Online ISSN: 2981-0396 Print ISSN: 2981-0388

International Journal of Iron & Steel Society of Iran

Journal Homepage: https://journal.issiran.com/

Research Article

Enhancing the Production Rate of Mobarakeh MIDREX Direct Reduction Mega Module Unit Using Numerical Simulation and Geometric Parameter Study of its Ejector Stack

M. Darbandi *[1], M. Fazeli [2], D. Rashidi [3], B. Basiriani [4], S. M. Hosseinian [5], M. Allah Dadi [6]

[1] Faculty of Aerospace EngineeringSharif, Sharif University of Technology, Tehran, Iran
[2, 3, 4, 5, 6] Esfahan's Mobarakeh Steel Company, Esfahan, Iran

ARTICLE INFO

Keywords:

Ejector, Stack, Nozzle, Computational Fluid Dynamics, Sensitivity Analysis, Geometrical Parameters.

Article history:

Received 20 May 2024
Received in revised form 06 September 2024
Accepted 26 November 2024

ABSTRACT

This study uses the numerical simulation to successfully improve the performance of a subsonic ejector stack, which is used for the expulsion of flue gas in a Direct Reduction Iron unit located in Esfahan's Mobarakeh Steel Company. The current extended computational fluid dynamics (CFD) software uses the finite-volume method to solve the governing equations representing the incompressible turbulent flow through the ejector stack. The study specifically focuses on evaluating the influence of key geometric parameters, such as the mixing chamber length, diffuser length, and the nozzle ejector and diffuser angles, on enhancing the deteriorated performance of the ejector. The performed simulations show that the mixing chamber length cannot be treated as a good geometry parameter to enhance the present performance. As another important parameter, it is shown that the diffuser length can reliably enhance the ejector performance. It needs about 38% increases in this length to achieve an optimum performance value. Examining the convergence and divergence angles of the nozzle and diffuser of the ejector, respectively, it is shown that the optimal angles are 4.9 degrees for the divergence part and 12.5 degrees for the convergence part. Indeed, this study provides a novel and fundamental approach, which helps the industry people to improve the efficiency of their defective ejector stack systems.

1. Introduction

Indeed, the ejector is a device with so many applications across various industries including the steel, electrical, and chemical sectors. Notably, the ejector system is free from any moving part, which consequently leads to a lower maintenance cost [1]. There are many different parameters, which can affect the performance of an ejector stack, of which the geometric parameters emerge as the most impactful ones. Contrary to the numerous studies that have been performed on the supersonic ejectors [2-7], there has been less research to pay attention to the effect of stack geometry parameters on the subsonic ejector performances. Among the leading works, Watanabe [8] experimentally concluded that the maximum ejector performance would occur when the distance from the nozzle to the mixing chamber inlet was

* *Corresponding Author*
Email: darbandi@sharif.edu
Address: Faculty of Aerospace EngineeringSharif, Sharif University of Technology, Tehran, Iran
1. Professor, 2. M.S., 3. M.S., 4. M.S., 5. M.S., 6. M.S

DOI: http://10.22034/IJISSI.2024.2029716.1293
Published by ISSI (Iron & Steel Society of Iran)

2.56 times the diameter. Additionally, he found that the optimal ratio of mixing chamber length to the diameter would be between 5.8 to 6.0. Vias and Kar [9] experimentally investigated the impact of various geometrical parameters on the ejector's performance. They showed that the length of the mixing chamber should exceed 44 times the ejector nozzle diameter for optimal performance. However, this has been corrected by the next researchers including the present study.

The researchers in the past twenty years have used both experimental approaches and numerical tools to evaluate the impacts of geometric and applied parameters on the resulting ejectors' performances. Riffat and Omer [10] explored the nozzle's position and its consequential effect on the ejector system's performance. They reported that the substantial impact of the nozzle's location on the ejector system's performance would be attributed to its influence on the length of the primary and secondary fluid mixing zones. Riffat, et al. [11] have presented a complete review of the ejector technology development. Yadav and Patwardhan [12] investigated the influence of parameters such as the suction chamber diameter, divergence angle, and nozzle position on the ejector system's performance. Meakhail, et al. [1] assumed a constant ratio of diffuser exit diameter to mixing chamber diameter and investigated the effects of variations in geometrical parameters such as the number of inlets, nozzle location, length of the mixing chamber, and the divergence angle of the diffuser section on the achieved performances. Zhang, et al. [13] conducted a careful research to explore the nozzle location's effect on the ejector system's performance, utilizing both numerical simulation results and laboratory data. Their findings indicated that the optimal geometric composition of the ejector would vary under different operating conditions. Li, et al. [14] investigated the configuration dependence and the optimization of the entrainment performance for the gas–gas and gas–liquid ejectors. Yang, et al. [15] performed a numerical investigation to study the mixing process in a steam ejector with different nozzle structures. Kong and Kim [16] studied the performance of a two-stage ejector–diffuser system both analytically and computationally. Wang, et al. [17] used numerical tools to optimize the ejector primary nozzle geometries. Chen, et al. [18] used numerical methods and enhanced the ejector performance via using the combined adjustable geometry and bypass methods. Tashtoush, et al. [19] have presented a comprehensive review of the ejector design, performance, and applications. Yan, et al. [20] performed numerical investigations to optimize the ejector primary nozzle geometry with fixed/varied nozzle exit position. Hadi, et al. [21] used the CFD method to optimize the hydrocarbon ejector. Tavakoli, et al. [22] used numerical approaches and incorporated a fluidic oscillator as the primary nozzle and enhanced the performance of a subsonic ejector. Schillaci, et al. [23] presented a detailed investigation of the numerical modeling of air ejectors covering supersonic, subsonic, and closed-port operations. Kumar, et al. [24] have provided a detailed review of the research made in the field of ejector systems including the design methodology, geometrical parameters, operating parameters effect, CFD studies, turbulence model selection, working fluid, and irreversibility of the ejector system.

The primary objective of this study is to investigate an ejector stack for expelling the hot flue gas generated by a Direct Reduced Iron (DRI) unit into the atmosphere. The DRI is located in Esfahan's Mobarakeh Steel Company. Therefore, it is very crucial to explore how the geometrical parameters of the ejector stack would influence its overall performance and efficiency. The DRI Unit indicates that the ejector stack does not perform normally and that it performs much less than its in-design performance. Therefore, the current research explores the impact of geometric parameters, namely mixing chamber length, diffuser length, diffuser divergence angle, ejector nozzle angle, and mixing entry nozzle angle, on enhancing the performance of the available subsonic ejector stack. This is the simplest way to improve the performance of an already defective ejector stack.

2. Introducing the Present Ejector Stack

As known, the ejector stack can play a crucial role in enhancing the efficiency and performance of industrial exhaust systems. Fig. 1. presents a general overview of an ejector stack's configuration. As illustrated in this figure, the motive air enters the nozzle to expel the secondary flow, which is the flue gas in the present work. Additionally, the stack comprises four other essential components including a suction chamber, a throat or mixing entry, a mixing chamber, and a diffuser. The main objective of this ejector is to generate a suction force within the primary process and facilitate the extraction of flue gases from the original gas storage such as a combustion chamber. The ejector stack system consists of a fan that propels air into the stack and an ejector nozzle, which converts the air into a high-speed jet. The high-speed jet generated by the ejector nozzle induces a pressure reduction precisely at the nozzle outlet, resulting in a substantial pressure gradient between the inlet of the flue gas and the interior of the stack. This pressure gradient sucks the flue gas and enhances the mixing of gases within the stack. Subsequently, the mixed gas traverses a diffuser, where its kinetic energy undergoes conversion into pressure energy and causes pressure recovery before being discharged into the atmosphere.

As said previously, the primary objective of the present work is to explore the impact of geometric parameters, namely mixing chamber length (L_{MC}), diffuser length (R), diffuser divergence angle (θ), ejector nozzle angle (β), and mixing entry nozzle angle (α), on enhancing the performance of present DRI ejector stack using the CFD. It should be noted that the internal diameter of the refractory has been carefully taken into account in all the present calculations and simulations. As was said previously, the current ejector stack is to expel the hot flue gas generated by a DRI Unit into the atmosphere. The DRI Unit is located in Esfahan's Mobarakeh Steel Company. This ejector is designed for a discharge of 327,190 (Nm^3/h) of flue gas, which is fed from two separate inlets, into the atmosphere.

2.1. The Governing Equations and the CFD Methodology

The CFD software is used to simulate the physics of flow through the ejector stack in expelling the flue gas, obtain its performance at design conditions, and evaluate the variation of its performance with the variation of some geometric parameters. The fundamental assumptions are that the turbulent flow through the ejector stack is steady, incompressible, and viscous. The continuity, momentum, and energy conservation equations, commonly known as the Navier-Stokes equations, can be expressed as

$$\nabla . V = 0 \quad \text{Eq.(1)}$$

$$(V . \nabla)V = -\nabla p + \mu \nabla^2 V + \beta \quad \text{Eq.(2)}$$

$$\nabla \cdot \left(\frac{|V|^2 V}{2} + eV + pV - \tau \cdot V - q \right) = 0 \quad \text{Eq.(3)}$$

where V is the velocity vector, p the pressure, μ the dynamic viscosity, β the thermal body forces, e the total energy, τ the stress tensor, and q is the conduction heat transfer. Back to our past experiences, see Refs. [4, 6, 25-27], this research also employs the widely accepted standard k-ε turbulence model to reproduce the true turbulent airflow dynamics within the ejector stack. Darbandi et al. [25-27] have conducted an in-depth exploration in this context. The standard k-epsilon model involves

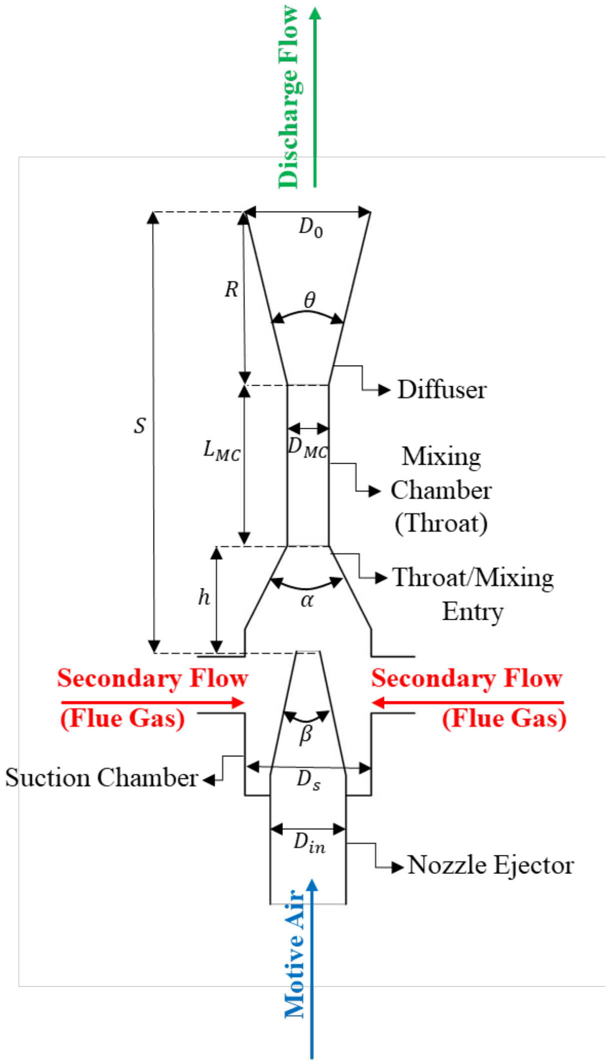

Fig. 1. A typical configuration close to the current ejector stack.

two transport equations, of which one governs the turbulent kinetic energy (k) and the other one the turbulent kinetic energy dissipation rate (ε). The conservation laws for the turbulent kinetic energy and its dissipation rate are given by

$$\frac{\partial(ku_i)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right] + 2\mu_t E_{ij}E_{ij} - \varepsilon \quad \text{Eq.(4)}$$

$$\frac{\partial(\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k} \quad \text{Eq.(5)}$$

$$2\mu_t E_{ij}E_{ij} - C_{2\varepsilon}\frac{\varepsilon^2}{k}$$

The above equations are simply called in the k-ε turbulence model. In these equations, u_i is the mean velocity component, μ_t the turbulent or eddy viscosity. The other variables are treated as the k-ε turbulent model constants. These constants are given by C_μ= 0.09, σ_k=1.00, σ_ε= 1.30, $C_{1\varepsilon}$= 1.44, and $C_{2\varepsilon}$= 1.92, which have been derived through extensive data fitting for diverse turbulent flows.

The present developed numerical tool uses the finite-volume method to treat the aforementioned governing equations. References [4, 6, and 25-27] provide details of computational modeling and the CFD method. As is raised in these references, a pressure-based solver is employed to solve the aforementioned steady-state governing equations. This solver integrates the energy equation and the species transport equation to simulate the mixture of H_2O, O_2, N_2, and CO_2 with varying volume fractions in distinct sections of the ejector stack. Moreover, the semi-implicit method for the pressure-linked equations (SIMPLE) scheme is utilized to couple the pressure and velocity fields suitably. As known, this scheme is a widely accepted approach for the steady-state solution algorithms.

2.2. Geometry, Meshing, and the Applied Boundary Conditions

Fig. 2. illustrates the actual geometry of the ejector stack, which is under investigation in the current study. The present ejector stack has six separate parts, namely the ejector nozzle serving as the inlet for the motive air propelled by a fan, a suction chamber, two inlets for the secondary flow (or the flue gas), a mixing entry nozzle, a mixing chamber, and a diffuser. In this figure, the ejector nozzle is represented by the red color, the flue gas inlets are colored in green, and the remaining components forming the stack walls are shown in blue.

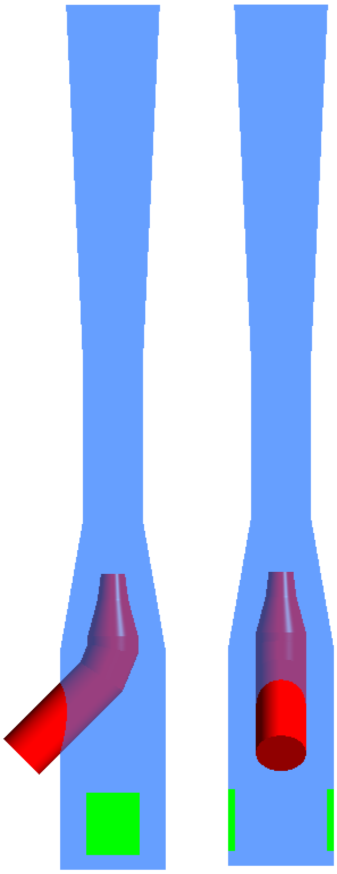

Fig. 2. The geometric model of the current ejector stack consisted of six separate parts.

As was mentioned in earlier sections, the objective of this research is to assess the impact of variations in several geometric parameters on the performance of the present ejector stack. We have chosen the flue gas flow rate as a key parameter, which can directly quantify the ejector stack's performance. So, the other interesting parameters such as the ejector entrainment ratio will not be addressed in this work.

The geometric parameters investigated in this research include the lengths of the mixing chamber and diffuser and the angles of the nozzle ejector, the nozzle diffuser, and the mixing entry, see Fig. 1. Table 1. describes their symbols and units. The table also provides the current sizes. They are referred to as the design values.

Table 2. describes the names of the chosen parameters and their ranges of variations. The selected ranges have been determined after a long investigation on choosing different values for them and a long discussion with the ejector stack operator. In other words, we have considered all the limits determined by the industry specialists, specifically, those who work in the DRI Unit. On the other hand, the original ranges have been much wider than those, which are reported in Table 2. However, the wide ranges do not necessarily help to achieve better-optimized geometry parameters. This means that the optimum geometry choices are very close to the original design cases. So, wider ranges for the selected parameters do not necessarily lead to more optimum values for them.

Fig. 3. illustrates the entire computational domain, the mesh distributed over the computational domain, and the major associated boundary conditions. As a part of the computational domain, a large red cylinder is positioned at the top of the stack outlet. It is because we intend to accurately implement the effect of the surrounding ambient in the simulations. One simple idea is to implement the outlet boundary conditions right at the exit plane of the ejector diffuser. It will certainly reduce the number of grid points. However, one should note that the simulation will miss the correct buoyancy effect, enforced by the warm or hot air at the top of the stack and its surroundings. On the other hand, one will certainly miss implementing the influences of environmental winds on the flow field inside the ejector stack. So, this study considers a large volume of space around the stack as the computational domain. Generally, four boundary conditions are implemented around the computational domain. 1- The air passing through the ejector stack is subject to an intake fan boundary condition. 2- A pressure boundary condition is applied at the flue gas flow intake.

Table 1. The selected geometrical parameters.

item	symbol	the selected parameter	design value	unit
1	L_{MC}	the length of the mixing chamber	7700	mm
2	R	the length of the diffuser	18915	mm
3	α	the angle of the nozzle ejector	19.8	deg
4	β	the angle of the diffuser	15	deg
5	θ	the angle of the mixing entry	5.39	deg

Table 2. The ranges of variations for the selected geometrical parameters.

Simulation Number	θ degrees	β degrees	α degrees	L_{MC} mm	R mm
1	4.5	0	14.85	5000	2125.28
2	4.9	10	16.15	5555.56	5077.06
3	5.3	12.5	18.15	6111.11	8028.84
4	5.39	15	19.8	6666.67	10980.6
5	5.7	17.5	23.1	7222.22	13932.4
6	6.1	20	26.4	7700	16884.18
7	6.9	22.5	29.7	7777.78	18915
8	-	-	-	8333.33	19836
9	-	-	-	8888.89	22787.7
10	-	-	-	9444.44	25739.5
11	-	-	-	10000	28691.3

3- An ambient pressure boundary condition is applied around the red color surface, which lets us observe the exit jet from the stack and phenomena beyond the design point at the stack exit. 4- The wall boundary condition is applied to the ejector stack and ejector nozzle walls.

Fig. 3. The computational domain, the outer boundary grid distribution, and the implemented boundary conditions.

Furthermore, a high-quality mesh is generated to effectively capture all the physics occurring within the domain. To achieve this, the grid undergoes compression in regions characterized by narrow geometry, high curvature, proximity to walls, and a substantial gradient of flow parameters. It is noteworthy that more pronounced mesh compression occurs at locations where the cross-section of the ejector stack and the ejector nozzle change.

To assess grid independence, the volumetric flow rate of the flue gas is monitored under design conditions for the ejector stack. Back to our past experiences, see Refs. [4, 6, 25-27], this research also employs $y^+<1$ to ensure the accuracy of the achieved numerical solutions. This should be fair if and only if this consideration does not lead to a grid with a tremendously large number of grid cells. So, it is necessary to carry on an accurate grid-independent study at this stage. In this regard, seven different meshes with varying sizes are generated to perform the present grid-independent study. Fig. 4. presents the volumetric flow rate of the flue gas considering a different number of cells. As is depicted in the figure, the solution is nearly independent of the mesh size when the number of cells is over 4.8 million. The concluding remark is that this number of grid points is sufficiently low to avoid considering $y+>1$, e.g. $y+=30$, in our calculations.

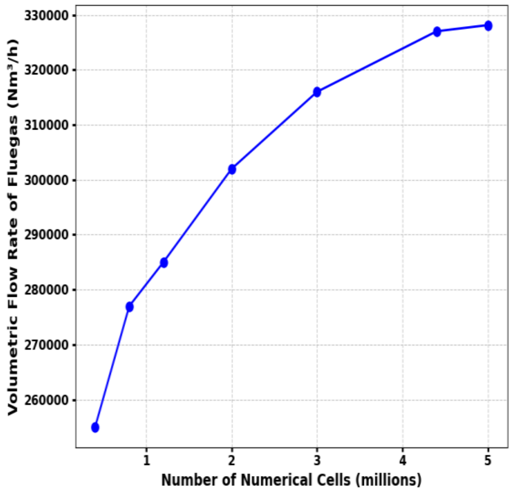

Fig. 4. Grid-independent study via monitoring the volumetric flow rate value of the flue gas versus the grid resolutions.

3. Results and Discussions

This section presents the results of present simulations and discusses the resulting achievements. The first step is to validate the CFD software code. The best validation is to model the ejector stack in the real working condition, i.e., off-design conditions, and compare the achieved numerical solutions with the online collected data, i.e., the measured data. However, it should be noted that there is no reliable measurement tool and no available measuring probe installed in the DRI unit of Esfahan's Mobarakeh Steel Company to measure the off-design conditions. So, it is not possible at all to validate the results of present numerical simulations experimentally. Fortunately and as was discussed previously, the DRI Unit has provided the ejector stack design working conditions, of which the volumetric flow rate of flue gas shows the most important one. The given design data indicates that the volumetric flow rate of the flue gas is about 327,000 (Nm^3/h) at the design working conditions. As was discussed previously in the mesh-independent study section, the present grid-independent study demonstrated that the volumetric flow rate of the flue gas would be around 327,000 (Nm^3/h) at design working conditions. So, the predicted numerical solution agrees well with the design value provided by the designer of the DRI Unit. So, the validation is carried out successfully. Despite the achieved validation, it is further possible to validate the current numerical solutions against the analytical solutions provided by Anderson [27] and White [28]. This has been carried on in this research. Fig. 5. illustrates the temperature distributions and Fig. 6. demonstrates the pressure magnitudes along the vertical direction, i.e., along the centerline of the stack. The presented plots in these two figures are broken into two parts by the three drawn vertical lines, which respectively indicate the beginning of the mixing chamber, the end of the mixing chamber, and the stack exit. As is seen in these two figures, the differences between the CFD solutions and the analytical solutions are minimal. The accuracy is more pronounced in the temperature distribution plot. The slight differences can be attributed to the many simplifying assumptions, which are necessary to apply in deriving the analytical relations. The numerical solution would be more accurate than the analytical relations due to its much less simplifying assumptions. Since the analytical relations are long, they are not presented here. It helps to shorten the length of this paper.

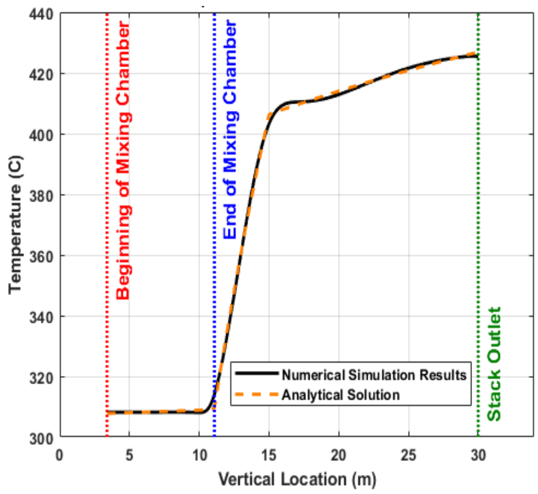

Fig. 5. Presenting the present numerical temperature distribution along the centerline of the stack and comparison with the analytical solution.

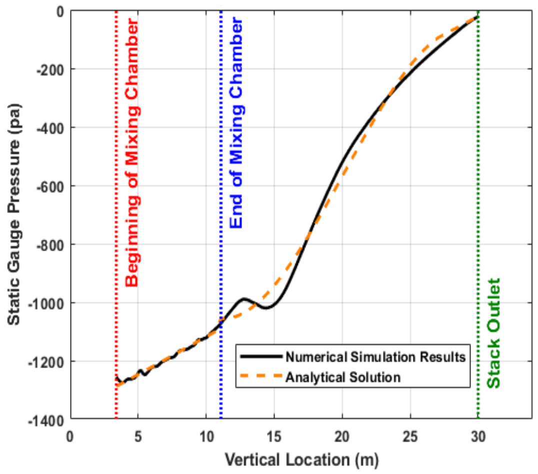

Fig. 6. Demonstrating the present numerical pressure distribution along the centerline of the stack and comparison with the analytical solution.

Fig. 7. displays the present numerical velocity contours at two longitudinal cross-sections of the ejector stack. The results are derived for the ejector stack in design conditions. As illustrated in these plots, the maximum velocity magnitude occurs right at the outlet of the nozzle ejector. So, it produces the necessary motive pressure gradient to effectively suck the flue gas into the stack. The current calculations indicate that the flue gas flow rate is approximately 144 kg/s in design conditions. We choose this flow rate as the reference flow rate to evaluate the positive/negative impact of the resized ejector stack on its resulting performance, see Tables 1 and 2. In other words, this reference flow rate establishes a suitable frame to determine the positive or negative impacts of the geometry parameter variations. It is said there will be an improvement in the ejector stack performance if the geometric parameter variation results in a stronger flue gas rate through the stack. These two plots indicate that the fluid flow pattern is not axisymmetric inside the ejector stack. It is quite evident because the ejector stack geometry and the implemented boundary conditions are not also axisymmetric.

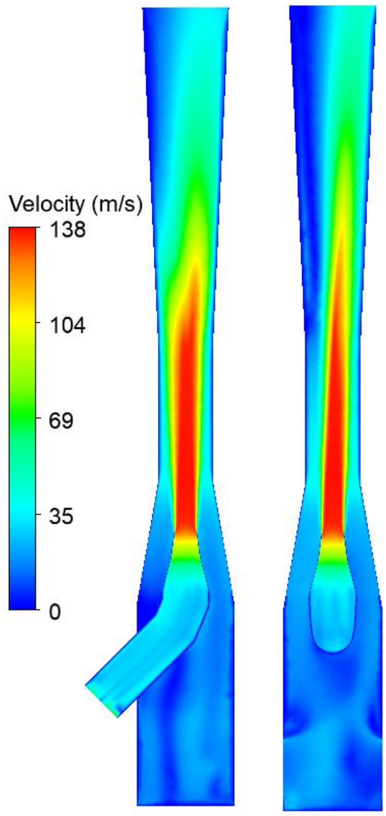

Fig. 7. Contours of the velocity fields in two longitudinal cross-sections of the ejector stack.

The next step is to examine the geometric parameter variation impact on the ejector stack performances, which is determined by calculating the flue gas flow rate. All simulations are carried on at in-design working conditions. As said before, these geometric parameters include the lengths of the mixing chamber and the diffuser, the angles of the nozzle ejector, the diffuser, and the mixing entry. These parameters are suitably varied within their specified ranges of variation, see Table 2. The results are illustrated in Figs 8 to 11. To have a better frame of comparison, the effects of different aforementioned geometrical parameters are evaluated separately.

Fig. 8. illustrates the impact of the mixing chamber length on the flue gas flow rate magnitude. The figure shows the design value as well, see Table 1. As is seen, increasing or decreasing this length does not lead to a contiguous trend of increasing or decreasing. In other words, there are several different maximum and minimum flue gas rates as the chamber length either increases or decreases. So, it is not generally advised to focus on this parameter as a serious remedy to improve the ejector stack performance. Hence, it is better to pay attention to the other geometric parameters.

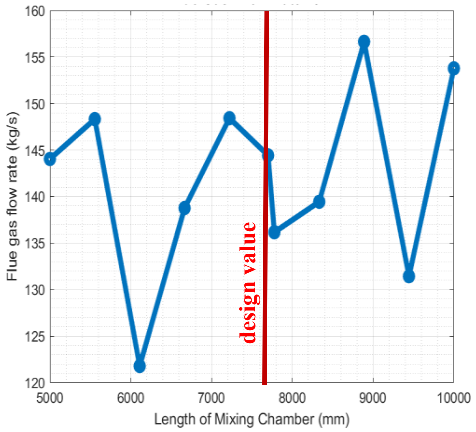

Fig. 8. Variation of the flue gas flow rate with the mixing chamber length changes.

The next step is to study the impact of the diffuser length variation on the achieved ejector stack performance. Contrary to Fig. 8. Fig. 9. shows that there is almost a smooth variation in the flue gas rate as the diffuser length increases. Considering the design point value, the figure shows that a decrease in the diffuser length will cause a continuous decrease in the performance, which is not in favor of this study. However, increasing the diffuser length, first shows a little decrease in the performance, which sounds inappropriate. However, the performance starts increasing at higher diffuser lengths. This can be attributed to the improvement in the pressure recovery magnitude. Inspecting the figure, indicates that a diffuser length of about 25 m can suitably enhance the achieved performance. This diffuser length will lead to a 10% increase in the flue gas rate, which is in favor of the present study. Unfortunately, the story will not be so good if one wishes to increase the diffuser length more. Due to the formation of flow vorticities and flow separation in the ejector stack, the rate of increase of the ejector stack performance becomes less and less. A diffuser length of 15 m will be very compromising if one does not wish to improve the performance of the present ejector stack.

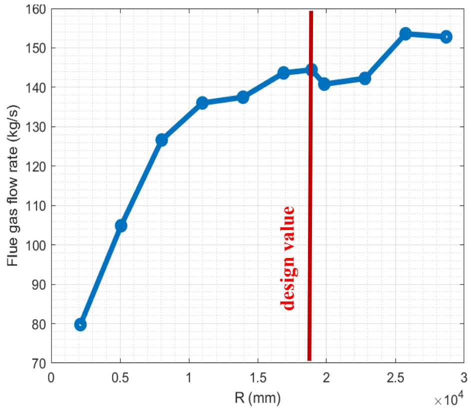

Fig. 9. Variation of the flue gas flow rate with the diffuser length changes.

The next step is to investigate the diffuser divergence angle impact on the achieved performance. Fig. 10. demonstrates the impact of the diffuser divergence angle changes on the flue gas flow rate value. The figure also shows the design value for this parameter. As before, the diffuser divergence angle has been either decreased or increased to see its consequences on the achieved performances. The figure shows that increasing the diffuser angle does not sound good because the achieved performance decreases continuously. This can be attributed to the flow separation occurrences in the ejector stack system. In contrast, a smaller diffuser divergence angle value can lead to a better ejector performance. However, there is a lower limit for the diffuser angle because the ejector stack performances show an abrupt decrease with little decrease in the diffuser length. This point should be very risky and should be avoided very cautiously. The figure indicates that the optimum diffuser divergence angle occurred at about 4.9 degrees. This angle enhances the ejector stack performance by about 4%. So this little increase in the performance suggests that this strategy does not sound great, specifically considering the instability of the ejector stack flow around this risky point.

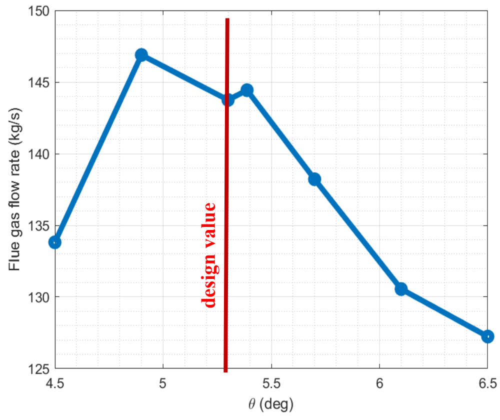

Fig. 10. Variation of the flue gas flow rate with the angles of diffuser changes.

The last step is to perform the sensitivity analysis on the impact of nozzle divergence angle variation on the achieved ejector stack performance. Fig. 11. illustrates the ejector stack performance in terms of the nozzle divergence angle. Again, the design value is demonstrated in this figure. The trends of changes are very similar to that in Fig. 10. The plot shows that the ejector stack performance gradually reduces as the nozzle divergence angle gradually increases, which sounds bad. However, a reduction in the nozzle angle improves the ejector stack efficiency. Unfortunately, there is a limit in decreeing this angle because there will be an abrupt decrease in the performance as the nozzle angle is further decreased, which is unsatisfactory. Generally speaking, the optimum angle for the divergence of the nozzle ejector is about 12.5 degrees. It leads to the highest ejector performance achievement, which is about 6.25%. It can be counted on this improvement.

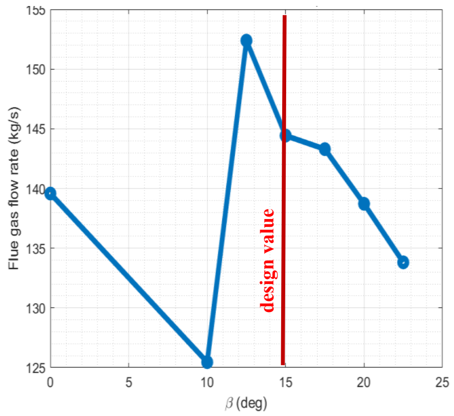

Fig. 11. Variation of the flue gas flow rate with the angles of the nozzle ejector changes.

The sensitivity analysis performed in Figs. 8-11. results in several conclusions. Fig. 9. indicates that if the diffuser length is increased by about 38%, the ejector performance will increase by about 10%. Fig. 10. indicates that if the diffuser angle becomes about 5 degrees, the flue gas rate will increase by about 4%. Fig. 11. also shows that if the nozzle ejector angle is adjusted at 12.5 degrees, the ejector performance will increase by about 6.25%. So, the best suggestion is to increase the diffuser length; however, this must be consulted with the industry specialists very carefully. However, it should be noted that this may require changing the fan because of a higher pressure loss along the stack. So, this should be checked very carefully before changing the diffuser heights.

4. Conclusions

Back to the inefficiency of an ejector stack system located in a Direct Reduction Iron (DRI) Unit, the main objective of this work was to possibly improve the efficiency of this system by improving the attributed geometric parameters of the ejector stack. The main purpose of this subsonic ejector stack is to expel the flue gas from the DRI Unit. After a careful study, five important geometry parameters were selected to perform the present sensitivity analyses, which were carried out using the CFD software and simulating the fluid flows passing through the ejector stack. The geometric parameters included the length of the mixing chamber, the length of the diffuser, the angle of the nozzle ejector, the angle of the diffuser, and the angle of the mixing entry. These parameters were suitably changed within their selected ranges and their impacts on the performance of the re-sized ejector stack were carefully monitored by calculating the volume rate of the flue gas, which was sucked by the re-sized ejector stack system. To validate the CFD software, the present numerical results were compared with the analytical solutions and the data provided by the manufacturer of the ejector stack. The agreements were excellent. The present simulations revealed the effects of varying the selected geometric parameters on the flue gas flow rate. It was shown that the increase or decrease in the mixing chamber length would not lead to a continuous trend of increase or decrease in the performance. So, it is not generally advised to focus on this parameter as a serious remedy to improve the ejector stack performance. However, the increase in the diffuser length would reliably enhance the performance. It was shown that if the diffuser length is increased by about 38%, the ejector performance will increase by about 10%. Further study showed that the optimal divergence angles would be about 4.9 degrees for the diffuser and 12.5 degrees for the nozzle ejector. It was shown that if the diffuser angle becomes about 4.5 degrees, the flue gas rate will increase by about 4%. On the other hand, if the nozzle ejector angle is adjusted at 12.5 degrees, the ejector performance will increase by about 6.25%. The study concluded that the best suggestion would be to increase the diffuser length; however, this must be consulted with industry specialists very carefully. The present literature review showed that the present study is novel and that it had not been fulfilled similarly in the other industries. So, the present study can be considered as the novelty of this work. Considering this novelty, the presented procedure can be equally applied in other industries, which suffer from their defective or inefficient ejector stacks.

Acknowledgment

The authors would like to acknowledge the financial support received from Esfahan's Mobarakeh Steel Company.

References

[1] Meakhail T, Zien Y, Elsallak M, AbdelHady S, Experimental study of the effect of some geometric variables and number of nozzles on the performance of a subsonic air–air ejector, Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy. 2008; 222: 809-818.

[2] Ariafar K, Buttsworth D, Sharifi N, Malpress R, Ejector primary nozzle steam condensation: Area ratio effects and mixing layer development, Applied thermal engineering. 2014; 71(1): 519-527.
[3] Ariafar K, Buttsworth D, Al-Doori G, Sharifi N, Mixing layer effects on the entrainment ratio in steam ejectors through ideal gas computational simulations, Energy. 2016; 95: 380-392.
[4] Darbandi M, Sabzpoushan S.A, Schneider G.E, Numerical study to evaluate the effect of phase change on performance calculation of a steam ejector, CFDSC Paper 2018-0539. The proceedings of the 26th Annual Conference of the CFD Society of Canada, (Ottawa, ON, Canada), CFDSC 2018, University of Manitoba, Winnipeg, Manitoba, Canada. 10-12, 2018.
[5] Ariafar K, Cochrane T, Malpress R, Buttsworth D, Pitot and static pressure measurement and CFD simulation of a co-flowing steam jet, Experimental Thermal and Fluid Science. 2018; 97: 36-47.
[6] Darbandi M, Sabzpoushan S, Schneider G.E, Geometrical optimization of a steam jet-ejector using the computational fluid dynamics, ASME-FEDSM Paper 2018-83203, Proceedings of the ASME 2018 5th Joint US-European Fluids Engineering Summer Conference (FEDSM2018), Montreal, Quebec, Canada. 2018; 2.
[7] Xiao J, Wu Q, Chen L, Ke W, Wu C, Yang X, Yu L, Jiang H, Assessment of different CFD modeling and solving approaches for a supersonic steam ejector simulation, Atmosphere. 2022; 13: 144.
[8] Watanabe I, Experimental investigations concerning pneumatic ejectors, with special reference to the effect of dimensional parameters on performance characteristics, Symposium on Jet Pumps and Ejectors. 1972: 97-120.
[9] Vyas B.D, Kar S, Study of entrainment and mixing process for an air to air jet ejector, The 2nd Symposium on Jet Pumps and Ejectors and Gas Lift Techniques, BHRA Fluid Engineering, Cambridge, England. 1975.
[10] Riffat S.B, Omer S, CFD modelling and experimental investigation of an ejector refrigeration system using methanol as the working fluid, International Journal of Energy Research. 2011; 25: 115-128.
[11] Riffat S.B, Jiang L, Gan G, Recent development in ejector technology- A review, International Journal of Ambient Energy. 2005; 26(1): 13-26.
[12] Yadav R.L, Patwardhan A.W, Design aspects of ejectors: Effects of suction chamber geometry, Chemical Engineering Science. 2008; 63: 3886-3897.
[13] Zhang X, Jin S, Huang S, Tian G, Experimental and CFD analysis of nozzle position of subsonic ejector, Frontiers of Energy and Power Engineering in China. 2009; 3: 167-174.
[14] Li C, Li Y, Wang L, Configuration dependence and optimization of the entrainment performance for gas–gas and gas–liquid ejectors, Applied Thermal Engineering. 2012; 48: 237-248.
[15] Yang X, Long X, Yao X, Numerical investigation on the mixing process in a steam ejector with different nozzle structures, International Journal of Thermal Sciences. 2012; 56: 95-106.
[16] Kong F, Kim H.D, Analytical and computational studies on the performance of a two-stage ejector–diffuser system, International Journal of Heat and Mass Transfer. 2015; 85: 71-87.
[17] Wang L, Yan J, Wang C, Li X, Numerical study on optimization of ejector primary nozzle geometries, International Journal of Refrigeration. 2017; 76: 219-229.
[18] Chen W, Huang C, Chong D, Yan J, Numerical assessment of ejector performance enhancement by means of combined adjustable-geometry and bypass methods, Applied Thermal Engineering. 2019; 149: 950-959.
[19] Tashtoush B.M, Al-Nimr M.A, Khasawneh M.A, A comprehensive review of ejector design, performance, and applications, Applied Energy. 2019; 240: 138-172.
[20] Yan J, Li S, Liu Z, Numerical investigation on optimization of ejector primary nozzle geometries with fixed/varied nozzle exit position, Applied Thermal Engineering. 2020; 175.
[21] Hadi M, Arshad A, Shaik N.B, Benjapolakul W, Gillani Q.F, Optimization of hydrocarbon ejector using computational fluid dynamics, Engineering Journal. 2022; 6: 5.
[22] Tavakoli M, Nili-Ahmadabadi M, Joulaei A, Ha M.Y, Enhancing subsonic ejector performance by incorporating a fluidic oscillator as the primary nozzle: a numerical investigation, International Journal of Thermofluids. 2023; 20: 100429,
[23] Schillaci E, Vera J, Oliet C, Vemula J.B, Duponcheel M, Bartosiewicz Y, Numerical modeling of air ejectors covering supersonic, subsonic and closed-port operations, Energy. 2024; 302: 131627.
[24] Kumar A, Yadav S, Kumar V, Kulkarni A, A comprehensive exploration of ejector design, operational factors, performance metrics, and practical applications, Journal of the Brazilian Society of Mechanical Sciences and Engineering. 2024; 46(39).
[25] Darbandi M, Jalali R, Schneider G.E, Robust 1-D fluid flow and heat transfer predictions in gas turbine cooling passages," AIAA Paper 2019-4057, AIAA 2019 Propulsion and Energy Forum and Exposition, Indianapolis, Indiana, USA, August. 2019: 19-22.
[26] Darbandi M, Jalali R, A cost-effective approach to generate accurate correlations via analyzing a minimum number of data points; Case study on the convection heat transfer problem, The Canadian Journal of Chemical Engineering. 2022; 101: 3634-3647.
[27] Darbandi M, Jalali R, Internal cooling sensitivity analysis to improve the thermal performance of gas turbine blade using a developed robust conjugate heat transfer method, International Journal of Engine Research. 2022; 24: 949-964.
[28] Anderson J, Fundamentals of Aerodynamics, 6th ed., McGraw-Hill Education, 2016.
[29] White F.M, Fluid Mechanics, 7th ed., McGraw-Hill Education, 2011.