International Journal of Enhanced Research in Science Technology & Engineering, ISSN: 2319-7463
Vol. 4 Issue 3, March-2015, pp: (121-127), Impact Factor: 1.252, Available online at: www.erpublications.com

Design Special Types of Blades in Air Box of Car Engines

Timur Choban Khidir
Kirkuk University / College of Engineering - Mechanical Dept.

Abstract: Air box and filter play major role in getting good quality air into car engine. It improves the combustion efficiency and also reduces air pollution. This study focuses on researching the geometry of an air box in car industry to reduce the pressure drop and enhance the filter utilization area by adding blades. 3D viscous Computational Fluid Dynamics (CFD) analysis was carried out for an existing model to understand the flow behavior through the air box, air filter geometry and ducting. Results obtained from CFD analysis of the existing model showed good improvement. Based on existing model CFD results, geometrical changes like blade placement in inlet plenum of the filter, solving and choosing the best of mesh size, removal of contraction in clean pipe of inlet system etc are carried out, to improve the flow characteristics. The CFD analysis of the optimized model was again carried out and the results showed good improvement in flow behavior. By using 3D CFD analysis, the best design of the air box for a car engine is achieved with considerable reduction in development time and cost.

Keywords: Air box, CFD, Filter geometry, Car engine.

1. INTRODUCTION

The main function of an air box system is to supply the engine with clean air with correct amount for the required air to burn in the manifold chamber. The box system of an engine has three main functions. Its first and usually most identifiable function is to provide a method of filtering the air to ensure that the engine receives clean air free of debris. Two other characteristics that are of importance to the engineers designing the box system are its flow and acoustic performance. The flow efficiency of the box system has a direct impact on the power the engine is able to deliver. The acoustic performance is important because government regulations dictate the maximum air mass flow level that vehicles can make during a pass-by test. The speed of air generated by the box system can be a significant contributor to this pass-by filter and separated flow. It may be noted that since the loss pressure from the inlet duct towards atmosphere, this paper assumes the inlet is at the entrance manifold and air filter duct and the outlet is at atmosphere or environmental pressure [1] [2].

To increase the performance of the Proton Wira 2004, it is required for the air to be as clean as possible in order for the engine to be as efficient as possible. Hence the air must in some way be “cleaned” before it enters the combustion chamber. The system that cleans the air and guides it into the cylinders are called an air box system. The air box system may be divided into three main parts: an inlet section in which the incoming “dirty” air is guided into, a filter-box section where a filter is located that cleans the polluted air and hinders soot and other particles from entering the cylinders, and an inlet to the engine where the clean air is guided to the cylinders. Air enters the filter through dirty pipe and inlet side plenum, which guides the flow uniformly through the filter media. Optimum utilization of filter can significantly reduce the cost of filter replacements frequently and keep the filter in use for longer time. To optimize inlet air system and filter duct area, understanding of flows and pressure drop through the system is essential. Computational Fluid Dynamics (CFD) is considered to be the most cost effective solution for flow analysis of inlet system along with filter media. Our research focuses on the studying and choosing of the best way of the proton wira, air box system and, with and without guide blade placement on the filter duct media by CFD analysis results.

The geometry of the inlet’s optimum performance is related to what is generally well known as a loss coefficient, typically identified by K_L, which represents the fraction of the dynamic head lost in the duct. This loss can be easily corrected or compensated for by proper design of the inlet duct [3]. Another phenomena is the s-duct flow, the flow in such as diffusing s-ducts Fig. 1 is complex in nature due to effects arising from the offset between the inlet plane and engine face plane. As the flow moves on through the duct it would perhaps be expected that a similar motion in the opposite sense be initiated at the second bend. However by this stage the low energy flow is largely on the outside wall relative to the second bend and is not driven back circumferentially [4]. One of the most significant drawbacks of such geometry is the appearance of a separated boundary layer located in the curve, which causes decrease of the total pressure of the gas entering the system. Moreover, the strong curve is responsible for the development of a secondary flow composing of counter rotating vortices and responsible for flow distortions. Both aspects significantly degrade the performance of the system. Consequently, it is highly desirable to avoid the boundary layer separation [5].

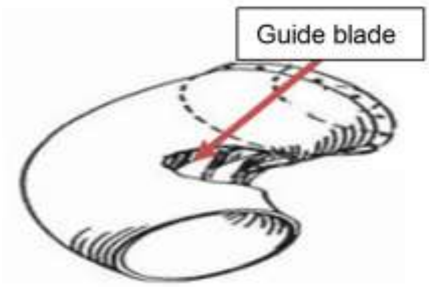

Fig. 1: Diffusing s-ducts with guide blade [4]

A research shows that the design of guide blades for use in expanding bends was investigated both experimentally and numerically. The primary application in mind is the use of expanding corners in wind-tunnels for the purpose of constructing compact circuits with low losses. The experimental results demonstrated that a suitably designed guide blades give very low losses and retained flow quality even for quite substantial expansion ratios [6].

2. MODELING

Fig.2 and. 3 shows the model of air box and filter. In order to save the CFD computational time and cost, trivial geometric details that are not important from fluid flow point of view, such as fillets, blends stiffeners and steps are avoided in the model. All the above-mentioned, so called a cleaned geometry was obtained from solid model. The modeled air box system is assumed to be driven under standard environmental condition neglecting altitude changes. CFD is a computerized method that is widely used in the car industry to study, e.g. the aerodynamics of a car and combustion processes but is also applicable in other industries such as the nuclear power and pharmaceutical industry. It divides the computational domain into small control volumes, known as cells and in these cells all the equations that described the flow field of interest are solved and explanation about flow around the car body. Together with predetermined values at the boundaries or initial conditions, the equations in the cells are solved [7].

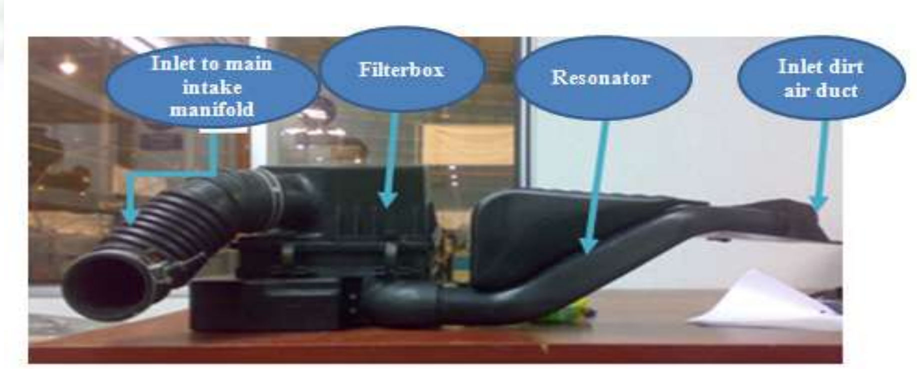

Fig. 2: Existing model of air box system proton wira

Fig. 3: Air box system located under the bonnet of proton wira

Considering the CFD analysis a solidworks model is developed and mimicking the actual parts in the air box system. Fig. 4 shows the exploded view of the air box system in solidworks.

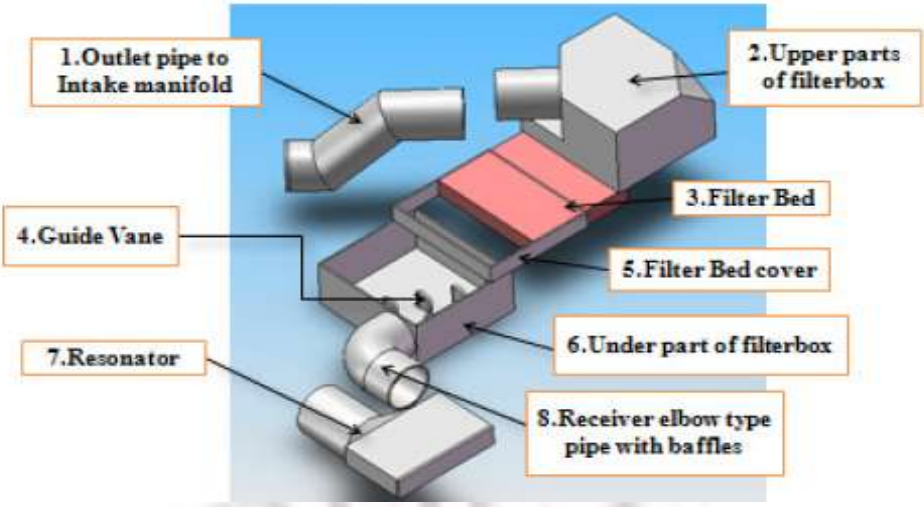

Fig. 4: Solidworks model of air box system

3. METHODOLOGY

Air was used as fluid media, which was assumed to be steady and incompressible. High Reynolds number $k-\varepsilon$ turbulence model was used in the CFD model. CFD generally solves fluid motion by solving the Navier-Stokes equation of mass, momentum and energy equation. The three equations can be written in the conservation form as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0 \qquad (1)$$

$$\frac{\partial \rho u_i}{\partial y} = \frac{\partial}{\partial x_k}(\rho u_i u_k - \tau_{ik}) + \frac{\partial P}{\partial x_i} = S_i \qquad (2)$$

$$\frac{\partial (\rho E)}{\partial y} + \frac{\partial}{\partial x_k}((\rho E + P)u_k + q_k - \tau_{ik}\, u_i) = S_k\, u_k + Q_H \qquad (3)$$

Where u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass, E is the total energy per unit mass, Q_H is a heat source per unit volume, τ_{ik} is the viscous shear stress tensor and q_k is the diffusive heat flux. This turbulence model is widely used in industrial applications. The equations of mass and momentum were solved using simple algorithm to get velocity and pressure in the fluid domain. The assumption of an isotropic turbulence field used in this turbulence model was valid for the current application. The near-wall cell thickness was calculated to satisfy the logarithmic law of the wall boundary. Other fluid properties were taken as constants. Filter bed media of the system were modeled as porous media using coefficients. For porous media, it is assumed that, within the volume containing the distributed resistance, there exists a local balance everywhere between pressure and resistance forces such that the porosity were defined as 85%. Fig. 5 shows the CFD model of the air box system.

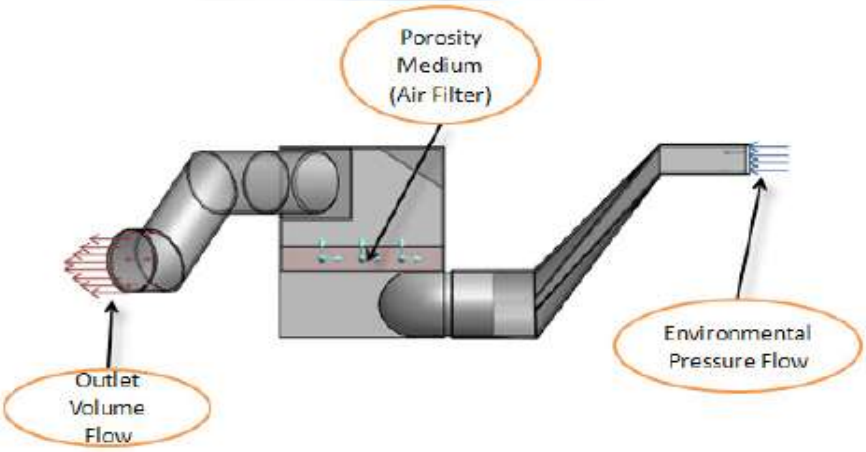

Fig. 5: Air box system in CFD model

It is evident that from the optimal design of engine air entry extensions, when incorporated as an integral part, would not be of conventional design but contains the necessary inlet air blades to approach optimum air flow. The actual bend and internal blade settings would be optimized using computer software to obtain the maximum uniform flow and minimize the value of K_L. The most common method used to determine these head losses or pressure drops is to specify the loss coefficient, K_L which is defined as:

$$K_L = h_{L\,minor} \cdot \frac{2g}{v^2} = \frac{\Delta\rho}{\frac{1}{2}\rho v^2} \qquad (4)$$

A guide-blade section was designed with the aim of minimizing the pressure drop for flow in a 90-deg corner. A central part of the design process was to use potential flow calculations in order to obtain blade geometry such that the velocity distribution on the suction side replicates that of a chosen single airfoil at the angle of attack for maximum lift/drag in a straight free stream. The choice was based on proven good drag characteristics at low Reynolds numbers. For the design of the pressure-side velocity, distribution advantage of the expansion in the middle part of the cascade was taken to obtain a high pressure coefficient on the pressure side, as compared to the single airfoil case [8]. Fig. 5 shows the guide blade design implemented in the air box system as the paper examines the flow in a diffusing shaped proton wira 2004 air entry using computational fluid dynamics (CFD) simulations.

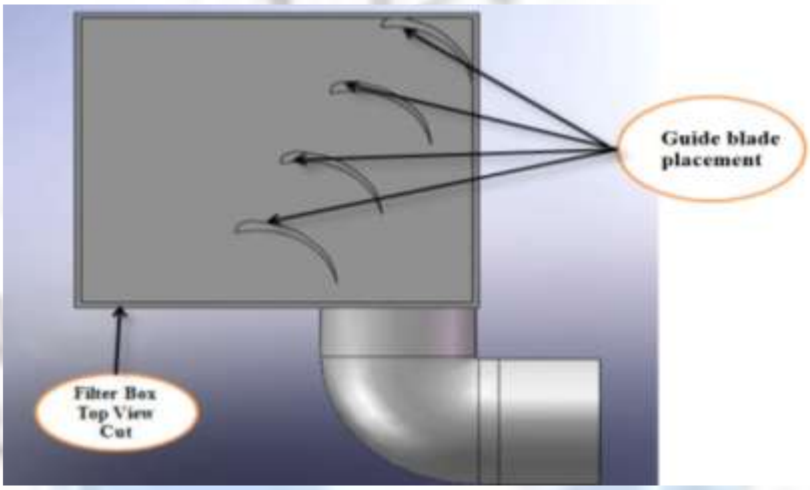

Fig. 5: Placement of guide blade in optimized design

The air flow rate required, V_a can be calculated from equation below for the particular engine speed if the engine displacement is known.

$$V_a = \frac{\eta_V N D_i}{2} \ (m^3/s) \qquad (5)$$

Where,

V_a : is the required engine air flowrate, m^3/ sec

η_V : is the engine volumetric efficiency (assumed 85% for normal engine specification)

N : is engine speed, r.p.m

D_i : is engine displacement, m^3

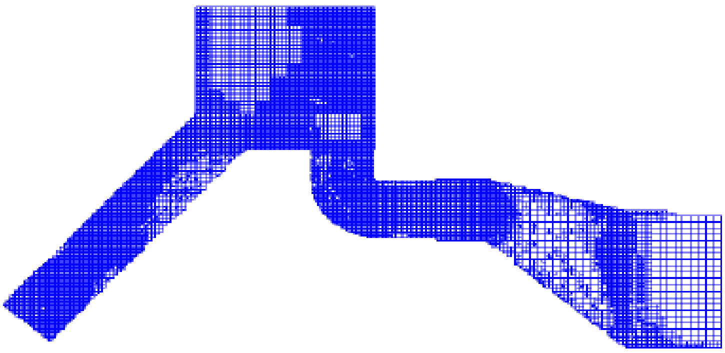

Fig. 6: Mesh analysis for air box system with guide blade

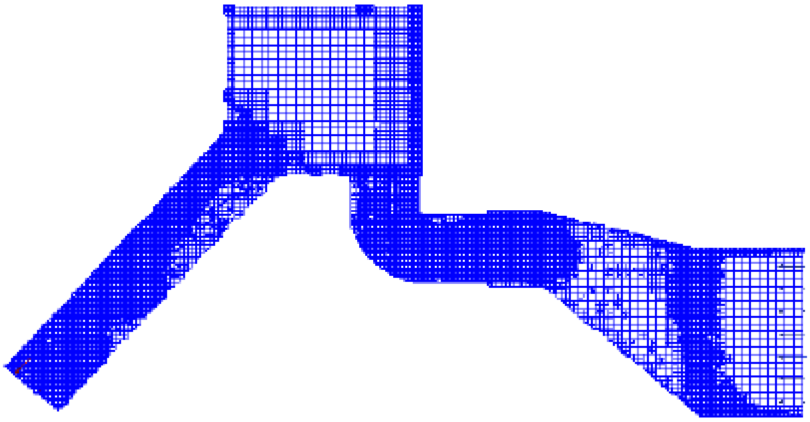

Fig. 7: Mesh analysis for air box system without guide blade

The mesh analysis shows top view of the air box system. The analysis is to do the improvement on the existing model as in Fig. 6. After the improvement was the done to capture the flow inside the system, the mesh of the final CFD analysis of the air box system is as in Fig. 7. The improvement mesh captures more accurately the presence of air at the guide blade wall surface.

4. RESULTS

The results are compared for condition of no guide blade and with the guide blade attached. The first being the pressure drop as shown in Table 4.1.

Table 4.1: Pressure Drop in Air Box System with and Without Guide Blades

	Pressure of air for air box system without guide blades (Pa)		Pressure of air for air box system with guide blades (Pa)	
Speed of air flow	**inlet**	**outlet**	**inlet**	**Outlet**
1000 rpm	**101322**	**101295**	**101322**	**101299**
2000 rpm	**101313**	**101211**	**101313**	**101228**
3000 rpm	**101298**	**101105**	**101297**	**101114**
4000 rpm	**101276**	**100919**	**101276**	**100959**
5000 rpm	**101249**	**100691**	**101249**	**100756**
6000 rpm	**101216**	**100416**	**101216**	**100525**

The analysis was done with 1000 r.p.m increment to the maximum performance of the engine at 6000 r.p.m. A graph was also plotted to see clearly compare the condition tabulated. Fig. 8 shows the pressure drop plot.

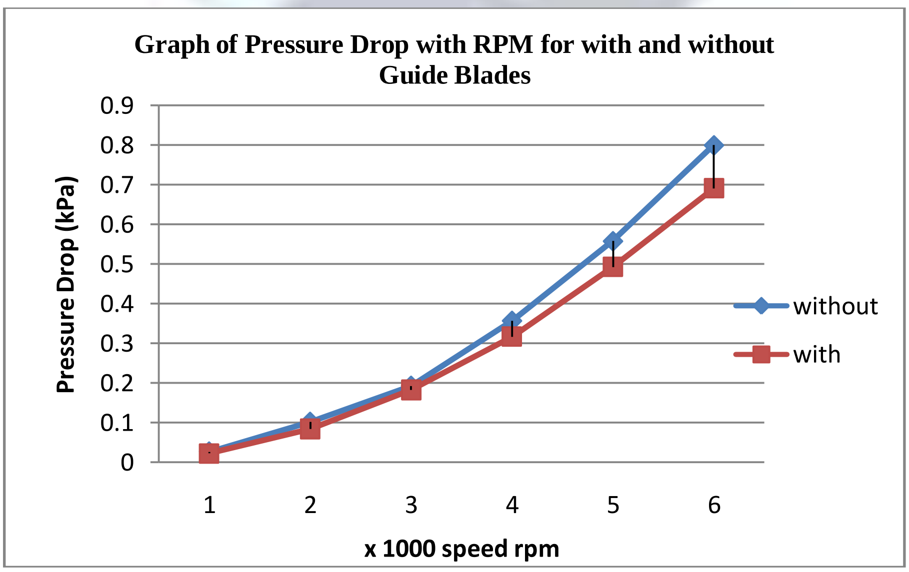

Fig. 8: Pressure plot of simulation

The higher car accelerator is pushed to wide open throttle hence, the more air enter the combustion chamber. From the graph it shows that the relation with using guide blade and without. If guide blade is used at higher rpm more pressure enters the combustion chamber as the pressure drop is smaller. High pressure in combustion area helps the combustion occur very efficiently then without using the guide blade. At low rpm speed noticed that the pressure drop difference between with and without guide blade are very small hence the graph drawn are similar plotted line.

These shows the guide blade help prevent the loss of pressure, from the post processing data the loss obtain by air box system without guide are bigger than without. This is due to the design of the guide blade that reduces the circulation inside the air box duct, when this happen the loss inside the filter box area is reducing. Propose of the baffle placement on the receiver elbow is to reduce the separated flow when the flow of the air enter the filter box duct media. This will help the uniform flow of air stream the S-shaped of the elbow and make the flow from the entry to the first bend receiver this pressure forces acting on the faster moving core. This cause it to move towards the outside of the bend (port side), where there is an adverse pressure region.

Table 4.2: Percentage improvement

RPM speed	Pressure drop between with and without guide blades, P	Percentage improvement, P %
1000	(27- 23)/27 = 0.1481	14.81%
2000	(102- 85)/102 = 0.1667	16.67%
3000	(193- 183)/193 = 0.05181	5.181%
4000	(357- 317)/357 = 0.1120	11.20%
5000	(558- 496)/558 = 0.1111	11.11%
6000	(800- 691)/800 = 0.1363	13.63%

By placement of guide blade in inlet duct filter media the pipe improved the flow and total pressure drop by an average of 12.1% that is significant in inlet system. This shows the pressure drop along the air box system is decreased and the flow is guided by the guide blade to decrease any separation flow and recirculation of the flow affected the system.

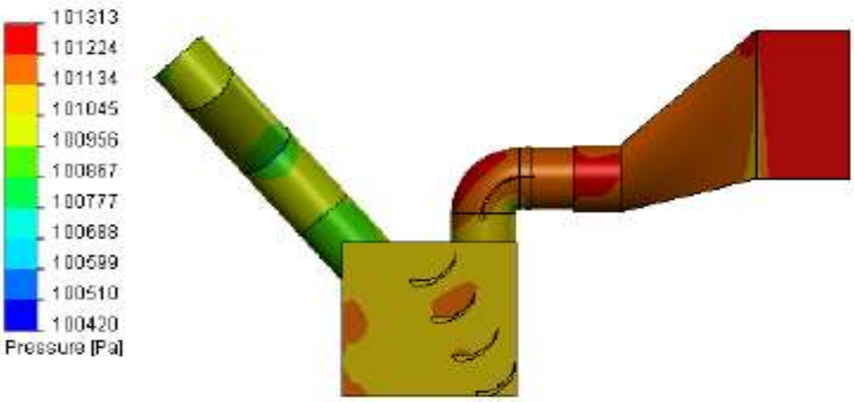

Fig. 9: Bottom view pressure of air box system for with guide blade for 4000 rpm

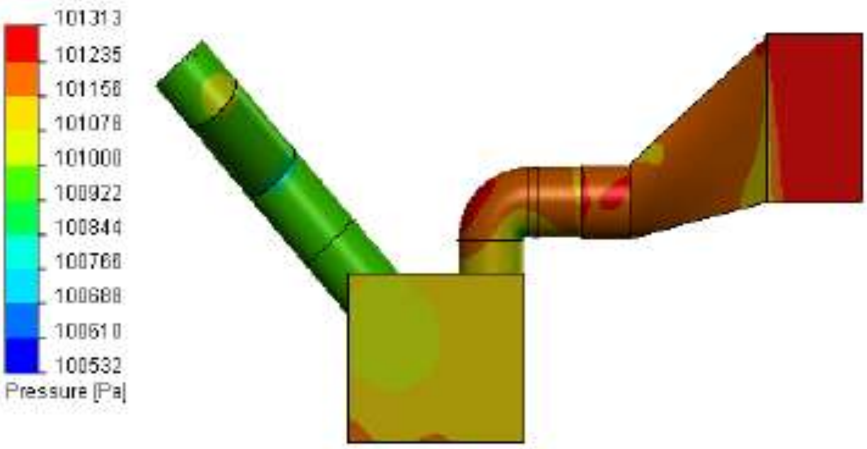

Fig. 10: Bottom view pressure of air box system for without guide blade for 4000rpm

Fig. 9 and Fig. 10 show the pressure drop difference in air box system CFD analysis with and without guide blade. Air box system without guide blade experience pressure loss along the entry elbow on the right back wall of air box system. This loss is resultant of the pressure drop in the model as compared model with guide blade. Analysis in the air box system with guide blade shows the pressure loss experience by the air box system at back wall is decreased. The pressure experience near outlet pipe to inlet pipe is increased. This pressure region near the outlet pipe is an improvement of the design that is hoped will help in the performance of engine. Further tests with actual air flow moving in the air box system will show validation of the simulation done in this research.

CONCLUSION

From this analysis it can be concluded as when there is high pressure number enters the outlet pipe to the inlet manifold this means that the pressure in the manifold is closer to atmospheric pressure. When the pressure drop is decrease air is being quite freely admitted to the engine, which in turn means that more air and fuel is being provided to it, which generates more power.

1. All the above changes incorporated in the design of the guide blade improved overall pressure drop by 12.1% for the rpm speed of 1000 to 6000.
2. Effect of adding more guide blade placement on the critical region may improve the design of air box system even further.
3. Building duct that has more flow features that can guide the air. Actual air flow test is targeted so that a flow validation may be done.

REFERENCES

[1]. A. Hamed 1, Z. Li, S. Manavasi (2009). Flow characteristics through porous bleed in supersonic turbulent boundary layers. Department of Aerospace Engineering and Engineering Mechanics University of Cincinnati.
[2]. Sláčik S.: "Optimalizace zástav by turbovrtulového motoru". Výzkumná zpráva. Walter a.s., Praha, 2003.
[3]. Falcão, C E G, Wildner, F D, Mello, P B, (2012). Experimental measurements of pressure waves in intake system. Automotive Mechanical Engineering Department, Lutheran University of Brazil.
[4]. Taguchi, G., Quality Engineering of Development and Design, Korea Standard Association, Translated to Korean (1991).
[5]. TAYLOR K, SMITH AG & Gomes I, "Further validation and evaluation of a CFD model to predict the pressure drop and flow distribution in filters" S&C report, contract no CBCBDE415, Jan 1999.
[6]. Les Duckers," Wave Power", Engineering Science And Education Journal, June 2000.
[7]. Bovo, Mirko, Computation procedure – CFD Simulation, Volvo cars Corporation, Gothenburg, 2007.
[8]. Bakar, R.A., Semin., Ismail, A.R., Ali, I. 2007.Computational Modeling of Compressed atural Gas as an Alternative Fuel for Diesel Engines, 2nd ANGVA Conference, November 27-29, Bangkok, Thailand.

AUTHOR

B.Sc. University of Technology–Baghdad / Mech. Eng. – 2005
MsD. Gazi University (Ankara) / Turkey – 2010
Specialist: Applied mechanic
Lecturer in College of Engineering / Kirkuk University

TIMUR CHOBAN KHIDIR