International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 627-633 www.ijaem.net ISSN: 2395-5252

Performance evaluation of Convergent-Divergent Nozzle Vanes of a Turbocharging System (Theoretical Study)

Uche G. Emmanuel

[1,2]*Uche G. Emmanuel, Federal University of Technology Owerri, Owerri, Nigeria*
Corresponding Author: Olisaemeka C. Nwufo

Submitted: 25-05-2021 Revised: 01-06-2021 Accepted: 05-06-2021

ABSTRACT:As the quest for clean and sustainable energy increases globally, reduction of exhaust emissions which is in line with the SDG 13 (climate action) is very vital and the turbocharger is a key technology in modern automotive internal combustion engines towards achieving this. This technology has been regarded as the next step in the downsizing of I.C. engines and has demonstrated its ability in increasing the power of small engines by approximately 34%.However, this has its limitations with regards to failure as a result of high back pressure, selection of appropriate air-fuel ratio which could either provide better transient response at low load condition or provide increased power at full load-condition, hence Variable Geometry Turbochargers (VGTs) were introduce [1]. This paper takes a theoretical approach toward analyzing the performance of the pivoting vanes of the VGT, to determine how it converts fluid energy to mechanical energy, varies the exhaust gas parameters (pressure and velocity), and by what amount does it increase or decrease the efficiency of the turbocharger at high and low engine speeds.A CFD analysis was performed on the pivoting vanes of the turbocharger at appropriately selected angles of 35^0 and 65^0 vane inclination producing the exhaust outlet pressure and velocity for the turbine volute only.The paper also includes findings from previousexperiments performed by researchers in the same areaas this serves as a validation for the result obtained and purpose of this research work.

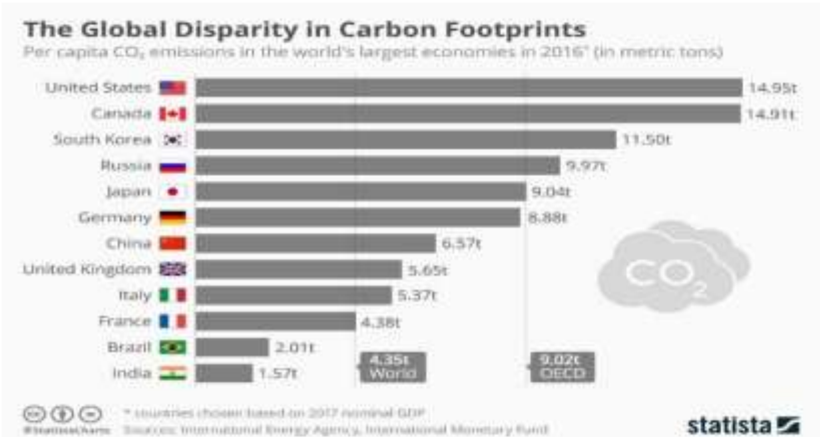
KEYWORDS: turbochargers, computational fluid dynamics, energy conversion, variable geometry turbochargers

I. INTRODUCTION

Turbocharging systems are key systems which increases the power of internal combustion engines, reduce exhaust gas emissions, and specific consumption. However, the application of this technology poses coupling problems between the engine and the turbocharger. For example, at low engine speed with a small mass flow rate, a turbine with a high expansion ratio (i.e., with a small effective section of the exhaust gas passage) is necessary to supply the power needed to meet the compressor requirements. However, for high-speed engine operating points, a turbine with a larger effective areawouldbeenoughtosupplythepower required by the compressor [2]. Maintaining transienttorque response is challenging on turbocharged engines because of the period of time required to accelerate the turbocharger. In addition, the transient response of conventional turbocharged engines is usually slow compared to naturally aspired engines. This is due to the period of time required to accelerate the turbocharger, achieve the target boost pressure, and reach the required engine torque [3]. In a turbocharger, a turbine propelled by exhaust gas is coupled via an axle to a compressor, which in turn boosts engine power by compressing inlet air above its default atmospheric pressure. Regardless of that, one of the main challengesfor turbochargers is a phenomenon known as turbo lag, a delay in boost pressure owing to gaseous and rotational inertia in the system [4]. Therefore, a proper design and analysis on key technologies to be embedded is inevitable in the production of high performance and environmentally friendly turbochargers for internal combustionengines.

As show in figure I below, various countries has set out target to CO_2 emission and in response to these increasing emissions regulations, engine manufacturers around the world have adopted a wide array of turbo charging technologies in order to maintain performance when downsizing their engines. New vehicles around the globe have to comply with emission legislations which are set by the government bodies, with global warming being the mainmotivator. Regularly, these emission legislations become stricter, reducing the allowance of CO, NOX, HCs and other emissions for the automotive industry. Downsized engines have been developed in order to obtain higher thermal efficiencies which are connected with the reduction of fuel consumption and CO2 emissions [5]. Among the evolving turbocharger technologies, a variable geometry turbocharger (VGT) has been recognized as the most profitable system. Its easiness in utilizing exhaust gas in different speeds and loads makes the engine operability applicable in a wide range of operating conditions and transient conditions. Furthermore, it has reached vast markets in the last decade, owing to its capability in improving acceleration performance and reducing exhaust emissions compared with the fixed geometry turbocharger[4]. Variable Geometry Turbine (VGT) turbochargers offer a route to improve the transient response of an engine hence making it one of the available boosting systems that have the potential to achieve rapid transient response and high fuel efficiency over a wide flow range [5]. The VGT can change the gas velocity and flow angle to vary the turbine characteristics In addition, the VGT turbocharger can allow all the exhaust gas to pass through the turbine. This study therefore presents numerical results gotten from the flow analysis performed on the pivotal vanes embedded in the turbine section of the turbocharger to determine exactly how the VGT turbocharger changes the gas velocity and flow angles of the turbine to vary the turbine characteristics.

II. GLOBAL DISPARITY IN CARBON FOOTPRINTS



III. EXPERIMENTATION

The cornerstone of computational fluid dynamics are the fundamental governing equations of fluid dynamics which are; the continuity, momentum and energy equation that define the physics of the flow phenomena. The methods stated below which include the flow modelling, geometric analysis, thermodynamic analysis, were very crucial in obtaining the result of this work. Some of the major assumptions that were made towards carrying out the analysisinclude: The flow is considered as a continuum

- There is no volumetric energy generation in the controlvolume
- The thermo-physical properties are a function of the fluid temperature and are constant throughout theflow.
- There is no combustion occurring in the turbine

IV. MODELING METHODOLOGY:

The first step was identifying best tools to be utilized in performing the modeling. The next step then involvedcollecting appropriate turbocharger design parameters for modeling the turbine volute and nozzle vanes for the purposes of CFD.Thefollowing essential steps were taken:

The Autodesk Inventor and ANSYS software were properly installed in a design machine and setup for design. The turbocharger parameters including inlet & outlet radius, and aspect ratio, were measured with measurement tools from a DAF 2800 Turbo Truck in an automobile garage. The truck was test driven at high engine speed and low engine and parameters such as exhaust gas mass flow rate (inlet), inlet pressure were recorded. This parameters were necessary for CFD analysis in ANSYS. Thiswillcharacterize surrounding fluid flow resulting pressure and velocityof the exhaust gas on the vane surface.

V. THERMODYNAMIC ANALYSIS:

The working fluid used in this analysis isexhaust gas from aDAF 2800 Turbo Truck. The combustion fluid chemistry can take the form:

$$C_xH_y + (X + \frac{y}{4})\, O_2 + \frac{79}{21}(X + \frac{y}{4})\, N_2 \rightarrow xCO_2 + \frac{y}{2} H_2O + \frac{79}{21}(X + \frac{y}{4})\, N_2 \ldots\ldots\ldots\ 1.1$$

The assumed type of hydrocarbon used was Octane, with 20% excess air, the stoichiometric equation is given as follows:

$$C_8H_{18} + (1.2)\ 12.5\ O_2 + (1.2)\ 47.02\ N_2 + 8CO_2 + 9\ H_2O + (0.2)\ 12.5\ O_2 + (1.2)47.02\ N_2 \ldots\ldots\ldots\ 1.2$$

Mass of fuel = (1) (8 × 12 + 1 × 18) = 114 kg/mole

MOLE FRACTION:

$CO_2 \frac{8}{9+8+56.42+2.5}$ X 100% = 10.54%

$H_2O \frac{9}{9+8+56.42+2.5}$ X 100% = 11.58%

$N_2 \frac{56.42}{9+8+56.42+2.5}$ X 100% = 0.74%

MASS FRACTION:

$CO_2 \frac{352}{3159.74+80+352+1}$ X 100% = 0.094

$H_2O \frac{352}{3159.74+80+352+1}$ X 100% = 0.043

$N_2 \frac{3159.74}{3159.74+80+352+1}$ X 100% = 0.841

EXHAUST GAS (C_p):

C_pCO_2: $\frac{12.50}{100}$ x 1.2984 = 0.1623 KJ/KGk

C_pH_2O: $\frac{14.06}{100}$ x 4.187 = 0.5887 KJ/KGk

C_pN_2: $\frac{73.45}{100}$ x 1.2699 = 0.9327 KJ/KGk

Cp of Exhaust Gas at 1323K: = 0.5887 + 0.9327 + 0.1623 = 1.6837 KJ/Kg K

GEOMETRIC ANALYSIS:

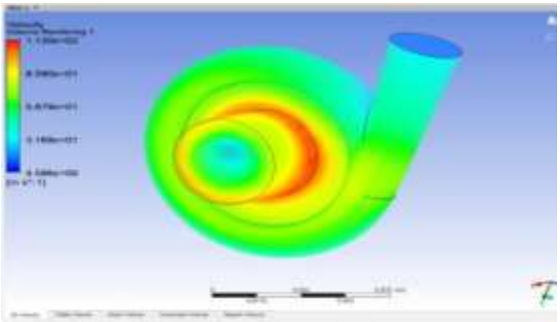
Appropriate engine and turbocharger data for the modelling of the turbine in Autodesk Inventor for the purposes of Computational fluid dynamics in ANSYS was collected from DAF 2800 Turbo Truck. The aerodynamic and thermal analysis was conducted in ANSYS V19.2 using Design Modeler, ANSYS CFX and CFD Post modules. The turbine geometry and the vane assembly were defined in Autodesk Inventor. In accordance with objectives of the project, a volute profile with the appropriate A/R ratio was developed. The number of vanes chosenwas eight (8) in accordance with the already existing turbocharger in the truck. Below shows the 3D model as generated in Autodesk Inventor showing the eight vanes:

Turbocharger showing eight vanes

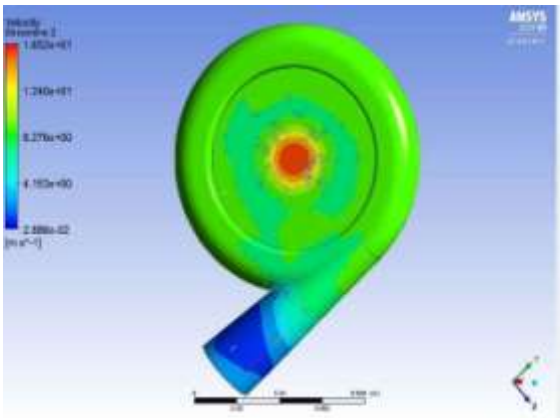
The turbine volute was imported into ANSYS to begin CFD on the turbine volute. With 51,223 nodes and 261,125 elements. For the vane position at 35° 51,223 nodes and 261,125 elements wereused.Finally,forthevanepositionat 65°, 51,223 nodes and 261,125 elements were used. Although the mesh is coarse, this is because a student version of ANSYS CFX was used and does not exceed this speculated amount of nodes and element given by ANSYS for a student version.The geometry of the vanes as seen inFigure II, required a turbulent flow analysis in order to accurately predict the pressure co-efficient acting across the length of the chord. For this purpose, a k – ω turbulencemodel was utilized in performing the analysis. Intensive study and research in fluid mechanics and heat transfer showed that the k – ωwas designed specifically for applications such as aerospace and also finds application in turbines.

VI. RESULTS:

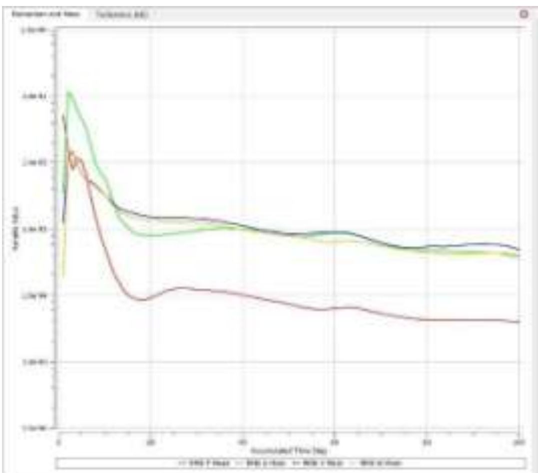
The fig; 7-9 shows the results obtained from the simulation performed for the three turbine geometries with small, medium and large air fuel ratios.



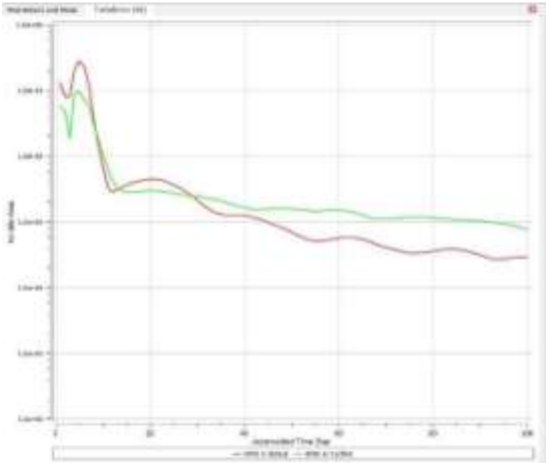
Volume Rendering - Velocity Plot



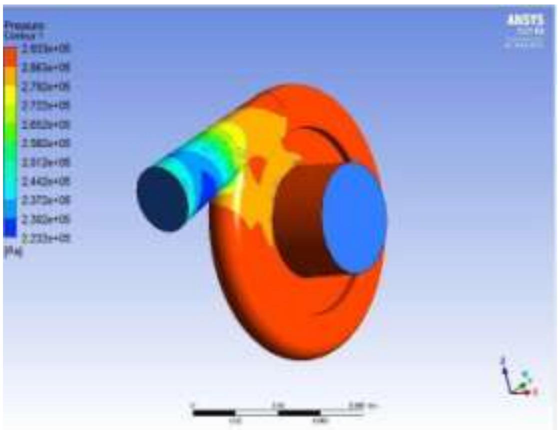
Velocity Plot for 35° inclination of the vane



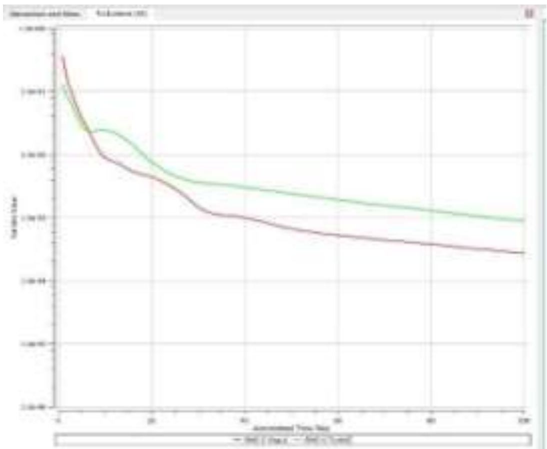
Velocity Contour Plot - Representing Nozzle vane at 35°



Turbulence Plot for 35^o Vane Inclination

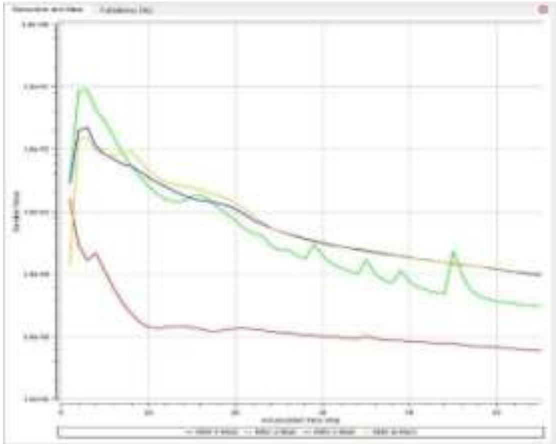


Pressure Plot - 65^0



VIII. Turbulence Plot - 65^0

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 627-633 www.ijaem.net ISSN: 2395-5252



IX. Momentum Plot - 65^0

VII. DISCUSSION:

VOLUTE PLOT:

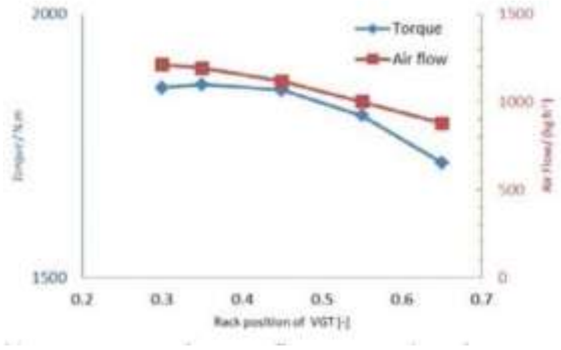
Fig III – VI shows an indication of the varying pressure and velocity of the exhaust gas in the turbine volute for a decrease in inlet area of the volute which represents a 35° inclination of the pivoting nozzle vanes in the turbine volute at low engine speed. From the contour plots it can be observed that at a low engine speed of 4.15m/s, the pivoting vanes converges thereby reducing the inlet area of the exhaust gas into the turbine volute. Following continuity equation offluid flow, for a converging nozzle, the velocity is expected to increase towards the exit of the nozzle. From the contour plots above in Fig III we can deduce that the velocity is highest at the exit of the turbine volute. In this analysis, the behavior of the exhaust gas at the exit is of major interest for all inclinations of the pivoting vanes. The contour gives a good visualization as it is observed that the velocity increases to 16.52m/s from 4.15m/s.

Fig VII – IX shows an indication of the varying pressure and velocity of the exhaust gas as it travels in the turbine volute. This analysis was performed for an input velocity of the exhaust gas into the turbine volute. The exhaust gas pressure in the volute varies significantly with the x-axis, having its highest values near the walls of the volute and reducing significantly towards the center of the volute. Thisoccursbecause the rotor fluid zone is located at the center of the volute, and since the exhaust gas expands, its pressure drops in that zone giving rise to a lower pressure value at the center of thevolute. Supporting this observation is the fact that the average inlet pressure is about 11180Pa while that towards the rotor outlet is 309.3Pa. The pressure increase significantly as the exhaust gas flow through the diffuser to an average pressure of 4966.6Pa enabling the fluid to flow out of the volute region. Since a positive pressure differential is required for an outflow. Negative Pressure values are noticed very close to the center ofthe volute region. This is due to the suction effect created by the shaft hole in the rotorgeometry.

From the contour plots it can be observed that at a high engine speed of 45m/s, the pivoting vanes diverges thereby increasing the inlet area of the exhaust gas into the turbine volute. Following continuity equation of fluid flow, for a diverging nozzle, the velocity is expected to drop towards the exit of the nozzle as the pressure increases. From the contour plotsaboveinFig4.8wecandeducethat the pressure increases towards the exit of the turbine volute. In this analysis, the behaviour of the exhaust gas towards the exit is of major interest for all inclinations of the pivoting vanes. The contour gave a good visualization as it is observed that the pressure increases to 293300Pafrom 223200Pa.

VIII. RESEARCH RESULT VALIDATION:

Variable Geometry Turbine (VGT) turbochargers offer a route to improve the transient response of an engine hence making it one of the available boosting systems that have the potential to achieve rapid transient response and high fuel efficiency over a wide flow range [5]. From the results obtained above we could observe that the turbocharger has an improved response at transient engine conditions as the pressure of the gas increases at a high vane inclination at high engine speed. This creates a larger passage of theexhaustgasasittravelsintheturbine section of the turbocharger hence the vane tends to incline to a larger angle. The vane also responds to a decrease in engine speed by reducing the vane angle there by creating a small passage of the exhaust gas and increasing the velocity as it flows through the volute. Hence the results obtained clearly agree withwhathas been stated in literature on variable geometry turbochargers. The chart below shows the comparison of the vane rack position of the VGT and the airmass flow rate. From the chart above, we observe that as the VGT rack position increases, the flow capability of the VGT increases [6].



X.Impact on air mass flow rate and engine torque

IX. CONCLUSION:

Analysis of all these flow variables has given an insight into the fluid dynamics characteristics of flow around the turbine section of a turbocharger and has supported the turbo-machinery theory that, turbines are power producing devices that converts flow energy into mechanical energy by imparting kinetic energy on the fluid through the impeller and it also proved that the gas dynamics theory of Laval (converging and diverging) nozzles are valid and holds in the design and operation of Variable Geometry Turbocharger. Theoretical evaluations performed using the data obtained from the analysis shows thatthe pivoting nozzle vanes has a great positive impact on the flow of the exhaust gas in the turbine section of the turbocharger at high, and low engine speed of the automobile which in turn increases the efficiency of the turbocharger, reduce exhaust emission, and increase engine power. More so, this will drastically solve the problem of wobbling turbine blades and back pressure in the turbocharging system as the engine speed of the automobile varies.

REFERENCES

[1]. Vivek, S;Apostolos, P; and Shettigar, H., 2014, “Materials Selection for Variable Geometry Turbine Nozzle for Gasoline Engine Application,” ASME Turbo Paper No. 98102

[2]. INSTITUTE OF PHYSICS JOURNAL, J Institute of Physics Publishing 2004;15(6):1185}1187 (DOI:10.1088/0957-0233/15/6/020)

[3]. Akehurst, L; Sam B; and Smith, L;2014, “Optimisation of transient response of a gasoline engine with variable geometry turbine turbocharger,” IMECHE Paper No. 10.1533/978081000342.163.

[4]. JOURNAL OF ENERGIES, J energies 2018; 287(11):1}2 (DOI: 10.3390/en11020278)

[5]. Tang, H. Application of Variable geometry turbine on gasoline engines and the optimization of transient behaviours. PhD Thesis, University of Bath, UK,2016.

[6]. INTERNATIONAL JOURNAL OF FLUID MACHINERY AND SYSTEMS, J STAGE 2016; 9(4):332}335 (DOI: 10.5293/IJFMS.2016.9.4.332)