Numerical Analysis of Diesel Engine Performance Fitted with Turbonator in the Intake Manifold

K.Selvarasu[1*] and R.Mohan[2]
[1]*Research Scholar, Department of Mechanical Engineering,*
Sona College of Technology, Salem, Tamilnadu
[2]*Associate Professor, Department of Mechanical Engineering,*
Sona College of Technology, Salem, Tamilnadu
E-mail-kselvarasu111@gmail.com[1], kselvarasu.aol@gmail.com[1]
rmohan12@gmail.com[2]
**Corresponding Author*

Abstract

Now a day's ecological imbalance due to air pollutions is growing day by day, due to the rapid growth in the usage of automobiles. The reduction of exhaust gas emission from automobiles is necessarily by improving fuel combustion. While the time required for the combustion is short, there is no fraction of fuel in combustion and returns along with exhaust. To avoid this, an adequate mass of air in air-fuel mixture (lean mixture) in the burning of diesel engine results economical fuel intake and better volumetric efficiency. The volumetric efficiency of the diesel engine is improved by a supercharger namely a turbo-charger. However, the turbocharger requires more maintenances, size and space for engine to be installed in the automobiles. Thus a Turbonator is introduced in the diesel engine air intake manifold which generates swirl in the flow of cylinder-air.In this paper, the design of Turbonator involves evaluation of blade angle (θ), root angle (α), number of blades (n) and Flow Parameters (velocity V_i&V_O) usingCREO software. In turbonator, the number of blades could be varying such as 4, 8 and 12. The CFD analysis is conducted for intake air into cylinder through Turbonatorusing "ANSYS – FLUENT". By evaluation the results of contour pressure and velocity, the Turbonator with 12 blades shows better performance than that of 4 and 8 blades which in-turn improves life of engine.

Keywords: *Diesel engine, engine performance, Turbonator, Blade parameters, CFD analysis.*

1. Introduction

The transportation demand for people is growing more, such that number of vehicle was running on the roads. Consequently, this led to a high rate of emissions from vehicles. Mostly, vehicle engines are coupled with Turbocharger to increase the air flow during suction strokes. When exhaust gases drive the turbine, which is coupled to a compressor through the turbine shaft. During combustion and power strokes, the both turbine and compressor speed gets reduced results in flow variations. The compressor is more sensitive for variations in mfr(mass flow rate) and pressure ratios than the turbine, results in compressor surge causes turbocharger stability problems. It is well known that, operating the CI engine in presence of compressor surge may stop the flow to cylinders that lead to loss of engine power and also cause engine knocks. Moreover, compressor surge can cause vibrations and damages to the engine parts which increases the maintenance cost. To avoid this problem, the modification options in air-intake manifold studied with Turbonator.

*K.Selvarasu[1]

Turbonator is a device to charge the engine by inducing a swirl into the engine which increases the engine's breath. Due to pressure drop inside the cylinder, air with maximum velocity rushes into engine, hence turbulence effect would reduce the velocity in the air-intake manifold. In Turbonator a series of static blades places diametrically similar to aero-fan turbines, blade are designed in such a way to produce a vortex flow of air in Air-Intake Manifolds in such a way that the swirly air is formed through suction and compression stroke. By this way, the combustion of intake fuel should be completely burnt and improve efficiency.

2. Literature Review

Nik Rosli Abdullah et.al, [1] has discussed about the fuel economy and exhaust emissions at variations of air intake pressure. The air intake pressure is influenced by the degree of opening throttle plate and venturi effect which draw the fuel to the combustion chamber in carburettedengine. The experiment test carried by the variations of engine speed and load using a single cylinder four stroke SI engine attached with 5kW dynamometer.

Mohd Faisal Hushim et.al, [2] has studied that the air flow behaviour inside a different intake manifold angles in IC engine.Six angles of intake manifold have been investigated using CFX simulation tool which are 30°, 60°, 90°, 120°, 150°, and 180°. From the study, results indicated 180° as the best option for intake manifoldangle due to better air flow behaviour inside the intake manifold. This show an agreement to the previous submitted resultsthat was done by using GT-POWER.

The present tests ignored the blade entry and exit phases at the catch and finish of the stroke, respectively, and this could have a limiting effect on blade design. However, the potentially large improvements in blade performance suggested here would likely outweigh any negative influences of blade design on blade entry at the catch or finish of the stroke. In order to examine the influence of oar blade design on rowing performance, the fluid force coefficients from either the experimental trials, or any future CFD models, would need to be used as inputs to a mathematical model of rowing to predict the practical significance of changing blade shape on rowing performance by Nicholas Caplan and Trevor N Gardner [3].

V.RagaDeepuand R.P.KumarRopichrla [4]was observed that in the preliminary design, the rotor blades after being designed were analysed only for the mechanical stresses but no evaluation of thermal stress was carried out. In this paper the first stage rotor blade of the gas turbine is created in CATIA V5 R17 software. The material of the blade is Ni-Cr alloys. This model has been analysed using ANSYS11.0. The gas forces namely tangential, axial were determined by constructing velocity triangles at inlet and exist of rotor blades.

M Sai Vastav**[5]**has designed the 3D model of the turbocharger turbine wheel by using pro-e software and the analysis taken by different materials and the analysis taken by the ANSYS software. This project we are analysing the pressure acting on the turbocharger impeller turbine wheel by the three materials namely Inconel alloy 740, Inconel alloy 783 and wrought aluminium 2219. Then the thermal analysis is done to determine the total heat flux in the 11 and 12 plates for the given temperature conditions.ShubhamPatilet.al [6] has studied the failure of the centrifugal compressor is because of surge and stall. Surge occurs because pressure at the receiver is greater than the pressure at the compressor. So, gas flow will reverse and surge occurs. This work-study is based on reducing surge and stall. We can prevent the tendency of surge and stall by changing the factors affecting the change in pressure.Solimanet. et.al [7]presents an effort to model the flow from inlet to the exit of a turbocharger compressor stage consisting of all the components in place and performance prediction by providing mathematical computation model and numerical analysis using CFD tools and these were verified by experimental work. Using of the computational fluid dynamics (CFD) gave a better understanding of the behaviour of flow through turbocharger compressor stage and how it impacts the turbocharger efficiency and how the turbocharger compressor perform combined with the diesel engines.

L.UmamaheswaraRao and Mohammed Ashif [8] has carried the experimental analysis which gives the output parameters such as static pressure, static efficiency, total efficiency, velocities at entry and exit of the impeller. In the simulation the CFD analysis carried on exact model of final product and which gives result same as in experimental analysis. In comparison there is 9-10% of deviation in the static pressure, 23-24% of deviation in the shaft power and 12-13% of deviation in the efficiency. From comparison 10-15% increase in outlet area thickness of volute casing gives the improvement in results.

Chirag V. Kapuria and Dr. Pravin P. Rathod[9]wereconcluded that, design of inlet manifold configuration has very much importance in IC-engine. Uniform combustion air distribution to each inlet port of cylinder head is the main function of inlet manifold. Flow is evenly distributed to the piston inlet valves by an ideal inlet manifold. Optimum efficiency and performance of the engine is obtained when this even distribution occurs. Volumetric efficiency is also influenced strongly by inlet manifold. If the distribution of air is uneven, then it leads to reduction in volumetric efficiency, power loss and fuel consumption also increases.

3. Importance of Turbanator in the Intake Manifold

Diesel engine development began in decades; it does not suit the fuel economy. The unsuccessful nuclearisation of the combustion between fuel and air produces a decline in combustion, which affects engine performance in terms of fuel economy, due to the excessive use of air in the engine. Some of the techniques was used to enhance the Engine efficiency are adjusting the intakes manifold, improving the vane swirl with tumble guidance devices and modifying the piston profile [9]. This paper deals with replacement of tumble guidance devices as spline blade which named as Turbonator which improves the fuel economy and efficiency.

3.1 Design of Turbonator Blades

Turbonator design begins with selection of blade twist angle, blade height and number of blades using Blade Element Theory, which emphasizes the procedure relevant to aerodynamic parametric and selection of blade parameters in the complete design process which is followed by the structural concern. Assumptions involved in the Turbonator blade design is composed of aerodynamically spline strips or elements.

3.2 Blade Element Theory

The Blade Element Theory highlights a simple method of predicted performance of Turbonator blades, on this theory the blades are separated into a number of impartial sections alongside the length. A major complexity in applying this theory arises when trying to determine the magnitude of the two flow components V_0 and V_2. V_0 is roughly equal to the device forward velocity ($V\infty$). V_2 is roughly equal to the blade section's angular speed (Ω_r) but is reduced slightly due to the swirling nature of the flow. When comparing with real blades layout effects, this principle is expecting to increase in theoretical performance of 5% to 10%. This concept is very beneficial for optimising blade setting for a given flow parameter. As the Turbonator blade is set at an assumed geometric pitch angle (θ) the local velocity vector can make a flow angle of attack on the section shown in Fig.1.which represents the flow vector section of the blade.

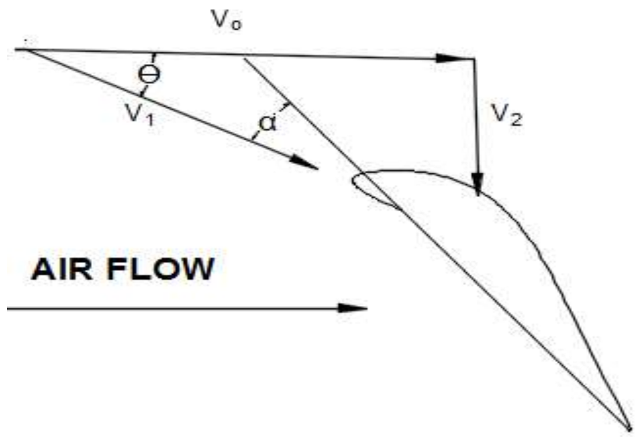

Figure1. Resultant Flow Vectors

V_0 -- Axial Air flow at Turbonator inlet.
V_2 -- Angular flow velocity vector at Turbonator outlet.
V_1 – Flow velocity vector at Turbonator outlet.

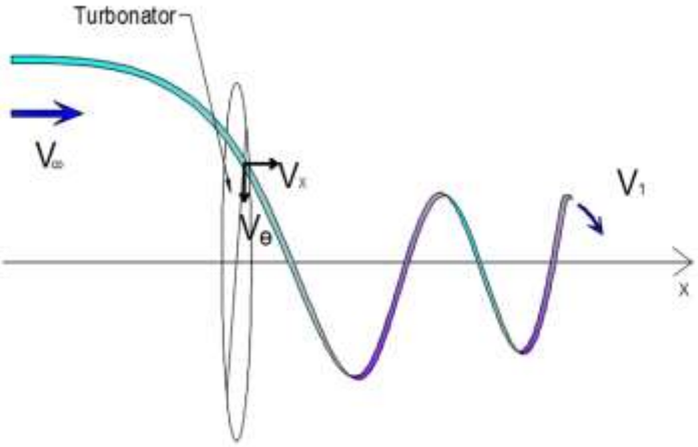

Figure 2. Flow through Stationary Blade

A standard flow tube through section AA would have speeds,
$V_\theta = V\acute{\alpha}\,(1+a)$

So the velocities V_0 and V_2 as shown in the Fig.1
$V_0 = V_{\acute{\alpha}} + V_{\acute{\alpha}} a$
Where a is the axial inflow factor,
$V_{2r} = V_2 - b.V_2$
Where b is the (swirl factor)

The local flow velocity and the angle of attack for the blade section is thus
$V_1 = \sqrt{(Vo^2 + V2r^2)}$

$$\alpha = \theta - \tan^{-1}\left(\frac{Vo}{V2r}\right)$$

Pitch of Blades (p) = 2η r tan (θ)

The local flow velocity and the angle of attack for the blade section is shown in figure 2.

4. Modelling of Turbonator

Turbonator with 4, 8 and 12 blades were modelled using the 3D/CAD CREO design software with the following dimensions given in the Table 1.

Table 1. Turbonator Specification

Design Parameters	Dimension
Intake radius (r) in mm	64.5mm
Length of the blade in mm	40mm
Height of the blade in mm	32.25mm
Blade root angle (α)	3°
Blade Angle (θ)	3
Root angle (θ_1)	35
Tip angle (θ_2)	4
Number of Blades (n)	8
Root Span (r_1) in mm	18
Root Span (r_2) in mm	15
Root width (w_1) in mm	74
Tip Width (w_2) in mm	60
Number of blades	4, 8, 12

The CREO models of Turbonatorare then imported as IGES format into the CFD programming, redesigned into various segments, and refined to create a limited volume meshing. This is a significant advance, where points of interest of the geometrical shape should be characterized correctly. The flow space is additionally made, and the last meshing of all segments should be precise. The errors in the models and flow zone should be adjusted before proceeding. The Turbonator with 4, 8, and 12 blades models are shown in the figures 3a, b, and c respectively.

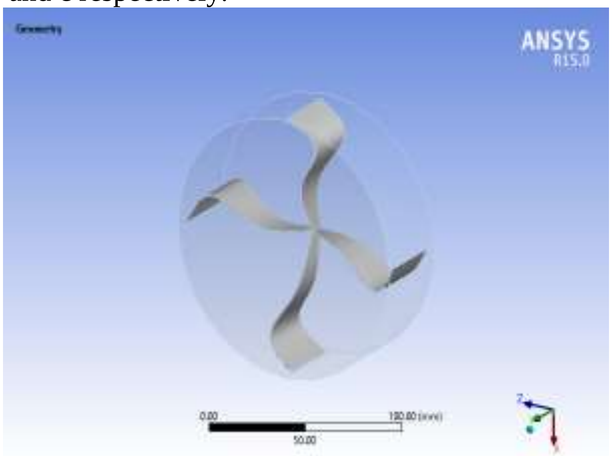

Figure 3 (a).Turbonator with 4-Blades

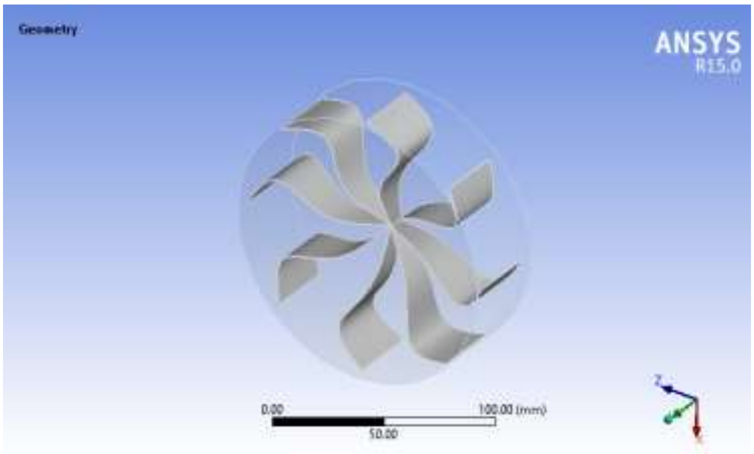

Figure3 (b).Turbonator with 8-Blades

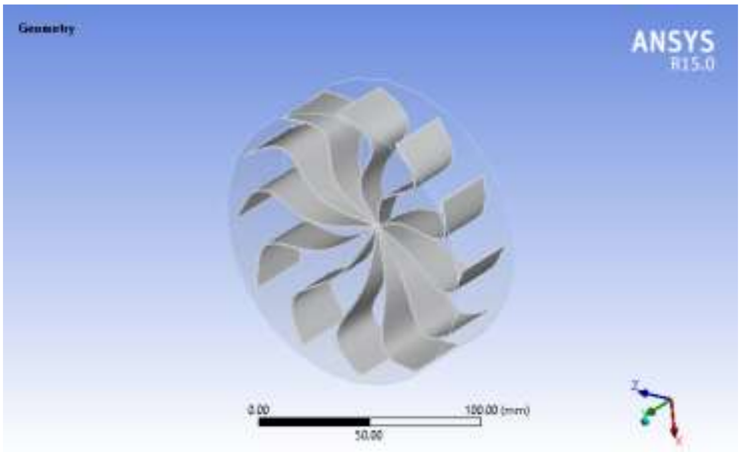

Figure 3 (c).Turbonator with 12-Blades

4.1 Meshing

Meshing is done after import to ANSYS software. For volume meshing, a tetrahedral mesh by and large furnishes a more programmed arrangement with the capacity to add mesh controls to enhance the precision in basic regions. Then again, a hexahedral mesh for the furthermost part gives a more exact arrangement yet is harder to produce. Figure.4. shows the discretization of number elements and nodes of Turbonator. The Way-autonomous strategy was utilized for meshing which utilizes top down approach (makes volume work and concentrates surface mesh from limits).

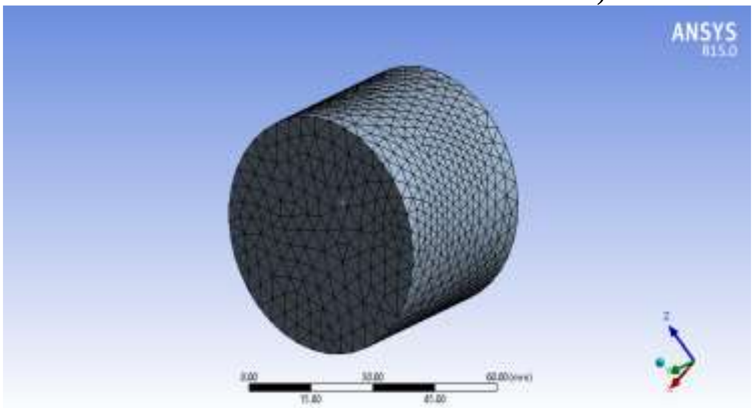

Figure 4. Discretization of Turbonator

4.2 Post Processing

Computer aided design Preparation as indicated by numerical outline information: The CAD demonstrating is isolated into three sections by means of

- Modelling of airfoil booster blade,
- Demonstrating of blade wall
- Demonstrating of housing.

Particulars of the booster-Grid Generation: The point by point CAD show is set up in CAD software's and is meshing utilizing two distinctive programming for surface and for volume meshing respectively.

The second step is to import the IGES format file into the CFD code preprocessor, which will understand the flow conditions. These incorporate inlet air mass flow, outlet pressure, liquid properties, and flow area representation, for example, moving inner zone and stationary strong walls. The subsequent stage is to set the recreation procedure as a 3-D consistent and turbulent issue.

The simulation is followed by the CFD code preparing the details, applying the fundamental hypothesis of liquid mechanics by modifying the conditions of mass coherence and force in numerical form and making numerical forecasts of the flow factors from that point on. The issue setup process is finished by characterizing the boundary conditions, solver controls, and visible screens. Accepting the flow to be perfect and dry air at standard environmental pressure, the boundary conditions include solid wall, moving inner zone, the CFD code stationary zero pressure at outlet, and variable mass flow rate at inlet. The leftover estimations of all factors tackled are observed during the cycle procedure. This emphasis procedure should be observed for conference and repeated if the numerical mistake conditions are not fulfilled. The last advance is to separate the yield information and present them as speed streamline.

5. Results and Discussion

The simulation results for the case of with (4, 8 and 12 blades) and without Turbonator are discussed in the following section. The inlet velocity of 0.15m/s flows into cylinder is considered for all geometry conditions of with and without Turbanator.

(i) Without Turbonator

The velocity contour streamline for without Turbonator is shown in the Fig.5. The red colour represents the maximum range of attained velocity whereas blue colour represents minimum.

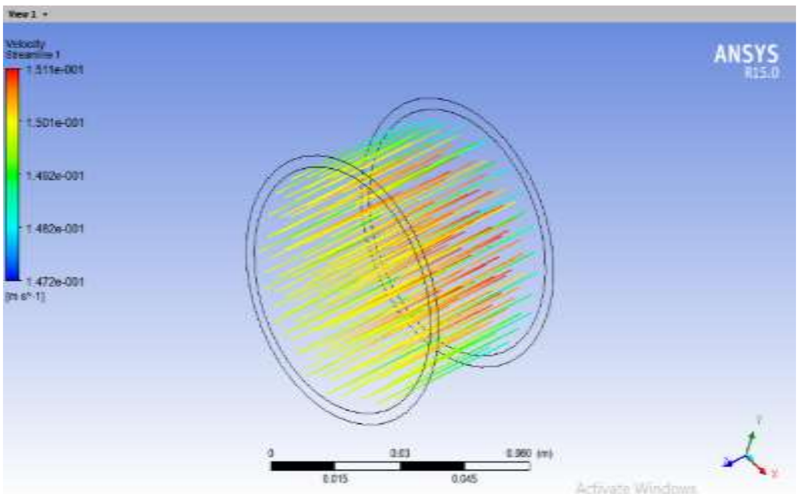

Figure 5. Velocity contour - Without Turbonator

(ii)With Turbonator

Simulations results for the case of Turbonator with 4, 8, and 12 blades are studied and the pressure and velocity contours shown in the Fig 6-11. The result of pressure and velocity representing the maximum range through red in colour whereas minimum in blue.

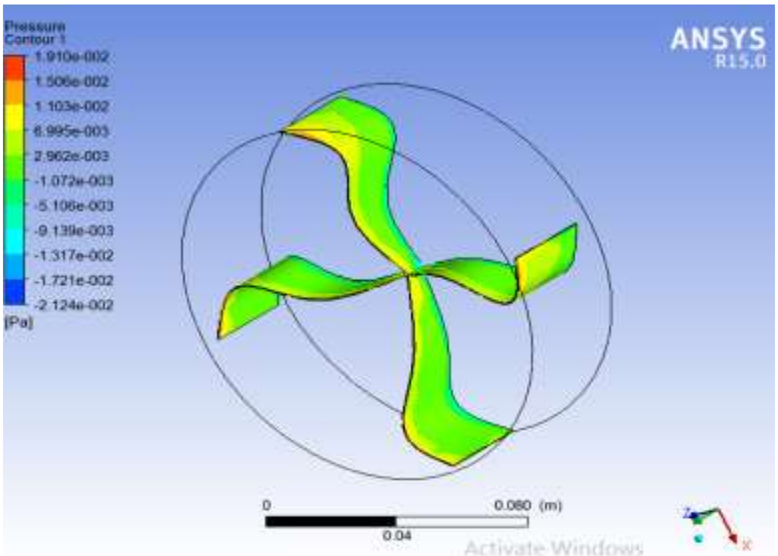

Figure 6.Pressure contour- Turbonator with 4 blades

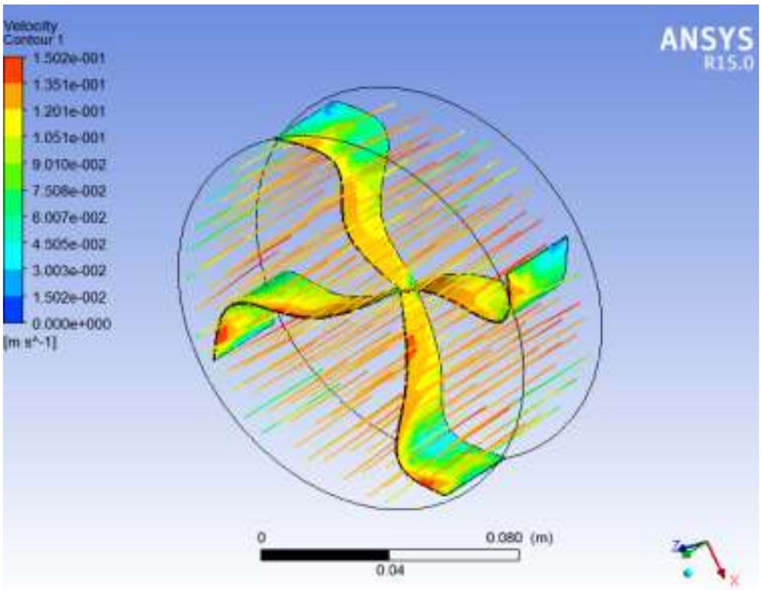

Figure7. Velocity contour- Turbonator with 4 blades

Fig. 6 illustrates that the pressure formation inside the wall surface of the blade and Turbonator. It was observed that that pressure ranges from negative to positive scale; consequently, making a pressure zone at the outlet.

Figure.7 shows that the simulations result of outlet velocity for the Turbonator with 4 blades. The outlet velocity has increased while compared with inlet velocity to the Turbonator.

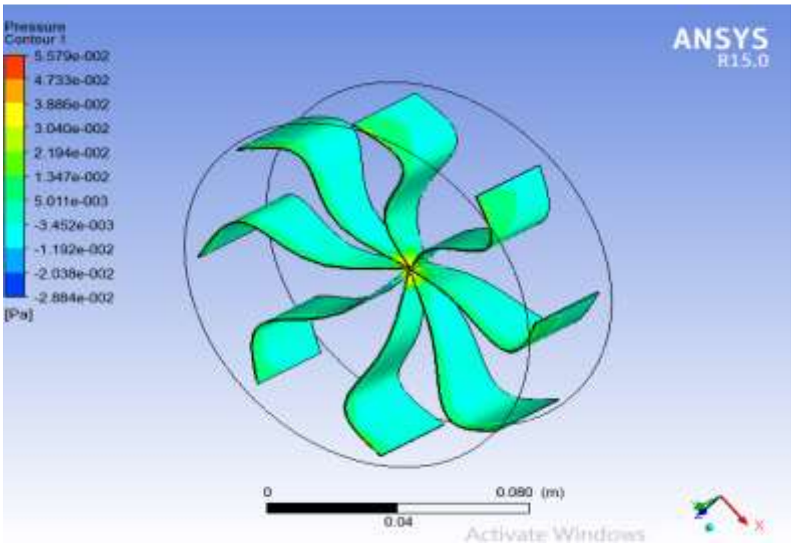

Figure 8. Pressure contour- Turbonator with 8 blades

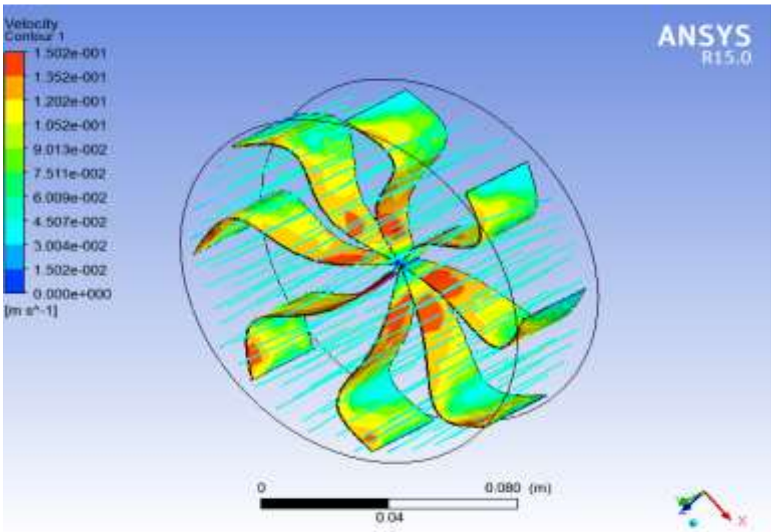

Figure 9. Velocity contour- Turbonator with 8 blades

The pressure and velocity contour results of Turbonator with 8 blades condition shown in the Fig 8 and 9 respectively. The results shows that flow velocity is same as that of the turbonator with 4-blades but the difference in pressure is observed between 4 and 8 blades.

While comparing the contour pressure for Turbonator with 4 & 8 blades has attains maximum difference upto 3.669e-2 Pa.

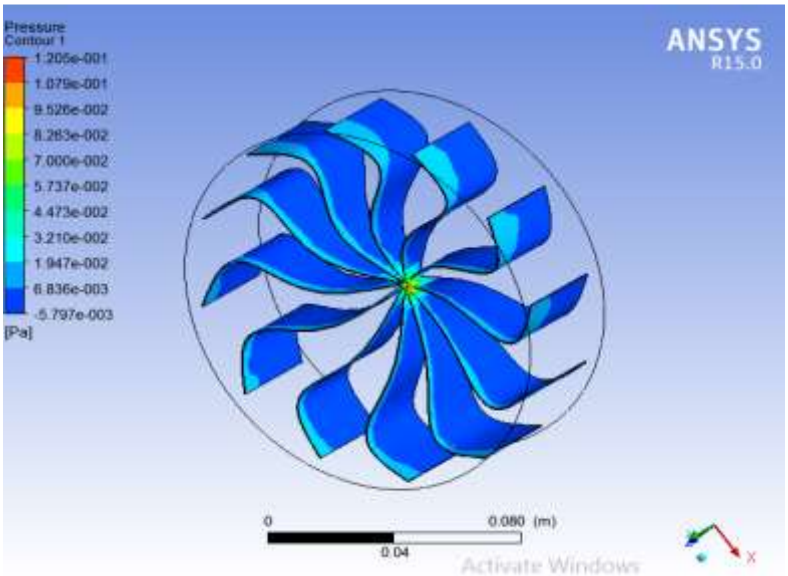

Figure10. Pressure contour- Turbonator with 12 blades

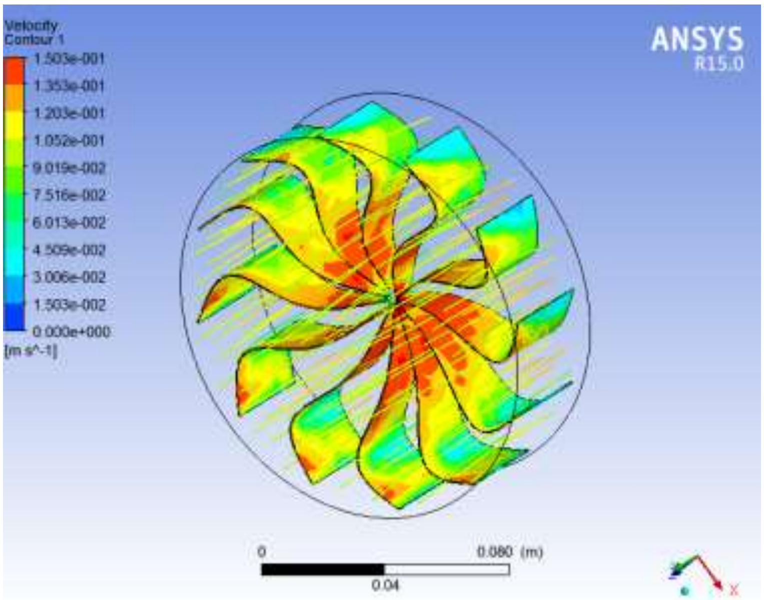

Figure 11. Velocity contour- Turbonator with 12 blades

Turbonator with 12 blades simulation analysis are shown in Figure 10 and 11. It was observed that pressure and velocity increases compared with 4 and 8 blades.

The evaluation of CFD results of contour pressure and velocity to suggest suitable turbonator model are describes as follows.

6. CONCLUSION

The Table 2 shows the comparison of simulation results of pressure and velocity of with (4, 8 and 12 blades) and without Turbonator.

Table 2. Comparison of Pressure and Velocity- With and Without Turbonator

Turbonator with no. of blades	Velocity of air inlet (m/s)	Velocity of air outlet (m/s)	Pressure Outlet (Pa)
Without Turbonator	0.15	0.14999	0.000255
Turbonator(4 blades)	0.15	0.150163	0.004925
Turbonator(8 blades)	0.15	0.150218	0.008031
Turbonator(12 blades)	0.15	0.150313	0.010871

The following conclusion were arrived from the simulations results

(i) The slight variation in outlet velocity between with and without Turbonator was observed because its diameter is same as intake manifold. But difference in velocity observed while increasing number of blades and it was maximum of 0.00032 m/s for 12 blades Turbonator compared to without Turbonator.
(ii) The outlet pressure increases with number of blades because of more turbulence generated in the intake manifold.
(iii) The intake air pressure 19.31,31. 49, 42.63 times respectively for the case of 4, 8 and 12 blades than that of without Turbonator.
(iv) The intake air pressure increases 1.63 and 2.20 times respectively for 8 and 12 blades when compared to Turbonator with 4 blades.
(v) The intake air pressure of 12 blades 1.35 times than that of 8 blades
(vi) Comparatively, the Turbonator with 12 blades produced more turbulence than that of 4 and 8 blades which results in leads increasing volumetric efficiency and better combustion in the diesel engine.

REFERENCES:

[1] Nik Rosli Abdullah, NafisSyabilShahruddin, AmanMohd. IhsanMamat,SalmiahKasolang, AminuddinZulkifli, RizalmanMamat, "Effects of Air Intake Pressure to the Fuel Economy and ExhaustEmissions on a Small SI Engine", Procedia Engineering 68 (2013) 278 – 284.
[2] Mohd Faisal Hushim, Ahmad JaisAlimin, MohdAzahariRazali, AkmalNizam Mohammed, AzwanSapit, Julio Cesar Mendez Carvajal, "Air Flow Behaviour on Different Intake Manifold Angles for Small 4-Stroke PFI Retrofit Kit System", ARPN Journal of Engineering and Applied Sciences.
[3] Nicholas Caplan, Trevor N Gardner, "Optimization of oar blade design for improved performance in rowing", Journal of Sport Science (in press).December 2007
[4] Raga DeepuV, Kumar RopichrlaR.P, "Design and Coupled Field Analysis of First Stage Gas Turbine Rotor Blades", International Journal of Mathematics and Engineering, Vol 13, no.2, pages: 1603-1612.
[5] Sai VastavM, "Automotive Design and Analysis of Turbo Charger with Eleven and Twelve Blades",International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, volume- 3, issue-11, nov.-2015, and pp.120-125.
[6] ShubhamPatil, Jagdish, LokavarapuBhaskara Rao, "CFD Analysis of Turbocharger Compressor to Study the Effect of Geometry Change on Surge and Performance of Compressor", International Journal of Performability Engineering, vol. 14, no. 1, January 2018, pp. 9-16.
[7] Soliman M.I., Emara A.A., Abdel Razek E.M., Moneib H.A., "Modeling and CFD Analysis of Air Flow through Automotive Turbocharger Compressor: Analytical Approach and Validation", 17th International Conference on ASAT - 17 – April 11 - 13, 2017.
[8] Uma Maheswara Rao L, Mohammed Ashif, "Fluid Flow Analysis of Centrifugal Fan by Using FEM", International Journal of Mechanical Engineering and Technology (IJMET), Volume 7, Issue 2, March-April 2016, pp. 45-51.
[9] Chirag V. Kapuria, Dr. Pravin P. Rathod, "Experimental Investigation on Performance and Emission of IC-Engine by Modifying Inlet Manifold - A Review", IJAERD, Volume 2, Issue 3, March -2015.