International Journal of Engineering Science Invention (IJESI)
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 7 Issue 6 Ver V || June 2018 || PP 96-102

Effect of Adding Guide Vane in Air Intake System of 1.6l Engine

Swagatika Acharya[1], Sai Satyananda Sahoo[2], Namrata Mishra[3]
[1,2]Assistant Professor, Department of Mechanical Engineering, Gandhi Institute For Technology (GIFT), Bhubaneswar
[3] Assistant Professor, Department of Mechanical Engineering, Gandhi Engineering College, Bhubaneswar

Abstract:*Air intake system and filter play major role in getting good quality air into automobile engine. It improves the combustion efficiency and also reduces air pollution. This paper focuses on optimizing the geometry of an intake system in automobile industry to reduce the pressure drop and enhance the filter utilization area by adding guide vane. 3D viscous CFD analysis was carried out for an existing model to understand the flow behavior through the intake system, air filter geometry and filter media. Results obtained from CFD analysis of the existing model showed good improvement. Based on existing model CFD results, geometrical changes like guide vane placement in inlet plenum of the filter, optimization of mesh size, removal of contraction in clean pipe of intake 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, optimal design of the intake system for an automobile engine is achieved with considerable reduction in development time and cost.*
Keywords: *Air intake system, CFD, optimization, automobile engine*

I. Introduction

The work of AIS is to driven air from environment after filter the dirt particles from the intake air and supply cleaner air to the automobile engine. Air enters the filter through dirty pipe and inlet side plenum, which guides the flow uniformly through the filter media. The intake 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 intake system are its flow and acoustic performance [1,2].Optimum utilization of AIS can significantly reduce the cost of filter replacements frequently and keep the filter in use for longer time. To optimize intake system of proton Waja 1.6 and filter duct area, thorough 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 intake system along with filter media. This paper focuses on the optimization of the intake system and filter by CFD analysis.

The geometry of the intake'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]. Figure 1 and 2 shows existing and solid model of intake system and filter. In order to save the CFD

computational time and cost, trivial geometric details that are unimportant from fluid flow point of view, such as fillets, blends stiffeners and steps. All the above-mentioned, so called a cleaned geometry was obtained from solid model. The modeled AIS are assumed to be driven under standard environmental condition neglecting altitude of the car to be analyzed.

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].

A research shows that the design of guide vanes 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 vanes give very low losses and retained flow quality even for quite substantial expansion ratios [6].

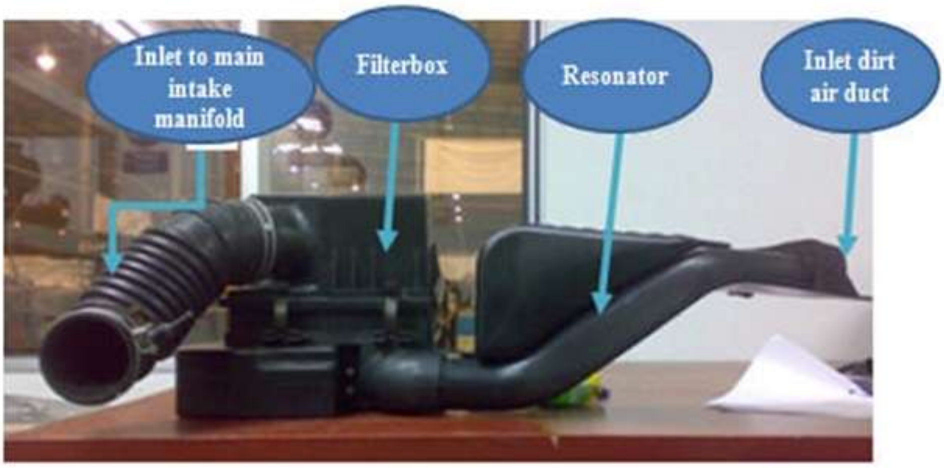

Figure 1: Existing model of AIS proton Waja 1.6

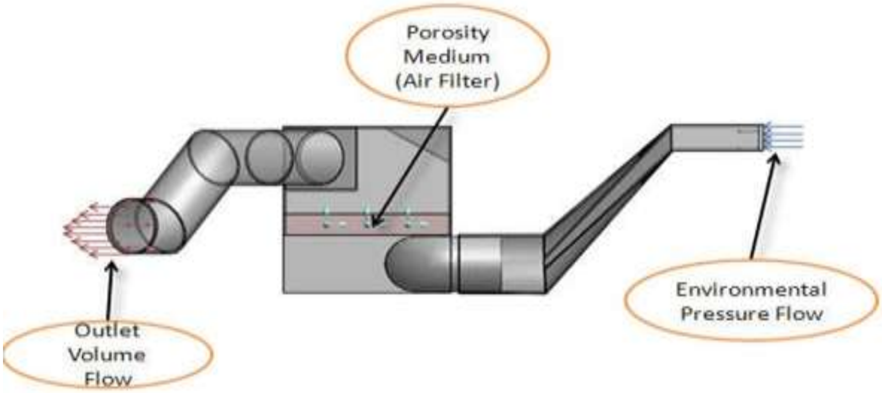

Figure 2: Existing AIS in CAD model

Figure 2 shows the fluid volume for the existing intake system and air filter, and Figure 3 and 4 shows the fluid volume for the modified intake system with baffles. Where filter media is approximated to rectangular volume and considered as porous media. For mesh generation, all surfaces and curves were extracted from the cleaned model. Figure 4 shows the placement of guide vane after the intake elbow where the flow will be guided to the outlet pipe to intake manifold.

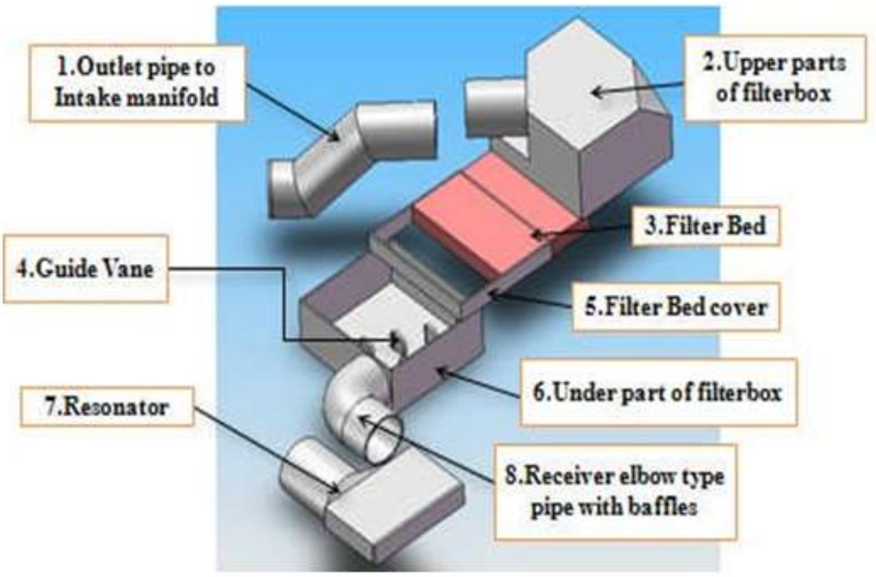

Figure 3 CAD model with guide vane

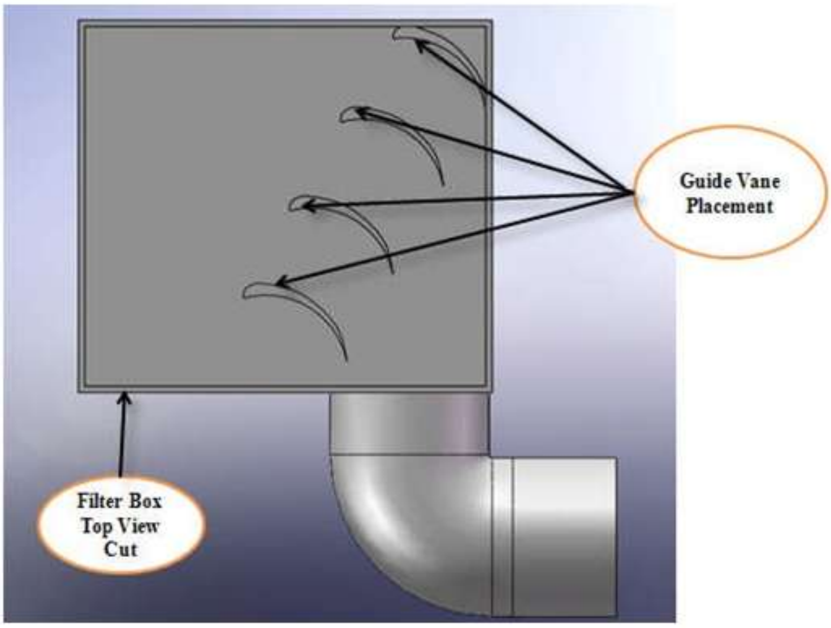

Figure 4: Placement of guide vane in optimized design

II. Methdology

Air was used as fluid media, which was assumed to be steady and incompressible. High Reynolds number k-ε turbulence model was used in the CFD model. The mass, momentum and energy conservation laws in a Cartesian coordinate system rotating at the Ω angular velocity about an axis passing through the coordinate system's origin can be written in the conservation form as follows:

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

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

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

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 intake 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%. Together with predetermined values at the boundaries or initial conditions the equations in the cells are solved [7] [8].

III. Results And Discussion

The Table 1 above shows different in pressure drop between two analyses done for existing model and optimized model. This annalist done with 1000rpm to it maximum performance exceeding it normal rpm speed level 6000rpm to 7000rpm speed.

Table 1: Pressure drop difference

Speed of air flow	Pressure of air for AIS Without Guide Vane (Pa)		Pressure drop across the AIS (Pa)	Pressure of air for AIS With guide vane (Pa)		Pressure drop across the AIS (Pa)
	Intake	**Outlet**		**Intake**	**Outlet**	
1000rpm	101322	101295	27	101322	101299	23
2000rpm	101313	101211	102	101313	101228	85
3000rpm	101298	101105	193	101297	101114	183
4000rpm	101276	100919	357	101276	100959	317
5000rpm	101249	100691	558	101249	100756	496
6000rpm	101216	100416	800	101216	100525	691
7000rpm	101177	100128	1049	101177	100233	944

Table 2 is the percentage improvement calculation table to show percentage pressure drop is improved over the installation of the guide vane. The table shows the pressure inside of the AIS is improved and the loss decrease over each rpm speed varies with 1000rpm. By placement of guide vane in inlet duct filter media the pipe improved the flow and total pressure drop by 12.01% that is significant in intake system. This shows the pressure drop along the AIS is decreased and the flow is guided by the guide vane to decrease any separation flow and recirculation of the flow affected the system. 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 [9].

Figure 5 and 6 show that analysis done with guide vane (red line) shows improvement of the pressure loss and the velocity flow guided to the outlet pipe to intake pipe to the manifold. Figure 6 that the flow is guided uniformly with the rpm speed and without (blue line) guide vane analysis in AIS. This means the flow is guided and guide vane reacts as flow guider and to avoid any separation flow and recirculation develops in the AIS filter duct media.

Table 2: Percentage improvement

RPM speed	Pressure drop between with and without guide vane, 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%

7000	(1049 - 944) / 1049 = 0.1001	10.01%

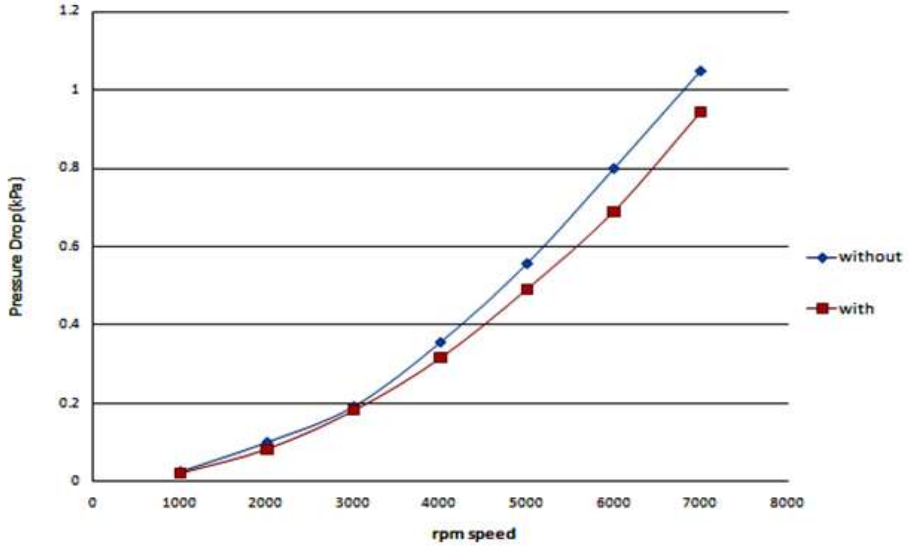

Figure 5 Pressure drop vs. rpm speed

Figure 7 and 8 shows the pressure drop different in AIS CFD analysis with and without guide vane. The figure shows where the pressure loss experience by AIS without guide vane along the intake elbow on the right back wall of AIS this loss resultant of the pressure drop in the model of existing model. Analysis in the AIS with guide vane shows the pressure loss experience by the AIS at back wall is decreased. The pressure experience near outlet pipe to intake pipe is increased. This pressure region near the outlet pipe is an improvement of the design that shows good correlation that will help the performance of engine.

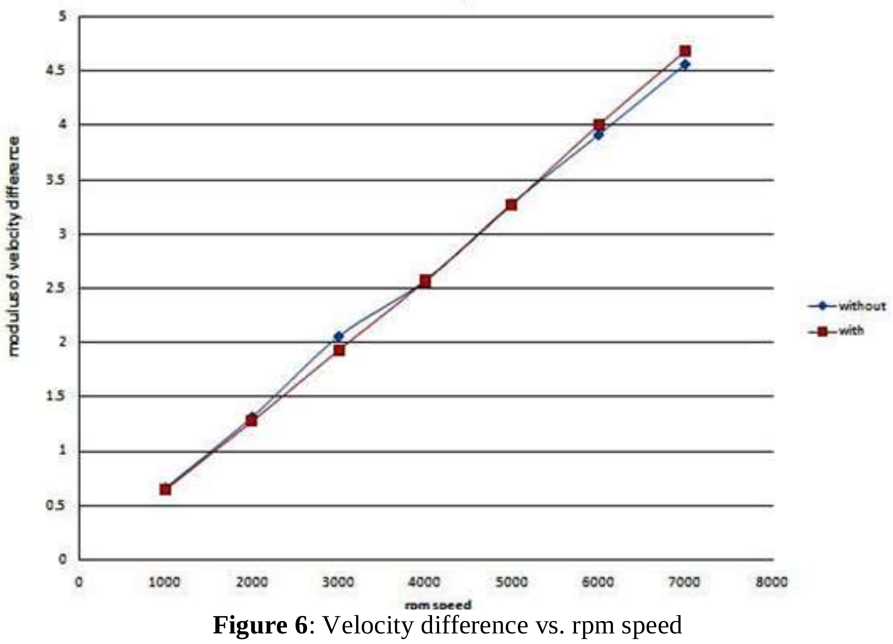

Figure 6: Velocity difference vs. rpm speed

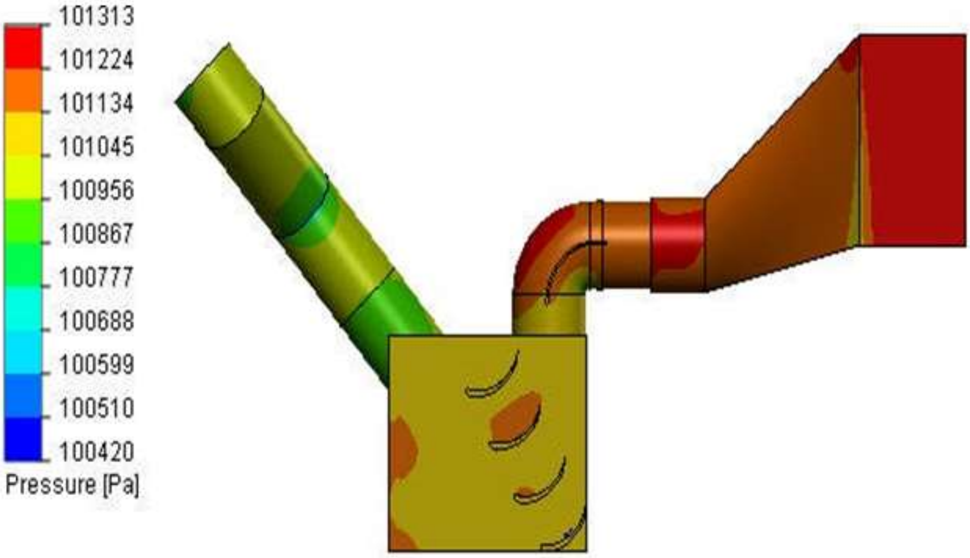

Figure 7: Bottom view pressure of AIS for with guide vane for 4000rpm

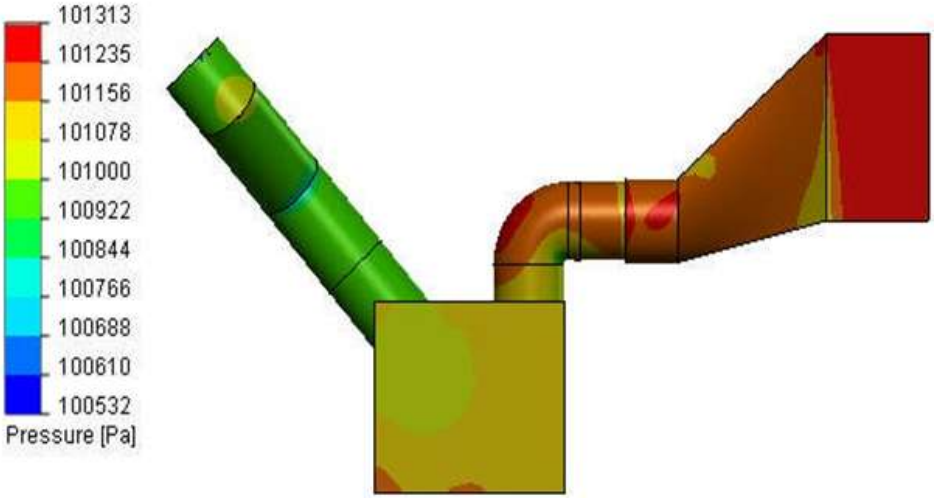

Figure 8: Bottom view pressure of AIS for without guide vane for 4000rpm

The mesh analysis shows top view this mesh analysis is to do the improvement on the existing model as shown in Figure 9. After the improvement has been done for AIS Figure 10 were the done to capture the flow inside the system. The improvement mesh is more accurately capture the presence of air at the guide vane surface and to help decrease the pressure losses along the filter duct area.

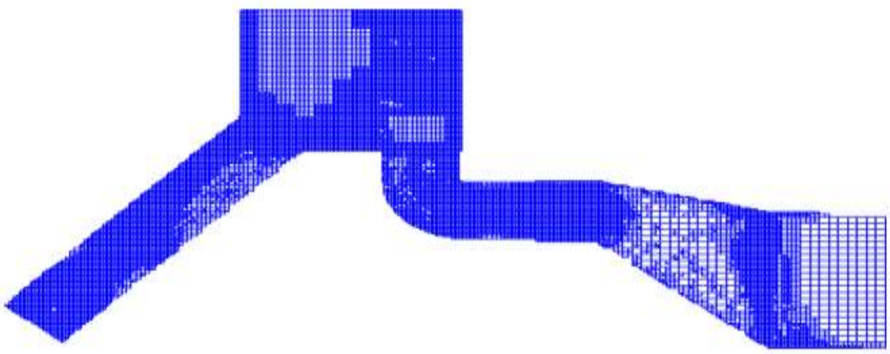

Figure 9: Mesh analysis for AIS with guide vane

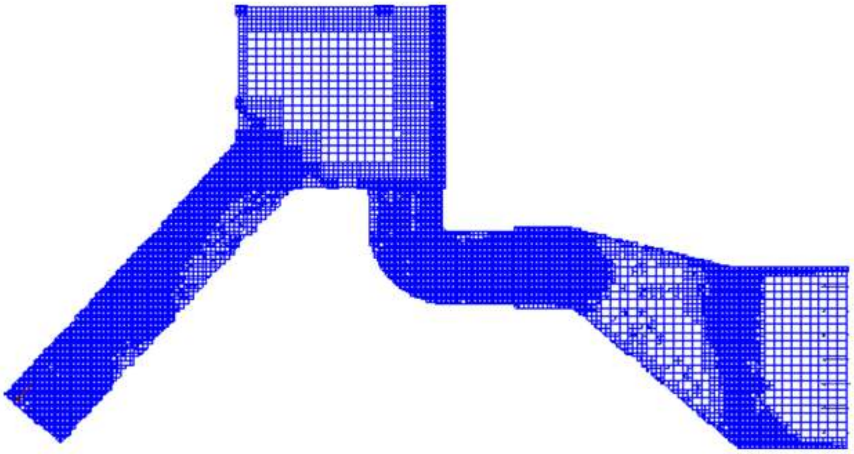

Figure 10: Mesh analysis for AIS without guide vane

IV. Conclusion

From this analysis it can be concluded as when there is high pressure number enters the outlet pipe to the intake 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.

i. All the above changes incorporated in the design improved overall pressure drop by 12.01% for the rpm speed of 1000 to 7000.
ii. Objective of the project achieved when the analysis data shows improvement optimization design using guide vane improved.
iii. Analysis effect of the water and dirt projection in the AIS that will disturb the pressure.
iv. Effect of design more guide vane placement on the critical region and improve the design of AIS.
v. Building duct that has more flow features that can guide the air. Introduction of bell-mouth in dirty and clean pipe.
vi. Analysis effect of minimizing the length of AIS.

V. Acknowledgements

The authors would like to thank Universiti Malaysia Pahang for laboratory facilities and financial support under grant RDU090366.

References

[1]. S. Das and J. K. Prasad. (2008). Characteristic of a Supersonic Air-Intake with Bleed. Department of Space Engg. & Rocketry, B. I. T, Mesra, Ranchi, India.
[2]. P. Chudý, K. Fiakovský, J. Friedl. (2004). Aerodynamic Analysis of Turboprop Engine Air Intake. Czech Technical University Publishing House.
[3]. M.F. Harrison and P.T. Stanev. (2004) Measuring Wave Dynamics in I.C Engine Intake System. School of Engineering, Cranfield University, England.
[4]. Primož Pogorevc, Breda Kegl. (2007). Optimal Design of the Intake System. University of Maribor, Faculty of Mechanical Engineering.
[5]. Kate TAYLOR. Anthony G SMITH, Stuart ROSS, Martin SMITH. (1999). The
[6]. Prediction of Pressure Drop and Flow Distribution in Packed Bed Filters. Second International Conference on CFD in the Minerals and Process Industries, CSIRO, Melbourne, Australia.
[7]. M. Suzuki and C. Arakawa. (2002). Guide Vanes Effect of Wells Turbine for Wave Power Generator. Department of Mechanical Engineering, The University of Tokyo, Tokyo, Japan.
[8]. Pär Nylander. (2008). CFD Modeling of Water Ingestion in Air Intake System. Lulea University of Technology.
[9]. Ravinder Yerram, Nagendra Prasad, Prakash Rao Malathkar, Vasudeo Halbe, Shashidhara Murthy K. (2006). Optimization of Intake System and Filter of an Automobile using CFD analysis. Quality Engineering & Software Technologies (QuEST), Bangalore, India.
[10]. Semin, Abdul Rahim Ismail, Rosli Abu Bakar and Ismail Ali. (2008). Steady- State and Transient Simulation of Gas Flown Temperature of Intake Port Engine. University Malaysia Pahang, Kuantan, Malaysia.