. Hope T. The zero continuous flaring technology. Statoil EPN ONS Brage; 2012.
8. Nowaishi K, AAL-Khaldi F, Ba-Shammakh MS. Flaring control,GPA-GCC specialized session on environmental challenges in gas processing; Doha: Qatar; 2008.

9. Alliance CAS. Management of routine solution gas flaring in Alberta. Clean Air Strategic Alliance, Edmonton. 1998; 29(52):27.
10. Machel HG. Concepts and models of dolomitization: a critical reappraisal. Geological Society, London, Special Publications. 2004 Jan 1;235(1):7–63.
11. Pulver S. Making sense of corporate environmentalism an environmental contestation approach to analyzing the causes and consequences of the climate change policy split in the oil industry. Organization & Environment. 2007 Mar 1;20(1):44–83.
12. Kasaeipoor A, Ghasemi B, Aminossadati SM, Convection of Cu-water nanofluid in a vented T-shaped cavity in the presence of magnetic field. International Journal of Thermal Sciences. 2015; 94:50–60.
13. Ramakanth M, Balachandar C, Venkatesan M. Prediction of channel diameter to reduce flow mal distribution in radiators using ANN. Indian Journal of Science and Technology. 2015; 8(S9):341–46. DOI: 10.17485/ijst/2015/v8iS9/65585.
14. Sastry DRVSR. MHD thermosolutalmarangoni convection boundary layer nanofluid flow past a flat plate with radiation and chemical reaction. Indian Journal of Science and Technology. 2015; 8(13):1–8.DOI: 10.17485/ijst/2015/v8i13/55226.
15. Rad MZ, Kasaeipoor A. A numerical study of similarity solution for mixed-convection copper-water nanofluid boundary layer flow over a horizontal plate.Modares Mechanical Energineering. 2015; 14(14):190–8
16. Kasaeipoor A, Ghasemi B, Raisi A. Magnetic field effect on nanofluid water-cu natural convection in an inclined T shape cavity. Modares Mechanical Engineering. 2014; 14(12):179–89.