DESIGN OPTIMISATION OF CONVERGENT-DIVERGENT AIRCRAFT NOZZLE

Tolumoye J. Ajoko[1] and Tolumoye J. Tuaweri[2]

Abstract— The challenges of convergent-divergent nozzle to be used in aero gas turbines have been a bottleneck for research. This paper is therefore geared towards proffering solutions in this respect. The study is carried out using computer based CFD Ansys Fluent simulation code with reference to Solidworks CAD tool for modeling. Analyzed results from study indicate $2.342\times10^{8}Nm^{-2}$ of total pressure against $4.176\times10^{8}Nm^{-2}$ of dynamic pressure of the designed model for the research. Reduction in this performance is observed as the fluid leaves the throat section and proceeds to the divergent region of the nozzle. This attests distribution of pressure in the nozzle as flow accelerates subsonically or supersonically with reduction in pressure drop. Also, turbulence intensity at the nozzle exit is estimated to be 14.7749% as vortex flow is created in the model. Meanwhile, as flow gets stabilized, the turbulence intensity reduces gradually to 0.0026%. This reduction and low turbulence intensity is in line with a laboratory test result in the literature. Consequently, low efficiency production on the part of convergent-divergent nozzle from study is identified as shock wave formation in the divergent portion of the model. The study confirms that CFD simulation code and Solidworks are effective predictive and prognostic engineering softwares capable of modelling, simulating and analysing thermo-fluid characteristics and hence convergent-divergent aircraft nozzle.

Index Terms—Aero Gas Turbine, Axisymmetric, Combustion Chamber, Convergent-Divergent Nozzle, Mach Number, Model, Shock Wave, Throat, Turbulence Intensity

1. INTRODUCTION

The utmost objective of aerospace engineering is to enhance more effective, efficient and reliable methods and equipment to transport payloads into space. The range and payloads are so strongly affected by the performance of the propulsion system as a whole; that it is necessary to use both a continuous variable intake and a nozzle capable of both throat and exit areas. The payloads are directly related to specific impulse (I_{SP}) which is one important criterion for propulsion system. It is noted that optimizing thrust for high altitude where ambient pressure is reduced yields high specific impulse [1]. Hence, the demand for high performance propulsion engine like the rocket has led to the critical assessment of its engine subsystems with the intent of minimizing losses. These losses such as flow rate, speed, direction and pressure of steam, etc are controlled by the nozzle system of an engine as they tend to pass through its exhaust system. Thus, the function of the nozzle in anengine is to harness energy made available by the propellant particles rejected in the combustion chamber, creating escape pressure which is turned into thrust force for uplift motion. The nozzle asymmetric configuration is designed to increase the speed of an outflow fluid from its system and equallycontrols its direction and shape.

Consequent of this positive development on the propulsion subsystem of aero gas turbines; numerous studies [1], [2], [3], [4] have been carried out with the goal of improving the propelling nozzle system for thetransformation of aerospace engineering. As such, designing an air vessel with nozzle system meeting the requirements for both environmental and economic metrics such as noise pollution, takeoff and cruise aero and propulsion performance like weight, mechanical complexity and structural reliability is the target of aerospace engineers [5]. It is reported [5] that the operation of the engine is mostly controlled by the geometry of the propelling nozzle either equipped with an afterburner or a reheat system. This also controls the strength of the shock waves which are capable of mixing high and low speed steams efficiently which create noise spectral outside the weighting range of the system [5], [6]. According to [7], the noise generated by the jet engine is a function of the jet velocity to a high power and the jet area which is predominantly from the exit of the nozzle. Furthermore, the results of a research [8] on convergent-divergent nozzle with and without obstacle at its exit section attests that, the Characteristic features for visual comparisons are in the expansion domains at exit corners and inclinations of shock waves generated at aft-throat positions, where radial shape in divergent nozzle domain changes to linear. From the study, it was confirmed that all the airflow parameters were in good agreement both qualitatively and quantitatively [8].

Massive thrust pressure and temperature created by the ignition of rocket fuel within the engine will lead to stress in the internal material of a nozzle cone. This stress acts on the

- *Tolumoye J. Tuaweri is a senior lecturer in the Department of Mechanical Engineering of Niger Delta University, Wilberforce Island, Bayelsa State. He is a research scholar and PhD holder from Loughborough University, UK.. Email:tuaweri@yahoo.com*
- *Tolumoye J. Ajoko is a lecturer of the Department of Mechanical Engineering of Niger Delta University, Wilberforce Island, Bayelsa State. He is a research scholar in Thermo-fluid option of Mechanical Engineering currently pursuing PhD degree program in Wind Turbine Renewable Energy at Niger Delta University. Email: johntolumoye@yahoo.co.uk*

circumferential direction of the nozzle and tends to blow it apart. Also, a major problem in a convergent-divergent nozzle is the flow used for modeling the compressible process of the system. Thus, the Occurrence of shock in the flow field displays one of the most prominent effects of compressibility over fluid flow [9], [10], [11]. Therefore, accurate shock prediction is a challenge hence nozzle research is redirected towards the design of nozzle cone capable of withstanding the extreme forces and temperatures generated within the nozzle during engine operation. These challenging features also have pressure gradients, velocity distribution, eddy location, stream line curvature, and stream wise vortices [12], [13]. Other challenges of the convergent-divergent nozzle would result to significant increases in engine weight, length, and diameter which would turn to major installation difficulties and penalty in aircraft weight. Likewise, at an operating pressure ratio less than the design value; a convergent-divergent nozzle of fixed proportions would certainly be less efficient because of the loss incurred by the formation of a shock wave at the divergent portion [14]. Meanwhile, the impact of the varying area along the convergent-divergent nozzle is another research concern. This reduction in area has caused velocity to reduce as opposed to increase. Nevertheless, this factor has created diameter constraint restricting the nozzle; and the exit area is inevitably far smaller than that to *run full* and achieve *fullflow* acceleration [15].

Therefore, it has become necessary to redesign the convergent-divergent nozzle to achieve maximum optimization for aircraft engines. However, an optimum design of the nozzle can be achieved if the real solution of a nozzle concept that is effective in reducing noise at a low jet exhaust velocity and weight reduction is attainable; then, the aero-acoustic suppression characteristics and the suppressed mode performance of the convergent-divergent nozzle design is viable. Thus, the effectiveness of this research work to achieve positive results for an optimal convergent-divergent nozzle design for aero gas turbine lies on the proficient use of Computational Fluid Dynamics (CFD) – ANSYS Fluentanalysis software into the airflow part of the nozzle in order to model, simulate, and analyze the gas flow path of the system.

2. GEOMETRIC ANALYSES

In order to design and model the nozzle there are various guidelines that has to be followed. The rocket nozzle equation is solved to enable accelerate the gases as it passes through the nozzle path. Also, the equation is to obtain the relationship between the exit velocity, pressure and mass flow rate. It reveals that input variables are vital for this task, thus two categories are considered.

They are the geometric variables which enable the creation of model geometry and a fluid property variable which is needed to define a constant specific heat fluid and inlet properties. Additionally, a predefined Mach number distribution $M(z)$is necessary to define the nozzle's inviscid area expansion $A(z)$from the throat depth z=0 (corresponds to sonic flow Mach number is unity) to the nozzle outlet depth $z = z_e$using the isentropic relation in equation 1 [16].

$$\frac{A(z)}{A_{th}} = \frac{1}{M(z)}\left[\frac{1+1/2(\gamma-1)M(z)^2}{1/2(\gamma+1)}\right]^{\frac{\gamma+1}{2(\gamma-1)}} \quad (1)$$

Instead of the interpolation to generate intermediate cross sections; the $A(z)$data calculated from Equation 1 using the predefined $M(z)$is represented by a continuous function developed from the Agnesi family of curves as presented in equation 2.

$$\frac{A(z)}{A_{th}} = F\left(1+\cos\left(\frac{\sqrt{\pi}}{\tan^{-1}\frac{z_e}{r_{th}D}}\tan^{-1}\frac{\frac{z}{r_{th}}-\frac{z_{eG}}{r_{th}}}{D}\right)^2+1\right) \quad (2)$$

The function of the nozzle is to accelerate gases produced by the propellant to maximum velocity in order to obtain maximum thrust. The amount of thrust produced by the engine depends on the mass flow rate through the engine, the exit velocity of the flow, and the pressure at the exit of the engine. The value of these three flow variables are all determined by the governing equations for rocket nozzle. Thus, for steadily operating rocket propulsion system moving through a homogeneous atmosphere total thrust and specific impulse are presented in equations 3 and 4 respectively.

$$F = \dot{m}.v_e + (p_e - p_o).A_e \quad (3)$$

$$I_{sp} = \frac{F}{\dot{m}g_0} \quad (4)$$

The rocket nozzle is usually so designed that the exhaust pressure is equal or slightly higher than the ambient fluid pressure. Hence, changes in ambient pressure affect the pressure thrust, therefore variation of the rocket thrust with altitude between 10 – 30% for Velocity of sound and Mach number is presented in equations 5 and 6

$$\alpha = \sqrt{\gamma.R.T} \quad (5)$$

$$M = \frac{v}{\alpha} \quad (6)$$

Other properties of the nozzle are best referenced against the stagnation properties with assumptions of an ideal gas and isentropic flow. Ratios of pressure, density and temperature can be related to thestagnation temperature,

ISSN 2229-5518

pressure and density at a given Mach number as seen in equations 7 – 9.

$$\frac{T_0}{T} = \left[1 + \frac{(\gamma - 1)}{2} M^2\right] \quad (7)$$

$$\frac{P_0}{P} = \left[1 + \frac{(\gamma - 1)}{2} M^2\right]^{\frac{\gamma}{\gamma - 1}} \quad (8)$$

$$\frac{\rho_0}{\rho} = \left[1 + \frac{(\gamma - 1)}{2} M^2\right]^{\frac{1}{\gamma - 1}} \quad (9)$$

3. MATHEMATICAL MODELLING

The study is focused on a convergent-divergent nozzle; hence a model of the nozzle is produced using a computer aided design tool known as Solidworks. The nozzle is modeled as a part with lines and a revolve features in the solidworks interface. The principles of symmetry is applied in the design since the model is symmetrical about the centerline and quarter model was sketch and revolved. Figure 1 is a sketch of the model; while, completed model is presented in figure 2.

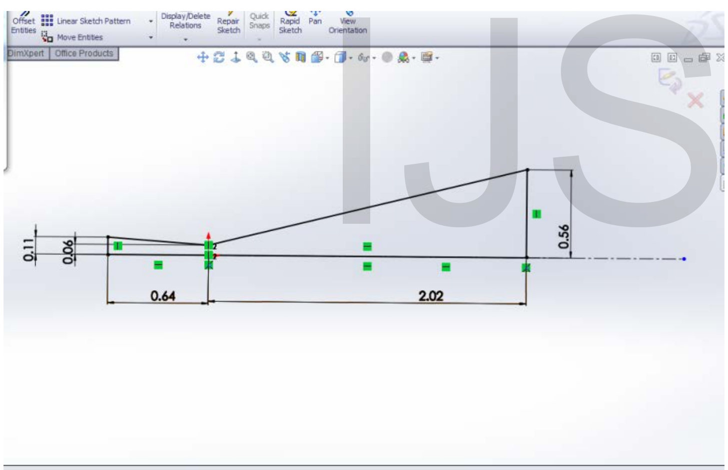

Fig. 1: Solidworks Sketch of Nozzle

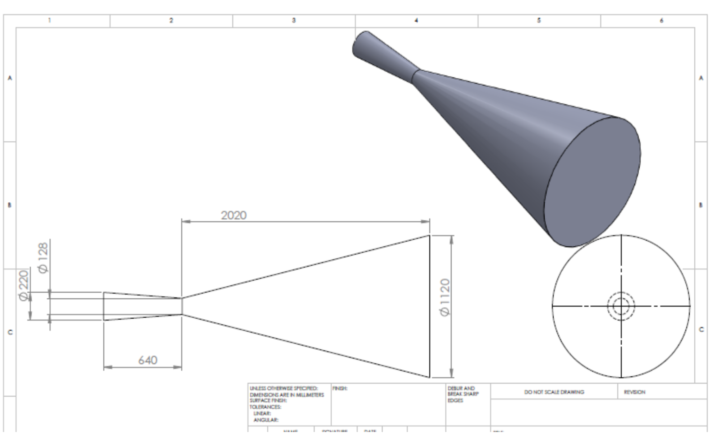

Fig. 2: Solidworks Model of Nozzle

The plots displacement thickness curves along the outer wall for viscous consideration according to a reviewed literature for T_w = 500°K. Then wall temperature of 500°K is defined as a conservative value based on the expectation that the nozzle material is an aluminum alloy where the material strength has not been compromised since the melting temperature for aluminum is 930°K [17]. Numerically, validating Edenfield's method of displacement thickness of the model geometry which gives $\delta^*_{out\,e}$ = 0.0049m from an evaluation using equation 10 – 14 against an estimated value from Nozzle Performance Analysis Code (NPAC);$\delta^*_{out\,e}$ = 0.0018m. The Edenfield's method is considered since it is more conventionally estimated. Thus an estimated swept wall corresponding to Edenfield's method by evaluation is $\delta^*_{sw\,e}$ = 0.0053m and outer wall arc angle of $\boldsymbol{\psi_e = 44.5°}$ which is used in the design process.

$$\delta^* = \frac{21}{50} \times \frac{L}{Re_{ref}^{0.2775}} \quad (10)$$

$$Re_{ref} = \frac{\rho_{ref} \times V_i\, L}{\mu_{ref}} \quad (11)$$

$$V_i = M_i \sqrt{\gamma R T_i} \quad (12)$$

$$\boldsymbol{\psi_e = x - tan^{-1} \frac{y_i}{x_i}} \quad (13)$$

$$L = \sum_{i=2}^{N_i} \sqrt{(x_i - x_i - 1)^2 + (y_i - y_i - 1)^2 + (z_i - z_i - 1)^2} \quad (14)$$

where L is the physical location downstream as measured from the nozzle throat along the given inviscid nozzle wall from i = 1 at the throat to cross section Ni andthe swept wall $\delta^*_{sw\,e}$defines L along points $P1$.

The model is imported to CFX-Pre for a pre-processing. Meshing involve discretization of the domain into finite volume. For this analysis due to the simplicity of the mathematical model, a CFD Mesh is used with 221635 elements and 42098 nodes. Prior to the mesh, the geometry is first prepared for analysis, the location for boundary conditions are indicated, which includes: the pressure outlet, the wall and the velocity inlet indicated as (p outlet, wall and v inlet). Figure 3 is the mesh model.

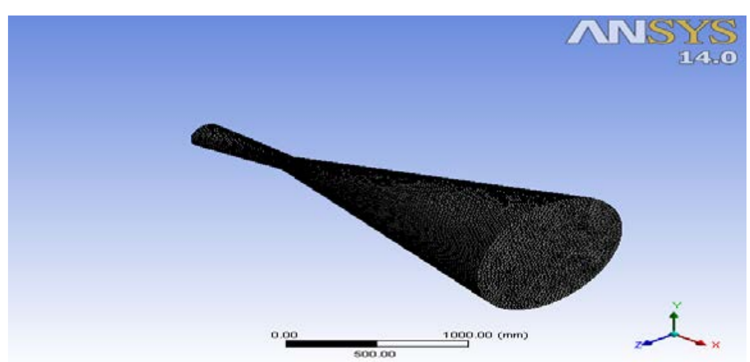

Fig. 3: Mesh Model

4. SETTING OF DESIGN MODEL

The CFD ANSYS Fluent 14.5 code is used to compute the flow properties within the flow path of the nozzle domain using time-dependent, three-dimensional, compressible Navier-Stokes equations as presented in equations 15 – 19. In this interface, a CFX statistical turbulence model is introduced which is capable of accounting for fluctuating quantities in a steady flow. The process is analyzed using density base solver with absolute velocity formulation methods. In setting the viscous model; the K-epsilon with a realizable model and standard wall function is used. The energy equation is put on to solve for the vast effect of temperature variation due to the nature of the nozzle. The well-established $k-\varepsilon$model has proven to be stable and numerically robust where k is the turbulence kinetic energy. The viscous model setting is shown in figure 4.

$$K = \frac{1}{2}\overline{u_i' u_j'} \qquad (15)$$

$$\varepsilon = -\frac{\mu}{\rho}\overline{\frac{\partial u_i'}{\partial x_j}\frac{\partial u_j'}{\partial x_i}} (16)$$

$$u^* = C_\mu k^{1/2} (17)$$

$$\mu_{eff} = \mu + \mu_t (18)$$

$$\mu_t = \frac{\rho C_\mu k^2}{\varepsilon} (19)$$

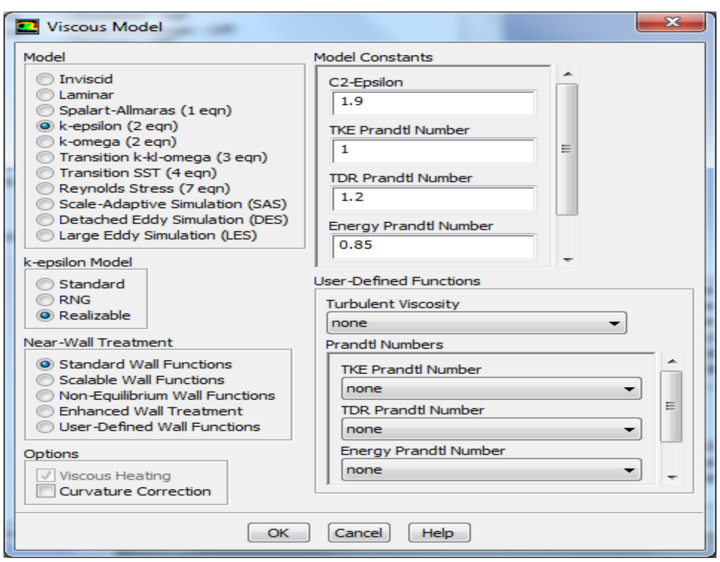

Fig. 4: Viscous Model Setting

Also, in the setting is the selection of materials for the designed nozzle model. Thus, the analysis is a single phase flow hence hydrogen fluid is selected. The reason for hydrogen consideration in the midst of other fluids is due to its relevance as fuel for rocket aircraft. The analysis for liquid hydrogen fuel for rocket engine in this simulation study is to withstand the test of time in the design of the convergent-divergent nozzle for aero gas turbines. To add hydrogen as the fluid, the fluent database is loaded and hydrogen is selected as the fluid to be analyzed. This is to change the default fluid – air in the simulation tool. Figure 5 and 6 shows how to create liquid-hydrogen as a fluid and adding it as the material for this analysis.

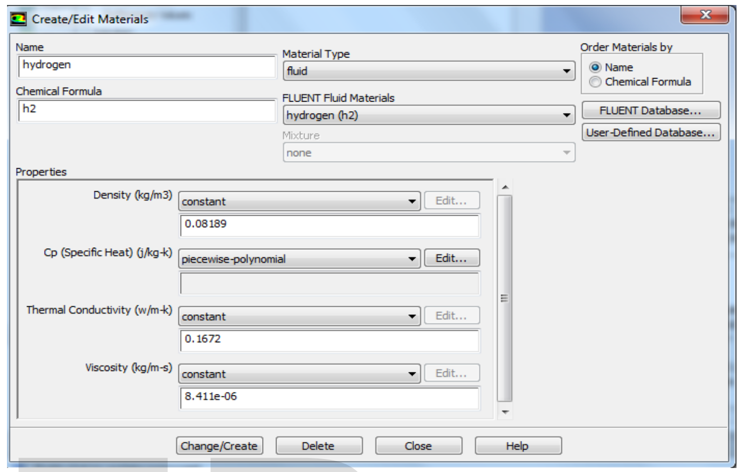

Fig. 5: Properties of Hydrogen as fluid for the analysis

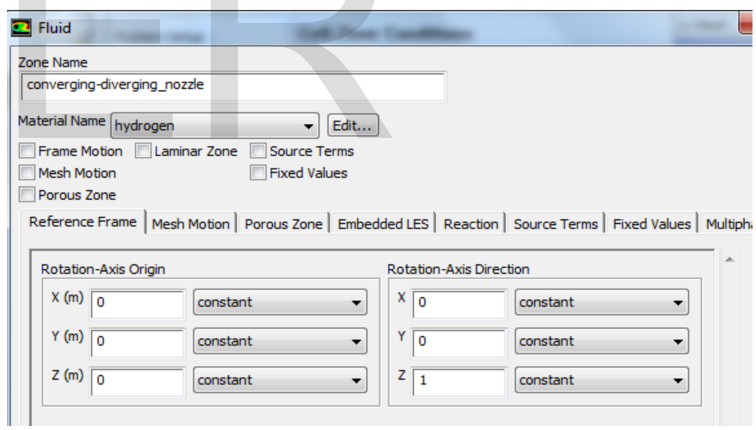

Fig. 6: Adding Hydrogen as the material for this analysis

In order to run accurate simulation process, it is necessary to set the boundary conditions of the model to be simulated. However, the simulation programme understands the equations above and run its calculations putting these expressions into consideration. Hence, two boundary conditions (Inlet and Outlet) are set for this analysis. The inlet conditions take care of the mass flow inlet and temperature which is set as 100kgs-1 and 1000°k respectively. Consequently, the outlet conditions were all set; like the pressure outlet set with a pressure equal to atmospheric pressure. Figures 7 – 10 shows these conditions.

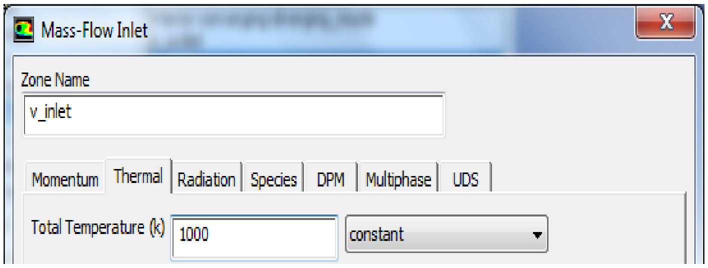

Fig. 7: Temperature of Hydrogen gas at inlet

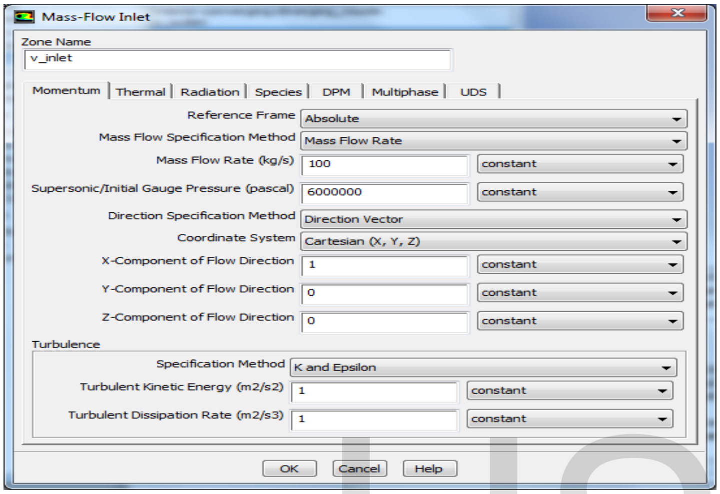

Fig. 8: Momentum Boundary Condition at inlet

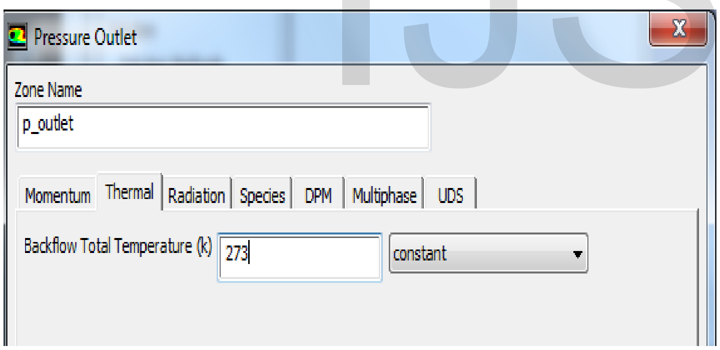

Fig. 9: Thermal Boundary Condition at outlet

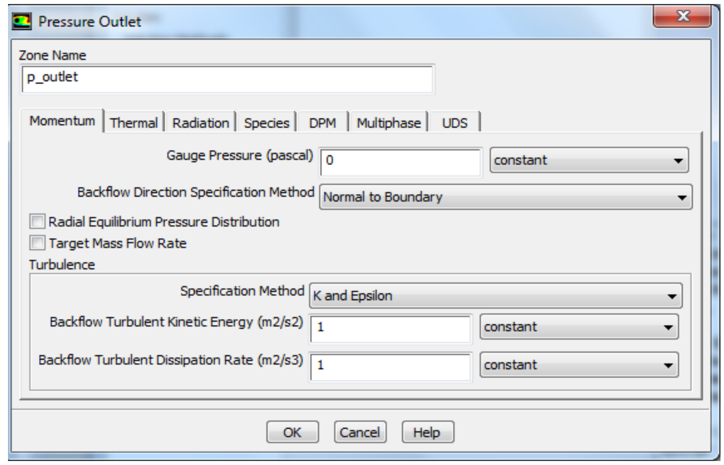

Fig. 10: Pressure Outlet Boundary Condition.

5. PRESENTATION OF RESULTS

The pre–processing stage is the most time consuming section though it involves the generation and discretization of the solution domain, the specification of the properties and setting of the boundary conditions. The simulation of fluid flow problems takes a longer time to converge; the solution involves specification of the numerical method to be used; as a result various turbulence models are created to simplify this procedure. Therefore, the result of solving the Navier-Stokes equation for a viscid and inviscid flow through a convergent-divergent rocket nozzle is presented in figures 11-.14.

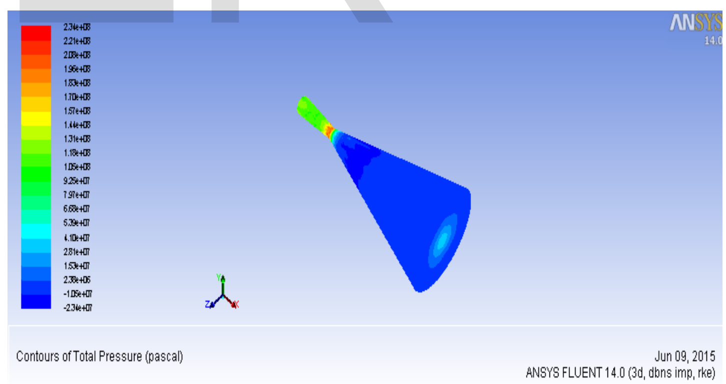

Fig. 11: Total Pressure across Nozzle

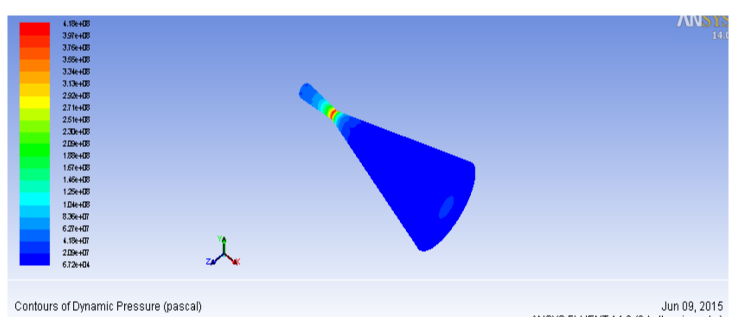

Fig. 12: Dynamic Pressure across the Nozzle

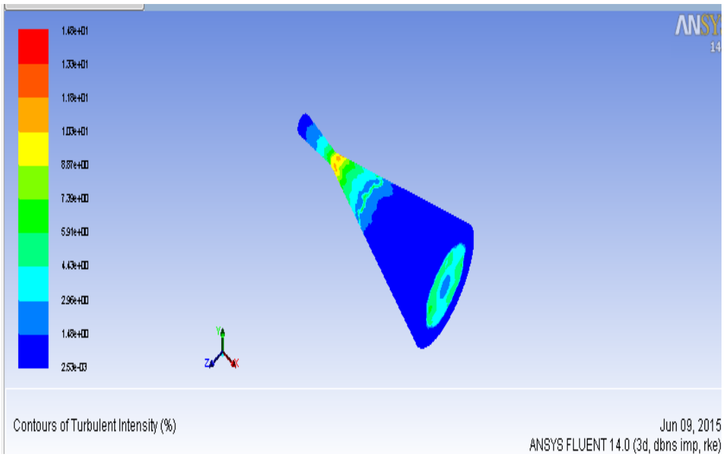

Fig. 13: Plot of Turbulence Intensity across Nozzle

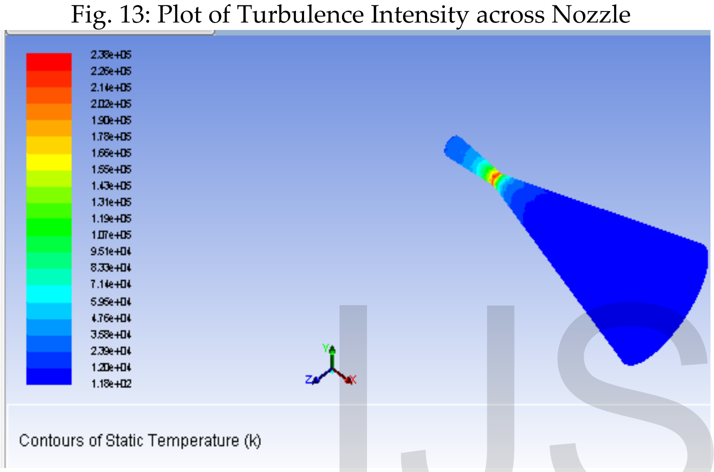

Fig. 14: Temperature distribution across Nozzle

6. DISCUSSION OF RESULTS

The simulated result for the total pressure of a gas flow across a nozzle for axisymmetric model as illustrated in figures 11 and 12 analyses the combustion by control volume method with CFD code. The simulated maximum total pressure in the model is $2.342\times10^{8}Nm^{-2}$, against an average dynamic pressure of $4.176\times10^{8}Nm^{-2}$. This highlights the effect of pressure on the model. As the fluid approaches the throat from the convergent section, the pressure increases and as it leaves the throat to the divergent section of the nozzle the pressure reduces significantly. This reduction continuous as the fluid flow far-field move towards the nozzle exit, thus at this point the minimum pressure cannot be obtained. The observed result confirms the scenario of pressure distribution in a nozzle as revealed in the literature where pressure drops drastically when flow accelerates either sub-sonically or supersonically. The result for intensity as shown in figure 13 describes the designed nozzle for streamline flow. Thus, the simulated result shows that high turbulence intensity is low in the combustion chamber as compared to the divergent section of the nozzle. The evaluated turbulence intensity at nozzle exit is about 14.7749%. This is as a result of the vortex flow created in the model. Furthermore, at downstream as flow gets stabilized, the turbulence intensity reduces and steadily decreases to 0.0026%. This result confirms and validates a study in the open literature on CFD analysis of a rocket nozzle with four inlets system [4]. It creates a uniform flow with low turbulence intensity at the nozzle exit. However, in the temperature distribution analysis in figure 14; it shows the effect of the temperature on the model mostly the throat section such that as the fluid flows across the nozzle the temperature increases rapidly. This is due to the formation of shock wave.

7. CONCLUSION

CFD simulation code allows the detail study of analysing the performance characteristics of thermo-fluid properties such as the pressure, speed, velocity, temperature, mass flow rate, etc of an aero engine nozzle. These parameters of the engine are in charge of maximizing the kinetic energy of the fluid by utilizing the heat and pressure in the combustion chamber as they passes through the nozzle. However, one significant challenge as revealed in the reviewed literature is the increase in engine weight as a result of the use of convergent-divergent nozzle in aero gas turbines. Meanwhile, revealed results in the study have justified and accommodated this problem since pressure reduced drastically as flow leaves the throat and flow along the divergent section of the nozzle. The reduction in pressure is a simple confirmation that the weight is equally reduced because the pressure depends on the weight and the area in the expression of pressure as force per unit area; hence, if weight decreases it implies that pressure will decrease as well and vice versa. Thus, in this relationship; the generic expression of Newton's second law can be transformed to express weight as a force and it is substituted in the pressure equation.

As results indicate the formation of shock wave in the model; it is recommended that future design should consider convergent-divergent nozzles to operate at pressure ratios for which they are designed. Thus, the formation of shock wave in the design process is put into consideration because the flow will be accelerating under the influence of the pressure drop. Off-course, this will retard the conditional changes of the thermodynamic properties of fluid flow across the nozzle system in aero gas turbine in order to increase the thrust of the engine. Also, further design optimisation can be carried out using advanced algorithm that can combine the result from the CFD calculations and structural based design modeller such as the optimisation tool box in Ansys Mechanical.

Based on the improved geometric performance on the model, it is established that the CFD simulation tool utilised is capable of diagnosing and proffering solutions to the challenges of convergent-divergent nozzle of aero gas turbines.

NOMENCLATURE

k Turbulence Kinetic Energy
ε Turbulence Eddy Dissipation
A Cross section area
M Mach number
γ Specific heat ratio
$\dot{m}$ Mass flow rate
I_{sp} Specific impulse
ψ Aarc angle
R Gas constant
T Static temperature
α Air/rocket mass flow ratio
T_0 Total temperature
P_0 Total pressure
δ^* Displacement thickness
e Outlet
μ Dynamic viscosity
Re Reynolds number
ref Reference
ρ Density
μ_t Turbulence viscosity
i Index

REFERENCES

[1] M. Pires,"Turbulence Modeling and Applications to Aero-spike Plug Nozzle", 26th *International Congress of the Aeronautical Sciences*, 2008.

[2] V. Venkatesh, and C. Jaya pal Reddy, "Modelling and Simulation of Supersonic Nozzle Using Computational Fluid Dynamics", *International Journal of Novel Research in Interdisciplinary Studies*, Vol. 2, Issue 6, pp. 16-27, 2015.

[3] P. Parthiban, M. Robert sagayadoss, and T. Ambikapathi, "Design and Analysis of Rocket engine Nozzle by using CFD and Optimization of Nozzle Parameters", *International Journal of Engineering Research-Online*, Vol. 3, Issue 5, pp. 312-319, 2015.

[4] K. M. Pandey, and S. K. Yadav, "CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1", *International Journal of Chemical Engineering and Applications*, ISSN: 2010-0221 ,Vol. 1, No. 4, 2010.

[5] J. M. Seiner, and M. M. Gilinsky, "Nozzle Thrust Optimization While Reducing Jet Noise", *AIAA Journal*, Vol. 35, No. 3, pp. 420-427, 1997.

[6] Propelling nozzle – wikipedia, available at https://en.wikipedia.org/wiki/propelling_nozzle, Accessed on the 24th October, 2016.

[7] D. C. Willard, J. A. Mihaloew and E. E. Callaghan, "Turbojet Engine Noise Reduction with Mixing Nozzle-Ejector Combinations", *National Advisory Committee for Aeronautics – Technical note* 4317, 1958.

[8] O. P. Kostić, Z. A. Stefanović and I. A. Kostić, "CFD Modeling of Supersonic Airflow Generated by 2D Nozzle With and Without an Obstacle at the Exit Section", *FME Transactions*, Vol. 43, No 2, pp. 107-113, 2015.

[9] F. Frank, F. C. George, A. Johnson, D. Prohaska, and J. Hart, "Rocket Nozzle Design", Marquette University, Malwaukee, (Unpublished Rigorous Global Search Working Note 5), 1998.

[10] G. R. Rajeswara, U.S. Ramakanth, and A. Lakshman, "Flow Analysis in a Convergent-Divergent Nozzle Using CFD", *International Journal of Research in Mechanical Engineering*, Vol. 1, Issue 2, pp.136-144, 2013.

[11] K.P.S.S. Narayana, K. R. Sadhashiva, "Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis", *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),* e-ISSN: 2278-1684,p-ISSN: 2320-334X, Vol. 13, Issue 4. pp. 58-65, 2016.

[12] G. Satyanarayana, C. H. Varun, S. S. Naidu, "CFD Analysis of Convergent-Divergent Nozzle", *ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering*, ISSN 2067-3809, 2013.

[13] P. C. Wang, "Large eddy simulation of high speed convergent-divergent nozzle flows", PhD Dissertation, Loughborough University, UK, Unpublished PhD Thesis, 2013.

[14] H.I.H. Saravanamuttoo, C. F. C. Rogers, H. Cohen, and P. V. Straznicky, "Gas Turbine Theory (6th Ed)", Pearson Prentice Hall: Edinburgh Gate – Harlow, England, pp. 110-111, 2009.

[15] P. P. Walsh, and P. Fletcher, "Gas Turbine Performance(2nd Ed)", Oxford: Blackwell Science, pp. 216,401; 2004.

[16] D. J. Cerantola, “Rocket Nozzle Design with Ejector Effect Potential”, Carleton University, Canada, Unpublished MSc Thesis, 2007.

[17] G. P. Sutton, “History of Liquid Propellant Rocket Engines”, American Institute of Aeronautics and Astronautics, 2006.