Modenas CT115s performance enhancement by redesign internal geometry to improve the flow pattern of air box

Efi N E[1], Shahriman A B[1*], Rojan M A[1], I Zunaidi[1] , Z M Razlan[1], W K Wan[1], M S M Hashim[1], N S Kamarrudin[1], A Harun[2], I Ibrahim[1] and Azizul Aziz I[3]

[1] School of Mechatronic Engineering, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.
[2]School of Microelectronic Engineering, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia
[3] Modenas Research and Development Division, Kawasan Perusahaan, 08300, Gurun, Kedah.

*Corresponding author: shahriman@unimap.edu.my

Abstract. This paper reviews a study of engine performance MODENAS CT15s by redesign internal geometry to improve flow pattern of air box. Air box is an empty chamber on the inlet of most combustion engines combustion process. It collects air from outside and feeds it to the intake hoses of each cylinder. The entire air box collects the fresh air and later gets it filtered through the air filter present inside it. Engine can be easily breathing related to flow pattern by best design of air box that can contribute to performance. Flow pattern of air box affect the volumetric efficiency, mass flow rate and velocity which ultimately makes impact up on the engine power & torque. 3D Computational Fluid Dynamics (CFD) analysis was carried out for an existing model and several improvement models to make comparison of air flow behaviour through the air box geometry. Based on the results, geometrical changes like sigle blade placement in inlet plenum showed good improvement in flow behaviour compare to existing model. By using 3D CFD analysis, the best design of the air box for a motorcycle engine is achieved with considerable reduction in development time and cost.

Keywords: Air box, internal geometry, air flow pattern, CFD simulation, performance enhancement.

1. Introduction

The iteration of engine performance is the major concern in the automotive industry [1]. The aim for this project is to study engine performance MODENAS CT115s by redesign internal geometry to improve flow pattern of air box. Here, focuses on the CT115s air box model as the benchmark. However, this latest model still needs some improvement so that it can produce superior performance. The improvement that can be done is by redesign internal geometry of air box system in order to improve engine performance. The design of the internal air box geometry including the place of the air filter will effect on mass flow rate, volume flow rate and pressure in inlet and outlet of air box, thus will influences the engine performance[2]. Besides, this project has allowed to study about different air box internal geometry and to understand what gives the best engine performance with all parameter that will be deliberated in relations of graphical design methods before select the most appropriate design for further study.

Air box is an empty air chamber that collects air from the outside and feed it to the intake hoses of each cylinder for the combustion engine [3]. Generally, the old model engine takes air directly from surrounding into each carburettor different with now days engines takes air into an air box that linked by horses to every carburettor, thus avoiding an extra intake [4]. The research possible solution for optimization of air box especially on the internal geometrical design that gives a big impact in combustion process in addition to determine the performance of the engine can be. The analysis of the air box will be done by using computerized engineering software such as CATIA and CFD ANSYS. This analysis is designed so that optimization of the air box can be done, thus the best design of air box can be determined.

2. Methodology

2.1. Air Box Model in CAD Software

The software that has been used to model this air flow is CATIA V5. CATIA V5 is the drawing tools software that enables us to draw the exact dimension of MODENAS CT115s air box model. The air box modelling in this project can divided into two types which are the actual design and the proposed designs as shown in figure below which are Figure 1 until 3 shows overall view of CAD model of proposed design with blade.

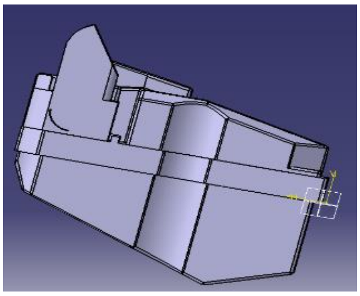

Figure 1: Design 1 with single blade

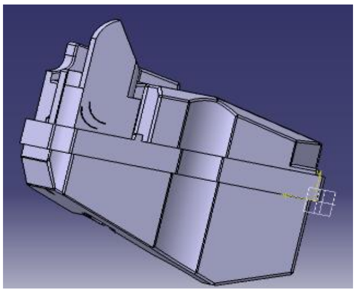

Figure 2: Design 2 with double blade

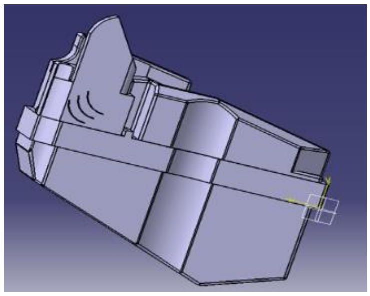

Figure 3: Design 3 with three blade

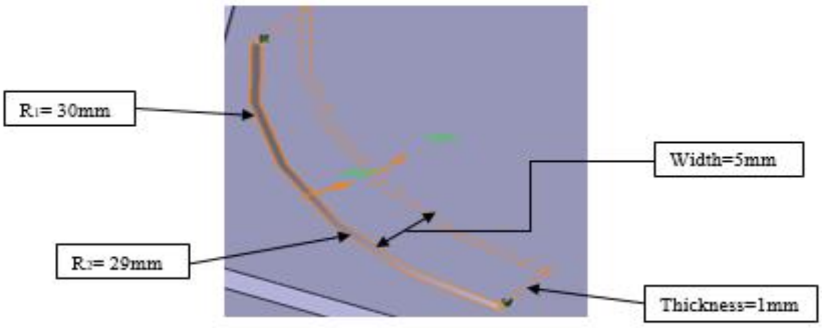

Figure 4: Dimension of blade

For proposed designs, blade are created in the inlet duct of the air box. It is because there have a statement said that adding more blade on the critical region, may improve the design of air box system [3]. In addition, all dimensions for each design of blade are the same including their thickness and width as shown in Figure 4. For thickness, 1 mm has been taken as a boundary because the best design of blade thickness is as thin as possible in order to improve the air flow [7], and its width, 5mm

2.2. Computational Fluid Dynamics Analysis

The analysis of this project is performed using the Computational Fluid Dynamic (CFD) fluid flow fluent to determine the efficiency of the air flow. Besides, CFD is an analysis tools that are used to improve the design of air box system by simulating the air flow in it. The specific name of the CFD tool that has been used in this project is ANSYS software. Based on figure 5 shows the velocity streamline and different result of every design.

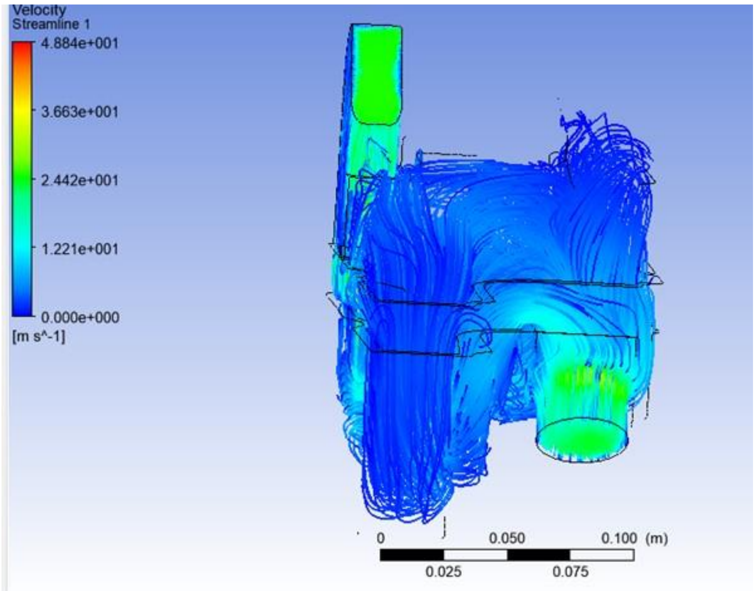

Figure 5: Air Flow of Existing Design of air box

3. Result and Discussion

The result will divided into two categories which are result for actual design MODENAS CT115s and proposed designs which are Design 1, Design 2 and Design 3. Table 1 below shows the comparison parameter of analysis data for all the designs.

Table 1: Comparison parameter of analysis data for existing design and proposed designs

Type of design	*Mass Flow Rate (kg/s)*	*Volumetric Flow Rate (m^3/s)*	*Reynolds Number*	*Velocity Magnitude (m/s)*	*Total pressure (Pascal)*
Existing Design	$3.20\,x10^{-5}$	$5.06\,x10^{-5}$	*Min = 0.00203* *Max = 5200*	*20.1545*	*256.8430*
Design 1	$4.82\,x10^{-5}$	$6.38\,x10^{-5}$	*Min = 0.00312* *Max = 5360*	*20.1682*	*257.2849*
Design 2	$3.66\,x10^{-5}$	$5.44\,x10^{-5}$	*Min = 0.00217* *Max= 5090*	*20.1609*	*256.4939*
Design 3	$3.02\,x10^{-5}$	$4.92\,x10^{-5}$	*Min = 0.00160* *Max= 4740*	*20.1626*	*257.4713*

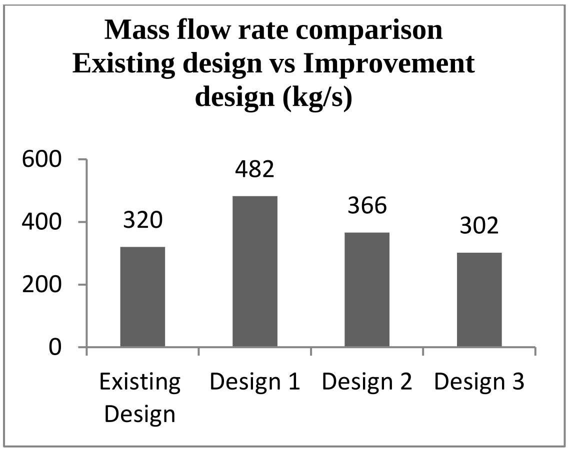

Figure 6: Mass flow rate comparison

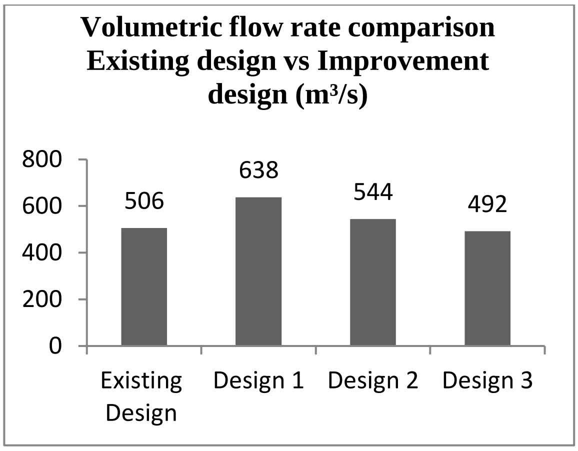

Figure 7: Volumetric flow rate comparison

Air box play main part in getting good quality air into vehicle engine. It advances the burning competence and also decreases air pollution. 3D Computational Fluid Dynamics (CFD) analysis carried out for an existing design and several improvement design to understand the flow actions through the air box system and geometry. Results got from CFD analysis of the existing design showed real situation for the air flow. Based on existing design CFD results, geometrical changes like sigle blade placement in inlet plenum showed good improvement in flow behavior compare to existing model. By expending 3D CFD analysis, ideal design of the air box system for vehicle is completed with substantial reduction in progress time and cost [3]. The ideal result to reach determined mass flow rate for air in the air box system it is need to reduce the pressure difference over the flow constraint in the air box [11]. When the air flow increases, the pressure drop in the air box might also increases [12]. The finest overall plan for this improvement design is to redesign the internal geometry of the air box so that can launch the air flow in the air box, thus can optimize the engine performance. From the data collected through the many simulations and analysis, it can be detected that a single design of blade has capability to achieve the highest value of mass flow rate and volumetric flow rate compare to existing design and other improvement design.

4. Conclusion

Conclusion that can be made from the analysis from Figure 6 and Figure 7, it shows that the improvement design of air box model which is also known as the Improvement 1 produced the best result in terms of mass flow rate and volumetric flow rate compared with the actual design MODENAS CT115s and other design. The CFD analysis studies that are performed on the air flow coefficient have provide important information that the reason why the air flow coefficient become difference are because it affected by the geometry design. From this study, it show that some new changes that can make for improvement on the air box so it can be function better for the system. This study also helps to develop the standard CFD work procedure for future air box geometric design optimization.

Acknowledgements

I would like to express my deepest appreciation to all those who provided me the possibility to complete this paper. A special gratitude I give to my supervisor and MODENAS Research and Development team whose contribution in stimulating suggestions and encouragement, helped me to coordinate my project students especially in writing this paper

References

[1] M. Rosli, "The Automobile Industry and Performance of Malaysian Auto Production," Journal of Economic Cooperation, vol. 27, no. 1, pp. 89-114, 2006.
[2] Makgata, K. W., " Computational Analysis and Optimisation of The Inlet System of A High-Performance Rally Engine," Reserch Master thesis, Faculty of Eng. Built Env. and Information

Tech., Pretoria Univ, South Africa, Pretoria, 2005

[3] T. C. Khindir, " Design Special Types of Blades in Air Box of Car Engines," International Journal of Enhanced Research in Sc. Tech. & Eng., vol. 4, pp. 121-127, March 2015.

[4] A.S Phulpagar and N.S Gohel, "CFD Analysis of Air Intake Systems," International Journal on Theoretical and Applied Research in Mechanical Eng. (IJTARME), vol. 4, pp. 2319-3182, 2015.

[5] A. J. Balinga and Dr. L. L. R. Rodrigues, " Business Plan – The Secret to Success," The International Journal of Business & Manangement, vol.3, issue 4, pp. 210-214, Apr. 2015.

[6] Cyr, L., "Creating a Business Plan", Boston, Massachusetts: Harvard Business School Press, 2007.

[7] Y. Zhang and Y. Jia, "2D Automatic Body-fitted Structured Mesh Generation using Advancing Extraction Method," Journal of Computational Physics, vol. 353, pp. 316-335, 2018.

[8] Aman Mohd. Ihsan Mamata, N. N., Salmiah Kasolanga, Aminuddin Zulkiflia and Rizalman Mamat. "Effects of Air Intake Pressure to the Fuel Economy and Exhaust Emissions on a Small SI Engine," The Malaysian International Tribology Conference (MITC), Kuala Lumpur, Malaysia, 2013.

[9] A. Z. Tayfik, N. Rashid, Mazlan, M. Faruq, and M. Zahir, "Electric car chassis design and analysis by using CATIA V5 R19," Journal of Mechanical and Civil Eng., vol. II, issue 4, Jul.-Aug. 2014.

[10] S. Pal, H. Reddy Tiyyagura, I. Drstvensek and C. S. Kumar, " The effect of post-processing and machining process parameters on properties of Stainless Steel PHI product produced by Direct Metal Laser Sintering," International Conference on Manufacturing Engineering and Materials (ICMEM), Novy Smokovec, Slovakia, June 2016.

[11] Sundar Raj, C., & Sendilvelan, S. Effect of oxygenated hydrocarbon additives on exhaust emission of a diesel engine. International Journal of Automotive and Mechanical Engineering, 2, pp 144-156. 2010.

[12] W. "Optimization of Air Intake System for Inline-4 Diesel Engine." Bachelor of Mechanical Engineering with Automotive Engineering, June 2012.