British Journal of Applied Science & Technology
10(3): 1-12, 2015, Article no.BJAST.18078
ISSN: 2231-0843

SCIENCEDOMAIN *international*
www.sciencedomain.org

Application of Double-Duct Idea to Replace Conventional Bell-mouth Intakes

Morteza Abbaszadeh[1], Hafez Ejlali[2] and Mina Shahrooz[3*]

[1]Boğaziçi University, 34342 Bebek, Istanbul, Turkey.
[2]Sharif University of Technology, Azadi Blvd, Tehran, Iran.
[3]Universitat Politècnica de Catalunya, Av Diagonal, 647, 08023 Barcelona, Spain.

Authors' contributions

This work was carried out in collaboration between all authors. Author MA designed the study, performed background checks and supervised the study and wrote the manuscript, author HE performed numerical tests and statistical analysis, author MS performed validation tests, contributed to interoperations of outcome data, and edited the text. All authors read and approved the final manuscript.

Article Information

DOI: 10.9734/BJAST/2015/18078
Editor(s):
(1) Yu Hai-Liang, School of Mechanical, Materials and Mechatronic Engineering, University of Wollongong, Australia.
Reviewers:
(1) Anonymous, Technical University of Liberec, Czech Republic.
(2) Anonymous, Central Institute of Aviation Motors, Moscow, Russia.
(3) Ali Helali, Department of Technologies and Transport Engineering, University of Sousse, Tunisia.
Complete Peer review History: http://sciencedomain.org/review-history/9890

Original Research Article

Received 1st April 2015
Accepted 29th May 2015
Published 20th June 2015

ABSTRACT

Aims: Controlling specifications of entering flow to a system is a critical issue in preventing any unexpected drop in the system efficiency. To provide the most satisfactory flow stream for internal combustion engines, jet engines, wind tunnels, air conditioning systems, water or oil intakes and thousands of other diverse applications, up until now the main interest was to design or optimize Bell-mouth intakes. Here we propose an innovative approach. Based on a novel idea proposed by Vertical Lift Research Center of Excellence in Pennsylvania State University, this investigation, for the first time, suggests replacing conventional Bell-mouth intakes by a new geometry called Double Duct (D-duct) in order to enhance performance of existing systems

Methodology: With regard to previous investigation, K-epsilon turbulence model was utilized through a Commercial Software package in this numerical investigation. Beside the reliability of the numerical method, which was declared by previous investigators working on similar applications, the method was also validated by comparing it with experimental data gained from one of the

---

*Corresponding author: E-mail: mina.shahrooz@gmail.com, mina.shahrooz@estudiant.upc.edu;

research references. An initial configuration of D-duct intake was compared to an elliptical Bell-mouth intake operating on leading edge of a pipe with a pressure intake at its end.

Results: Final results demonstrated nearly 8 percent increase in discharge coefficient and mass flow rate of D-duct system. Close observation of velocity distribution diagrams along the pipe length also proved that there is a noticeable difference between the flow in a conventional Bell-mouth system and a D-duct configuration.

Conclusion: D-Ducts could be used for improving flow characteristics in in-takes and to replace conventional Bell-mouth intake. However due to revolutionary nature of this idea and fundamental changes in the geometry a series of new investigations such for determination of vibration of the second duct etc. would be required.

Keywords: Double duct; bell-mouth intake; CFD; turbulence model.

1. INTRODUCTION

Unexpected behaviors of the mechanical systems, particularly those causing considerable drop in performance of the system, have always been a subject of high consequence for design of engineering systems In particular, in systems which use fluids or gases of the surrounding medium, many difficulties pop up regarding the behavior of fluid s and control of flow stream entering the apparatus which maintains the performance of the system as a whole in a reasonable range. In applications such as internal combustion engines, jet engines, ducted industrial fans, wind tunnels, ventilation systems, water pumps, water dams, and numerous other applications, one of the greatest concerns, is to prevent the entering flow to cause unwanted disturbance and vibration, and a tremendous slump in pressure. The key to achieve this lies in using a proper geometry in the entrance part. Common and widely-used solution to prevent such inconveniencies is applying a proper bell-mouth entry.

Design and optimization a bell-mouth intake did not prevail much among independent investigations. But studying any system which requires suction of any kinds of fluid in the apparatus reveals outstanding attempts made in controlling behavior of the entering stream. For instance, in case of internal combustion engines G.P. Blair and et al. studied extensively on an optimum shape of a bell-mouth air intake through numerical and experimental procedure [1,2]. Based on the findings of these researchers, they recommended an elliptical shape as the best profile to utilize in bell-moth intakes. These studies also provided a highly valuable set of numerical and experimental data. All these made the prediction of behavior of Bell-mouth intakes possible. Therefore, optimal shapes in different functions can be chosen. Later Ismael and Abu Bakr [3] investigated effectiveness of bell-mouth intake design in the performance of a diesel engine by interest set on discharge coefficient as the representative of intake efficiency. The result was that by applying a proper intake, the performance of such systems could be increased up to 10 percent.

Due to a higher sensitivity and extreme conditions in which they are operating studies in the field of aviation focused mainly on air intakes. One of the most interesting fields of study has been ducted fans systems. Ducted fans are used in many ways such as turbines, jet engines, planes, air conditioning systems etc. Due to effects of tip vortex on performance, vibration, and wake in turbo machines such as axial flow fans and propellers, it has always been emphasized that the problem of air intake design is to ensure that the turbine or fan are properly supplied with enough amount air under all conditions of operation with an acceptable level of pressure loss. For example, the intake of a gas turbine engine is an important component, which directly interferes with the internal air flow of the engine and affects its performance characteristics.

As it is clear, there is an intake separation at the input end of the pipe that creates disturbance in the air flow. This separation and disturbance are both rooted in the nature of suctioning free stream into a pipe or a ducted area and can bring about vibrations and noise in the fan and so consequently increase the potential drop and extremely reduce the mass and quality of the incoming air. So in that case, a more powerful fan is needed to counterbalance this amount of loss. This situation is not at least desirable in creating and sustaining high performance or in enormous systems like underground air conditioning. In the present day, ducted fan performance issues related to inlet lip separation still intensify and increase the overall technical challenges of the systems.

Most of the attempts in this field were in the form of analysis of input flow to fans or jet engines. The efforts made by Andersen [4], Briley [5] and levy [6,7] illustrated a clear pattern of merits and demerits of t use of C.F.D. methods in analyzing the flow stream inside an intake. On the other hand, researchers like Vakili [8] tried to compare the results of numerical tests with experimental models. For instance, C. E. Towne and E. F. Schum [9] analyzed the flow in the subsonic diffuser section of a typical modern inlet design. They studied the effect of curvature of the diffuser centerline and transitioning cross-sections to determine the primary cause of flow distortion in the duct. Their conclusion was based on reports of total pressure values in the engine compressor face. These efforts made it possible the use of computational methods in evaluation ring of optimization process hence one of the recent interests in this field was how to optimize air intakes with the use of computational methods. Graf et al. [10] enhanced ducted fan edgewise flight performance using a newly-designed leading edge geometry; a significant factor in offsetting the effects of the adverse aerodynamic characteristics.

The efforts that have been made to develop efficient flow intakes are not enclosed in two aforementioned fields or air as entering fluid. But the overview is quite sufficient to highlight the importance of flow intakes. This study exploits a new opportunity to furthermore improve performance of existing low speed intakes.

2. D-DUCT IDEA

The “Double Ducted Fan” (DDF) idea was developed by Akturk and Camci in Vertical Lift Research Center of Excellence in Pennsylvania state university in 2011 [11]. They worked on a computational study of a novel ducted fan inlet flow conditioning concept that would immensely improve the performance and controllability of VTOL (Vertical take-off and landing) UAVs and many other ducted fan based systems. Their purpose was to reduce the inlet lip separation in the edgewise flight zone using a secondary stationary ducted system which was self-adjusted in a wide edgewise flight velocity range. Their DDF concept applied a second duct with the shape of a lip airfoil that has a much shorter axial chord length than that of the standard duct. In a conventional fan inlet, the inlet flow distortion near the leading side becomes more problematic with the increasing vehicle speed and this severely limits the lift generation and controllability of VTOL UAVs. Distorted inlet flow prompts an asymmetrical overloading of the ducted fan which consequently increases the power required for level un-accelerated flight and noise level. In their paper, they compared the conventional baseline duct to two different double ducted fans cases. They also considered both hover and edgewise flight conditions. Fig. 1 illustrates velocity contours of one of the tests with a 90° angle of attack for a conventional duct and the case B.

They demonstrated that the poor edgewise flight characteristics of the reference duct as shown in Fig. 1(a) were effectively improved with the “Double Ducted Fan” concept as presented in Fig. 1(b). In this figure it can be seen that the Double Ducted Fan (DDF) flow simulations indicating the effective inlet flow distortion reduction due to the unique aerodynamic properties of the (DDF) system. As a result, they showed that the upstream lip separation near the leading side is almost eliminated, resulting in a more balanced rotor exit flow field between the leading side and the trailing side. As a conclusion, they declared that the Case-B was the best DDF configuration design. It has improved the mass flow rate passing from the duct by 40% and improved thrust force obtained from the ducted fan by almost 56% relative to baseline duct in edgewise flight condition. They also added that the immediate impact of DDF concept was in the reduction of fuel consumption of the flight vehicle or improved range. The elimination of the inlet lip region re-circulatory flow and its interaction with the rotor minimizes vibratory loads on the DDF based vehicle; moreover, in this case the fan exit control surface effectiveness was improved on account of increased rotor exit axial momentum and better uniformity of DDF exit flow.

In the first glimpse, the DDF idea may seem extraneous to the flow intakes subject, but in a closer look both systems have same goal. Bell-mouth intakes and DDFs both are supposed to eliminate unwanted velocity factors of incoming flow. In fact DDF deals with a critical version of the problem that Bell-mouth intakes are supposed to solve. This elegant point of view was the initiative of present investigation.

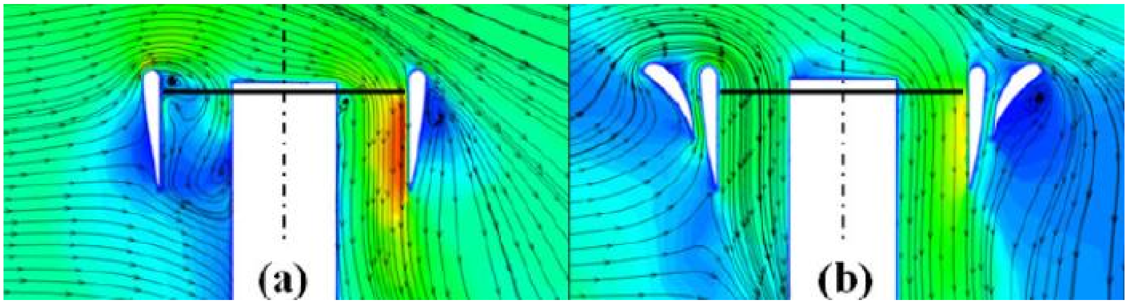

Fig. 1. D-duct idea implicated on a vertical lifting system [1]

3. MODELS AND NUMERICAL METHODS

This investigation was based on the use of numerical methods to acquire characteristics of each model. To ensure that the best numerical tools are being used, a series of previous attempts were studied meticulously in order that full supervision over each aspect of the numerical tools could be gained. The chosen numerical method was used in a conventional manner and one of the priorities of this study was selecting a simple and efficient means to acquire data from the models.

3.1 Models

The main goal of this investigation was to compare performance of a conventional Bell-mouth intake with D-duct geometry. Main path to achieve such purpose is modeling one system of each kind with similar design proficiency and comparing models' performance with different working conditions. In this study, intakes were supposed to perform in leading edge of a pipe with a length of 30 cm and a diameter of 20 cm, surrounded by air with a pressure intake at the end of the pipe. Such application is similar to numerous applications of air intakes such as internal combustion engines, jet engines and wind tunnels.

Bell-mouth intakes usually would be characterized by certain parameters: "Type", "length L", "exit diameter De", "entry diameter Di" and "entry corner radius RC". The "type" can be sharp edged plain pipe (PP), a simple radius (RAD), an aerofoil profile bell mouth (AER) or an elliptical profile (ELL). A bell-mouth intake would be profiled like this:

TYPE-L-De-Di-RC

Efforts have been made to compare performance of bell-mouths with different shapes [1,2,3]; moreover, certain recommendation have been made to help enhance the performance but there is not any specific geometrical equation(s) to calculate the exact shape of the curvature. The most important recommendations are first to keep the length of Bell-mouth intake equal to its exit diameter and secondly this that using elliptic type would create the best performance. The Bell-mouth intake model which was utilized in this investigation (Fig. 2) was an elliptical type with a length of 20 Cm equal to the pipe diameter with entry corner radius of 3 cm.

A double duct configuration according to its nature is defined by multiple parameters. From a geometric perspective, the most complicated part of a D-Duct system is the airfoils which are applied in the entrance. So in any investigation the parameters which define the airfoils like maximum thickness, place of maximum thickness on chord and etc. should be notified. Due to tip lines of Bell mouth systems' design, chord of the airfoil which would be denoted as a portion of pipe diameter, is a very important parameter too. Another characteristic of D-Duct systems is the gap between the main pipe and the secondary duct (the airfoil). This space between two ducts damps a considerable amount of developing boundary layer around the airfoil. The gap overlap of the secondary duct and pipe should be notified as well. As it is quite evident, there are more parameters than bell-mouth intakes in double duct systems, which make the design and analysis of such systems more complicated. But in this investigation an initial geometry was determined to compare with the bell-mouth intake through an iteration process (Fig. 3). Cross section of secondary duct was NACA-2412 airfoil with chord of 15 cm and gap of 2 cm.

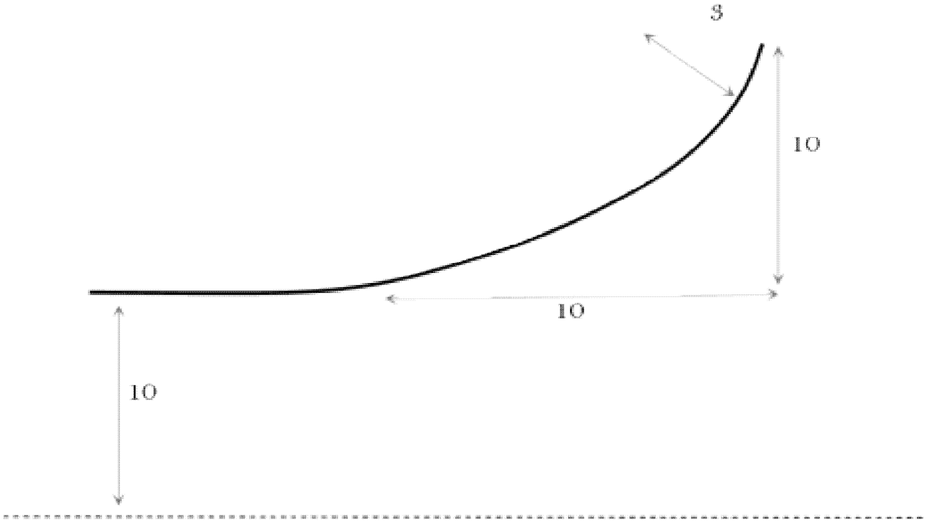

Fig. 2. The designed bell-mouth intake configuration for numerical tests

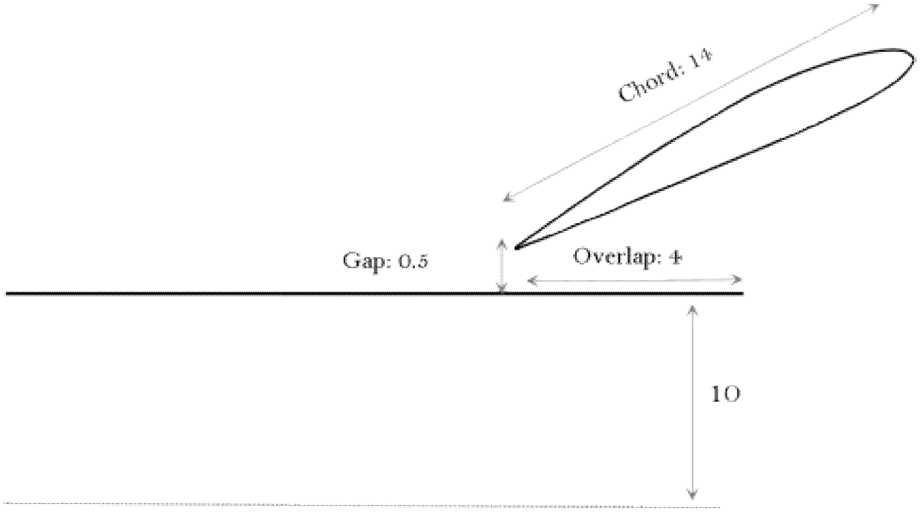

Fig. 3. The designed D-duct intake configuration for numerical tests

3.2 Numerical Methods

By studying previous investigations [12,13] K-ε: Standard turbulence model to achieve data from models through ANSYS fluent Commercial software package. Fluent is a CFD software using the finite sizes method and having the capacity of solving the flow and heat transfer problems in or around complex geometries. In the present Standard k–є turbulence model has been utilized with logarithmic surface function.

The equations of mass and momentum protection used by the program can be written for the compressible and incompressible steady flows as follows in the cartesian tensor rotation:
Continuity equation:

$$\frac{\partial}{\partial x_j}(\rho . u_j) = 0 \tag{10}$$

Momentum equation:

$$\frac{\partial}{\partial x_j}(\rho . u_j . u_i - \tau_{ij}) = \frac{\partial p}{\partial x_j} + S_i \tag{11}$$

In these two equations we have:

x_i Cartesian coordinate (j=1, 2, 3)
u_i Absolute velocity components in the direction of x_i .
p Piezometric pressure = $p_s - \rho_0 . g . x_m$ here, p_s is static pressure, ρ_0 is the reference dencity, g is the gravity acceleration andx_m is the coordinate defined by ρ_0
τ_{ij} Stress tensor components

Here, the stress tensor is as follows:

$$\tau_{ij} = \mu . s_{ij} - \frac{2}{3} \mu . \frac{\partial u_k}{\partial x_k} . \delta_{ij} \tag{12}$$

Here, μ is the viscosity of the fluid, δ_{ij} (Kronecker delta) and s_{ij} is the change of shape modification tensor and written as follows:

$$s_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_i} \tag{13}$$

If the Kronecker delta δ_{ij} is

$$i \neq j \Rightarrow 0, i = j \Rightarrow 1$$

Effective viscosity is:

$$\mu_{eff} = \mu + \mu_t \tag{14}$$

Here, turbulent viscosity is obtained from

$$\mu_t = \rho . f_\mu . C_\mu . \frac{k^2}{\varepsilon} \tag{15}$$

Turbulent kinetic energy (k):

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}\left(\rho u_j k - \frac{\mu_{eff}}{\sigma_k} . \frac{\partial k}{\partial x_j}\right) = \mu_t s_{ij} \frac{\partial u_i}{\partial x_j} - \frac{2}{3}\left(\mu_t \frac{\partial u_i}{\partial x_i} + \rho k\right) \frac{\partial u_i}{\partial x_i} - \rho \varepsilon \tag{16}$$

Dissipation rate of turbulent kinetic energy (ϵ):

$$\begin{aligned} &\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}\left(\rho . u_j . \varepsilon - \frac{\mu_{eff}}{\sigma_\varepsilon} . \frac{\partial \varepsilon}{\partial x_j}\right) \\ &\qquad = C_1 f_1 \frac{\varepsilon}{k}\left[\mu_t s_{ij} \frac{\partial u_i}{\partial x_j} - \frac{2}{3}\left(\mu_t \frac{\partial u_i}{\partial x_i} + \rho k\right) \frac{\partial u_i}{\partial x_j} \delta_{ij}\right] - C_2 f_2 \rho \frac{\varepsilon^2}{k} - C_3 \rho \varepsilon \frac{\partial u_i}{\partial x_i} \end{aligned} \tag{17}$$

In the above equations, the subscripts i, j, k and the empirical constants for the turbulence model $C_\mu, \sigma_k, \sigma_\varepsilon, C_{1\varepsilon}, C_{2\varepsilon}$ are equal to 1, 2, 3 and 0.09, 1.0, 1.3, 1.44, 1.92, respectively.

3.3 Validation

C.F.D. methods were used occasionally to analyze flow quality and specifications in pipes and this proved to be reliable in various applications [12,13]. Despite reliability of C.F.D. methods in analyzing flow characteristics in pipes, in this study in order to validate the use of the turbulence model, which was discussed before, a series of Grid independency and validation tests were conducted. Based on the grid independency tests a proper size was chosen for the grid which surrounds the models. Final Spacing of the utilized grid in this study was 0.5 Cm.

According to G.P. Blair and B.H. Lau [2] the numerical method also could be validated by comparing it with experimental tests. Blair and Lau conducted a series of experimental tests to measure discharge ratio on a specific pipe with different shapes of intakes. In order to justify use of numerical method, one of their test subjects was modeled through the numerical method using Fluent Software package. The P-46-23-23-0 model is a simple pipe having no specific intake shape with 26mm diameter which was tested in different pressure ratios ranging from 1.03 to 1.1 by Blair and Lau. As it is obvious from Fig. 4, comparison of results of experimental and computational tests of P-46-23-23-0model were fairly close and demonstrated that the application of the numerical method is adequate to analyze flow characteristics in pipes and intakes.

4. RESULTS

Depending on application of a flow intake they are evaluated by different parameters. In application for internal combustion engines or jet engines, intakes are supposed to provide a satisfactory amount of air so the main parameter would be Discharge Coefficient or mass flow rate. In the other hand, in more sensitive applications like wind tunnels intakes or some of the ventilation systems the main task is to provide flow stream with the lowest rate of turbulent so that the main parameter to consider for intake evaluation would be the behavior of velocity distribution diagrams or turbulence intensity in different parts of intake. In order to cover all the aspects, results were classified/ categorized in two groups: Performance and flow behavior.

4.1 Performance

As it was mentioned, in industrial applications like design of intakes for engines (Internal combustion or Jet), oil or water pump intakes and etc. main task is to provide a satisfactory amount of flow. To compare capability of different intakes to provide enough flow in different models or applications, usually their discharge coefficient and mass flow is taken into account.

Usually using a Bell-mouth on an intake provides nearly 30 percent increase in C_D at the most ideal circumstances including in case of using optimum shape of Bell-mouth in that specific application. In ordinary bell-mouth intakes C_D is a function of pressure ratio and by increasing suction strength discharge ratio will grow even by 20 percent in some conditions. Looking from a geometric aspect, discharge ratio of an intake is a function of its entry diameter and sharpness of bell-mouth intake profile. According to Blair best bell mouth profile would have less than 5 percent advantage to any other profile in a specific application [1].

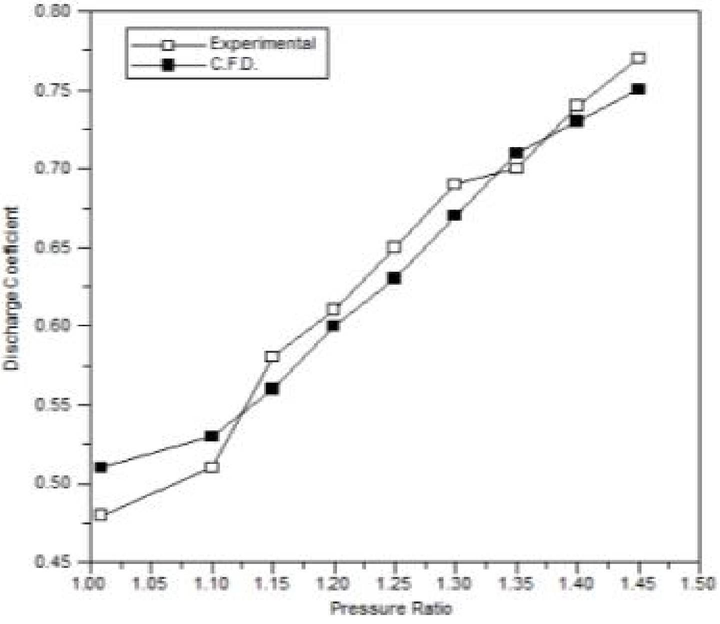

Fig. 4. Comparing results of C.F.D. tests with experimental values of ref [2]

Discharge coefficient is usually measured against the pressure ratio in pipes, but in this investigation output pressure at the end of the models has been set in a reasonable range to act like a fan or pump so the more realistic results could be achieved. The pressure range was set from 20 Pascal to 200 Pascal with a 5 Pascal step between each test.

As it has set in Fig. 5, results of this study indicated that application of a D-duct intake would have nearly 8 percent increase in discharge coefficient. Even with assuming that the bell-mouth model does not have the most optimum shape, the amount of advantage that is observed in the results is more than expectable advantage of an optimum Bell mouth to any other. Hence a not-optimized D-Duct entry would have a significant advantage to an optimum bell-mouth entry. The amount of the advantage is not solid; however, a poorly designed d-duct might have even disadvantages compared to a bell-mouth intake while a better designed D-duct entry would have more advantages. Other important point in Fig. 5 is the variance of C_D in a D-duct intake through different suction strengths. While the variance of maximum and minimum C_D in bell-mouth model was nearly 30 percent, same factor in D-duct entry was 15 percent. This confirms the claim that the function of a D-duct entry is steadier than a Bell-mouth intake.

Mas flow rate ($\dot{m}$) also demonstrated same advantages as discharge coefficient Fig. 6. Same differences in behavior of profiles could be observed.

Pressure counters of models were also interesting. Studying pressure contours, Such as presented in Fig. 7, demonstrated that the contraction zone or as Blair states “vennacontracta” is happening in deeper axial locations in D-duct intake. While the contraction zone is located in leading edge of Bell-mouth intake, in D-duct model, flow contraction is found in the pipe itself. Contraction zone doesn’t count as a beneficial area in pipes; however, since axial velocity, temperature and pressure get a considerable temporary boost in this section, D-duct might be an advantage in low length pipes.

Using D-duct entry in low length pipes which provide required flow for heating systems or jet intakes in some circumstances enables the designers to apply a slight boost in entering flow temperature or velocity by controlling location of contraction zone. This eventually leads to the enhancement of performance of the main system.

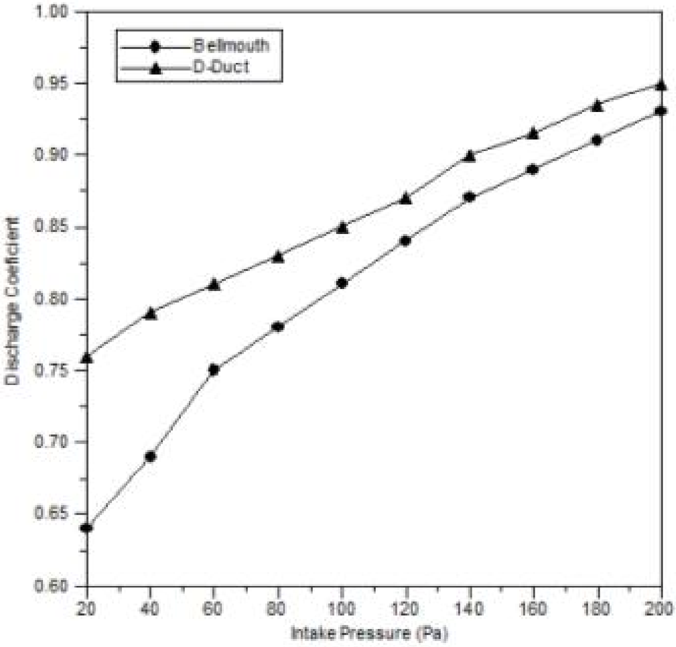

Fig. 5. Discharge coefficient of models

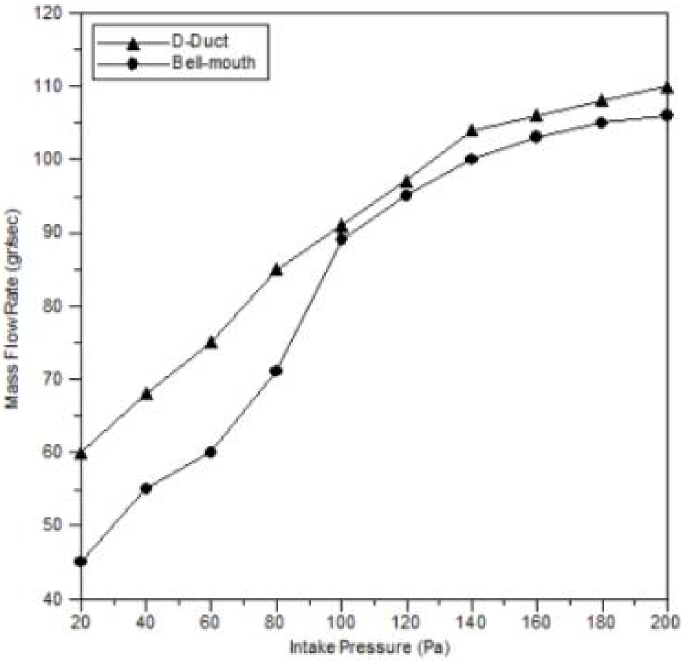

Fig. 6. Mass flow rate in two models

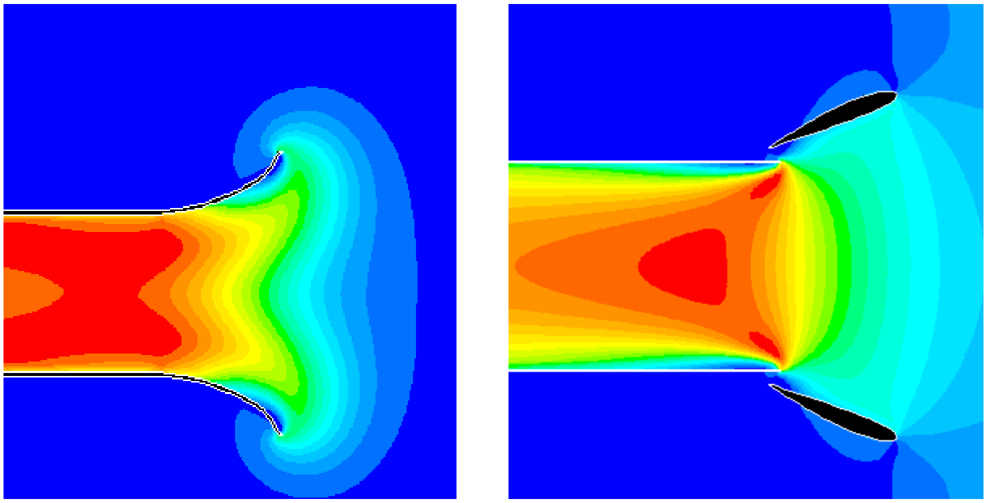

Fig. 7. Pressure contours around designed models

4.2 Flow Behavior

In more sensitive applications such as intake of wind tunnels, intake of high performance industrial or aviation fans or vibration-sensitive, especially those operating under the water, the foremost interest is to provide a steady and calm flow stream for the main system. The amount of provided flow and energy conservation are still important in such systems but they would be considered as the secondary parameters to evaluate the system. In this article, in order to prevent any confusion and present a clear definition about the flow behavior, relevant parameters will be studied in only one of applied output pressure (70 Pa).

The main means should be employed in order to study the behavior of flow in pipes or flow intakes is to observe velocity distribution diagrams in the pipe or intake. The study of velocity distribution diagrams can point out the development of boundary layer and any unexpected separation. Figs. 8 and 9 demonstrate changes in axial velocity distribution diagrams in different axial locations of the intakes. Fig. 10 presents axial velocity distribution on a virtual line located on the intakes leading edge and Fig. 11 demonstrates velocity distribution on middle of the intakes.

In Intake, velocity distribution profiles demonstrate the growth of boundary layer. The distribution profiles of D-Duct intake show a parabolic shape while signs of disturbance and turbulence in profiles of Bell-mouth intake are observable.

Lower turbulence in intake sections is due to the nature of flow behavior around aerodynamic shapes like NACA 2412 airfoil. A lesser turbulence in this area means producing fewer vibrations and noise by intake. Hence D-Duct downstream and its attachment to the main apparatus, especially wind tunnels and other sensitive applications, would have less distractive effects. The maximum axial velocity and mean axial velocity in this area are functions of contraction and intake cross section area in any specific axial location. Furthermore, it can be said that the variance of these parameters in intake is not a significant sign.

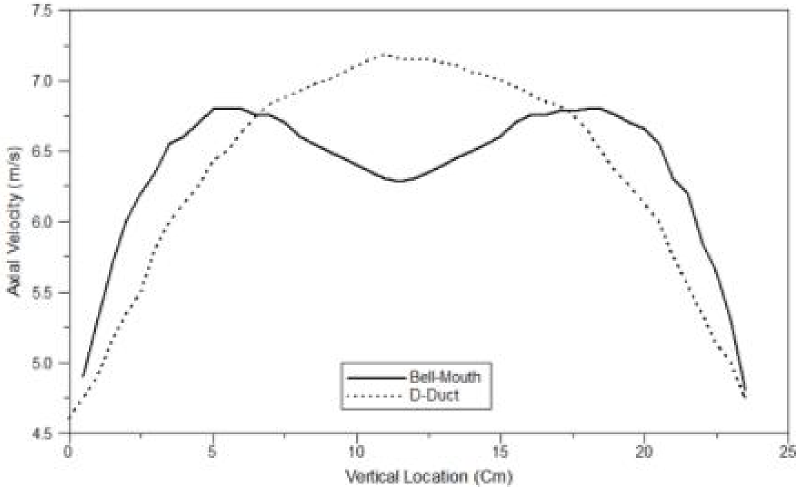

Fig. 8. Axial velocity distribution diagrams on intakes leading edge

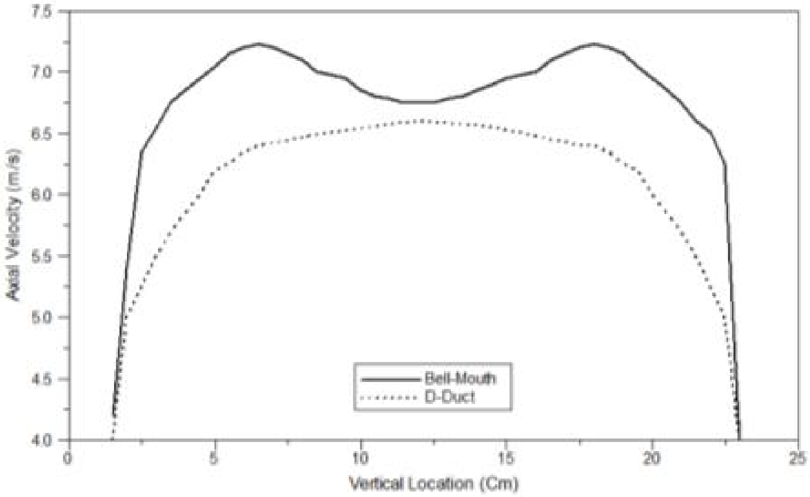

Fig. 9. Axial velocity distribution diagrams in middle of intakes

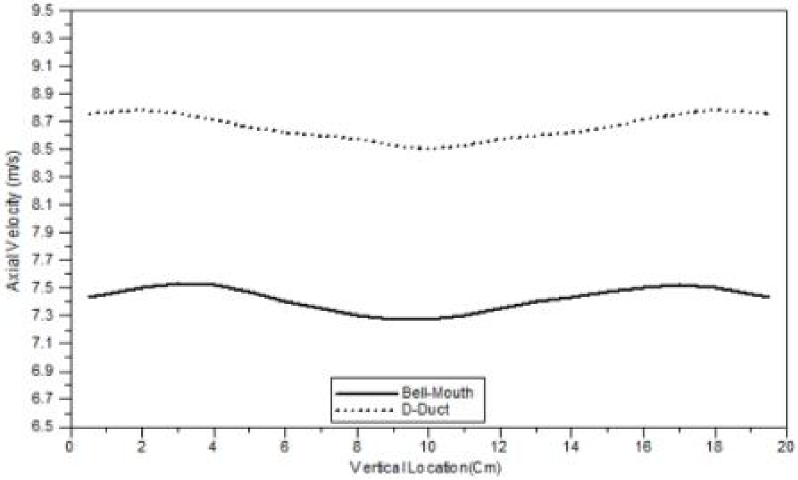

Fig. 10. Velocity distribution profiles on the pipe leading edge

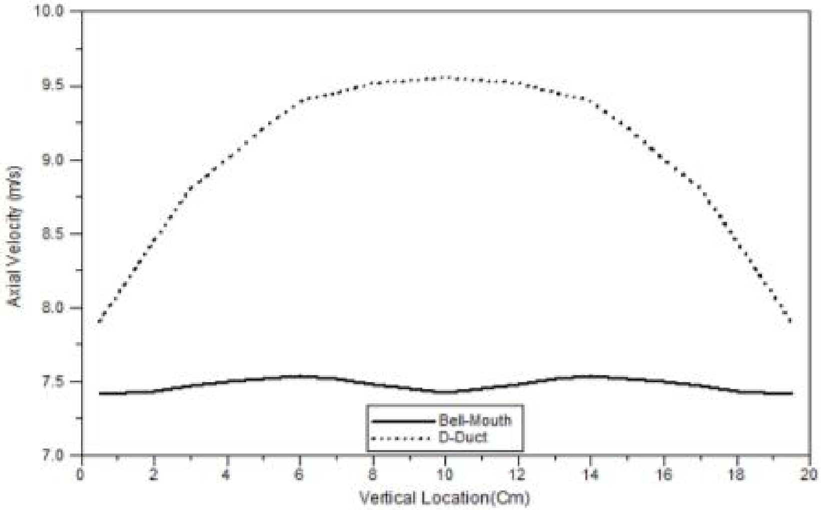

Fig. 11. Velocity distribution profiles in middle of the pipe

The interesting point in comparing velocity distribution profiles is their behavior inside the main pipe. Evidently as illustrated in Fig. 10, by entering main pipe due to the gap between main pipe and the D-duct a considerable disturbance is being produced, probably because of sudden damping of developing boundary layer around the D-Duct. In main pipe leading edge behavior of two profiles are almost the same. However, while in middle of the main pipe the bell-mouth distribution profiles are still suffering from severe turbulences, velocity distribution profile of D-Duct intake shows a fully developed laminar flow.

According to the experiences of leading authors in design of axial fans [14,15] such behavior of flow in D-duct model and having a considerable area with predictable velocity, is a relief for any ducted fan designer as it reduces complicity of calculating velocity measures entering each radial element of blade, specially induced velocities. The reduction of disturbance produced caused by the gap will be a promising subject for further studies on D-Duct in takes.

5. CONCLUSION

A D-Duct intake was compared to a conventional Bell-mouth intake and then the advantages of such application in different working regimes was described and analyzed. Expectations of a proper intake from different points of view were described; furthermore, the specifications of D-Duct and Bell-mouth intake were investigated with regards to each kind of concerns. In addition to aerodynamic advantages that have been counted, this is also worth noting that the nature of D-Duct intake, the gap and overlap between the main pipe and the second duct and the use of an aerodynamic cross-section can bring about new opportunities such as application of flow filters, heaters, fluid injections and else. Eventually, considering the results of this investigation, well-known authors recommended utilizing Double-Duct intakes instead of bell-mouth intakes. This article just serves as a kind of initiation of the use of D-Duct intakes; yet far more investigations in several different topics are needed in order to acquire a full understanding of such systems. A further attention should be given toward unsteady behavior of flow inside the canal and on the intake that could affect vibrations and ultimately flow quality of the main canal.

COMPETING INTERESTS

Authors have declared that no competing interests exist.

REFERENCES

1. Blair GP. Design and simulation of four-stroke engines. Society of Automotive Engineers, SAE reference R-186. 1998;813.
2. Blair GP, Lau HB, Cartwright A, Ragunathan BD, Mackey DO. Coefficients of discharge at the apertures of engines. SAE International Off-Highway Meeting, Milwaukee; 1995.
3. Ismail AR, Bakar RA, Semin. The effect of discharge coefficient of intake and exhaust valve for direct injection engine. International Conference on Mechanical Engineering; 2007.
4. Anderson BH. Three-dimensional viscous design methodology for advanced technology aircraft supersonic inlet systems. NASA 1M 83558, AIAA Paper 84-0192; 1984.
5. Briley WR, McDonald H. Analysis and computation of viscous subsonic primary and secondary flows. AIM Paper 79-1453; 1979.
6. Levy R, Briley WR, McDonald H. Viscous primary/secondary flow analysis for use with nonorthogonal coordinate systems. AIAA Paper 83-0556; 1983.
7. Levy R, McDonald H, Briley WR, Kreskovsky JP. A Three-dimensional turbulent compressible subsonic duct flow analysis for use with constructed coordinate systems. AIM Paper 80-1398; 1980.
8. Vakili A, Wu JM, Hingst WR, Towne CE. Comparison of experimental and computational compressible flow in an S-Duct. AIM Paper 84-0033; 1984.
9. Towne CE. Computation of viscous flow in curved ducts and comparison with experimental data. AIM Paper 84-0531; 1984.
10. Graf W, Fleming J, Wing N. Improving ducted fan uav aerodynamics in forward flight. 46th AIAA Aerospace Sciences Meeting and Exhibit; 2008.
11. Akturk A, Camci C. Double ducted fan (DDF) as a novel ducted faninlet lip separation control device. International Powered Lift Conference; 2010.
12. Cruz DOA, Pinho FT. Turbulent pipe flow predictions with a low Reynolds number k–ε model for drag reducing fluids. Journal of Non-Newtonian Fluid Mechanics. 2003; 114(2-3):109–148.
13. Hrenya C, Miller S, Mallo T, Sinckair J. Comparison of low Reynolds number k-ε turbulence models in predicting heat transfer rates for pipe flow. International Journal of Heat and Mass Transfer. 1998;41(11):1543–1547.
14. Abbaszadeh M, Shahrooz M, Farhaniyeh B. Effects of using different arrangements of fanlets on performance of twisted axial aerial fans. Proceedings of ASME Turbo Expo, Copenhagen, Denmark; 2012.
15. Rashahmadi S, Abbaszadeh M, Hosseini S, Alizadeh R. Design of a constant chord single rotating propeller. International Conference of Mechanical Engineering Proceeding, Paris, France; 2011.

Peer-review history:
The peer review history for this paper can be accessed here:
http://sciencedomain.org/review-history/9890