

Experimental Analysis of Temperature Distribution Along C-D Nozzle with Kerosene as Fuel

Udhaya sankar R¹, Karthic Subramaniyan I², Nagaraj V³

¹Asst. Prof. Mechanical Dept., & Periyar Maniammai Institute of Science and Technology, Periyar Nagar, Vallam, Thanjavur, Tamil Nadu, India - 613403

^{2,3}Asst. Prof. Aerospace Dept., & Periyar Maniammai Institute of Science and Technology, Periyar Nagar, Vallam, Thanjavur, Tamil Nadu, India - 613403

Abstract – This study focused to analyze the temperature distribution along the convergent divergent(C-D) nozzle while combustion occurred with kerosene fuel. The scale down model of C-D nozzle was designed with CATIA V5 software and analyzed with CFD software (ANSYS FLUENT). The fuel used for this analysis was kerosene. Different turbulence model was considered for those analysis such as spalart almaras, K- ϵ and K- ω . In this integrated component model, the inlet flow is assumed a two-dimensional, steady, compressible, turbulent and supersonic. The physics based mathematical model of the considered flow consists of conservation of mass, momentum and energy equations subject to appropriate boundary conditions as defined by the physical problem stated above. The Mach number was chosen for different cases of analyses were 1.2, 1.5 and 2. For each case, different turbulence were engaged and solved and all the results were compared finally. The liquid rocket nozzle model were fabricated using alloy steel with considerable thickness. Using kerosene as fuel, the combustion was carried out under different air speed and injection velocity. The temperature distribution along C-D nozzle noted and compared with simulation. The validation shows good agreement.

Key Words: C-D Nozzle, Kerosene, CFD

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentation. Its scope is not limited to fluid dynamics; CFD could be applied to any process which involves transport phenomena with it. To solve an engineering problem we can make use of various methods like the analytical method, experimental methods using prototypes. The analytical method is very complicated and difficult. The experimental methods are very costly. If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again tested. This is a time-consuming as well as a cost-consuming process. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionised the field of engineering. In CFD a problem is simulated in software and the transport equations associated with the problem is mathematically solved with computer assistance. Thus we would be able to predict the results of a problem before experimentation.

Some of the earlier investigation related with our current work are: Padmanathan et al, computational analysis was

carried out for shock waves on C-D nozzle. It was proven computational that increase in the static pressure, density and the static temperature across the shock. Mach number decrease across the shock[1]. Krishna Prafulla et al, numerical studies were conducted for C-D nozzle. The results from computational analysis of flow properties had good agreement with experimental data from Naval Science and Technological Laboratory, Visakhapatnam[2]. Sibendu Soma et al, studies were carried out for Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions. The flame structure and stabilization are also noticeably influenced by orifice geometry. The flame lift-off lengths are the highest and lowest for the hydroground and conical nozzles, respectively the amount of soot produced is highest with a conical nozzle, while the amount of NO_x produced is the highest with a hydroground nozzle[3]. Seyed Ehsan Rafiee et al, energy separation techniques were tested and achieved through using of C-D nozzle. The parameters are focused on the convergence ratio of nozzle, inlet pressure and number of nozzle intakes. The effect of the convergence ratio of nozzle is investigated in the range of 1–2.85. The experimental and simulated results were compared[4]. Balabel et al, turbulent models were assessed on 2 dimensional C-D rocket nozzles. From the results, the shear-stress transport (SST) k- ω model exhibits the best overall agreement with the experimental measurements[5]. K. Pougatch et al, compressible gas-liquid phase was modelled over C-D nozzle. It reveals that virtual mass force plays a major role in accelerating/decelerating flows with a relatively low interfacial drag[6]. Wen-Ya Li et al, optimization was done for design of convergent-barrel cold spray nozzle using numerical method. Particles can achieve a relatively low velocity but a high temperature under the same gas pressure using a CB nozzle compared to a convergent-divergent (CD) nozzle[7].

2. MATERIALS AND METHODS

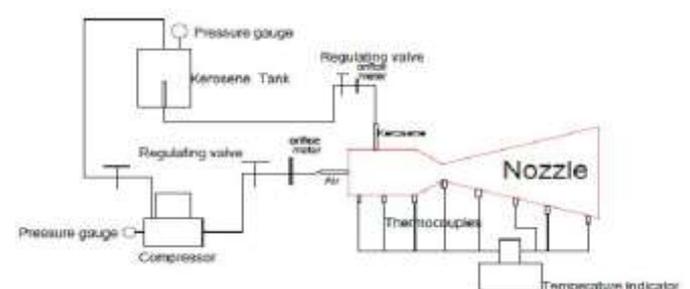


Fig.1 Experimental setup for subsonic c-d nozzle

Figure 1 shows that experimental setup for analyzing the temperature distribution of CD nozzle. First of all to inject the diesel injector and pneumatic nipple to be fitted in the nozzle vessel as per the following dimension.

Pneumatic nipple to be regulating the air and rectify the orifice meter, air will take through the compressor vessel and pressure gauges have been fixed in the compressor bar, for finding the pressure input level noted by air compressor.

The uses of for valve diesel injector to spray the diesel fuel from the diesel tank, it is also same that of pneumatic or air vessel. Diesel tank flow to be regulated by the orifice meter and regulating valve also to be maintained by them. Both of the injecting device, input value given in the pressure (bar) and output measured value taken through the temperature indicator by using the thermocouple (K type). Also figure 2 shows photographic view of entire setup of experimentation.



Fig.2 Experimental method

2.2 Nozzle



Fig.3 Nozzle fabrication

Figure 3 shows the C -D nozzle with thermocouple connections. The nozzle of the design was fully fabricated in mild steel (MS) and the overall steel will taken the weight of 180 kg. The above design doing in the job of the lathe works. Both the tapering and the boring option to be used by this steel.

The input side of inside the nozzle threaded to be bored, for injecting the device manipulation at pneumatic nipple injector. The tapering cutting taken in the both outside and inside the steel. Then to be fitted by the diesel injector, so boring the nozzle top half of the steel. It to be fixed well. For the purpose boring the nozzle ,to be replacement of diesel injector because incase any fault to be deformed in injector mean alternatively changed by the other type of the diesel injector.



Fig.4 Nozzle top view

Thermocouple also to be combined by the nozzle, to find the output value temperature, and to be fixed in the opposite of the diesel injector. the length of the thermocouple is 1m long because to avoid the causes from heating device, sometime heavy amount temperature passes through the nozzle ,that the time temperature to be heating well. by this case to avoid the failure, using the length of thermocouple assumed by 1m long.

Figure 4 shows the inlet of air and fuel intake to combustion chamber. A projecting part with an opening as at the end of a hose regulating and directing a flow of fluid. A Slang the human nose. A projecting pipe or spout from which fluid is discharged (Mechanical engineering) also called as propelling nozzle a pipe duct Gap in a jet engine or rocket direct the effluent and accelerate (furniture) a socket such as the part of a candlestick the hold the handle. A projecting pipe or spout from which fluid is discharged (Mechanical engineering) also called as propelling nozzle a pipe duct Gap in a jet engine or rocket .spout terminal discharging pipe or the like as of hose or bellows. A duct in a rocket engine in which the velocity of fluid is increase. If the nozzle is most be important in the analysis process. It's will be having convergent and divergent to position. It can measured for temperature pressure and also the velocity. When we have to apply the main functional will be getting a better to result. Due to the nozzle to measure the overall distance length and also must be nozzle process to explain as increasing the pressure at high level .The chamber is usually big enough so that any flow velocities here are negligible. The pressure here is denoted by the symbol pc. Gas flows from the chamber into the converging portion of the nozzle, past the throat, through the diverging portion and then exhausts. A simple example to get a basic feel for the behavior of the nozzle imagine performing the simple

experiment. Here we use a converging diverging nozzle to connect two air cylinders. Cylinder A contains air at high pressure, and takes the place. The CD nozzle exhausts this air into cylinder B, which takes the place of the tank. When the flow speed reaches the speed of sound. In a steady internal flow (like a nozzle) the Mach number can only reach 1 at a minimum in the cross-sectional area. When the nozzle isn't choked, the flow through it is entirely subsonic and, if you lower the back pressure a little, the flow goes faster and the flow rate increases. As you lower the back pressure further the flow speed.

2.3 Fuel tank

If the fuel tank is one of the storage devices. Fuel tank to have a some capacity of variable size, in this fuel tank at have a exhaust hole and inlet hole also to attach the air reliever to top of the point. Due to high pressure, it can be maintain the 2 bar tolerance at the stage of tank size. In this fuel tank to have a some of valve to attaching in this tank and convert the fuel and pressure.

This tank is filled with kerosene and pressure maintied by the bar is 2. One side of inlet value is taken through the compressor air and other is connected trough the diesel injector. And the tank is tightly closed by top of the layer. The purpose of given the temperature is used to enter the fluid through the injector valve. Its maintained by the injector pressor is 2 bar. Figure 5 shows that compressed fuel take which can supply the fuel to injector.



Fig.5 Fuel tank

A fuel tank is a safe container for flammable fluids. Though any storage tank for the furl may be so called, the term is typically applied to part of an engine system in which the furl is stored and propelled (fuel pump) or released (pessurized gas)into an engine. Fuel tank range in size and complexity

from the small plasti tankof a butane lighter to the multi-chambered cryogeni space shuttle external tank.

2.4 Injector



Fig.6 Injector

Figure 6 shows that injector for fuel supply .A device for injecting fuel into a combustion chamber high pressure by means of a jet of. FuelA mechanism consisting of a pump, valves, and nozzles for spraying fuel into the cylinders of an internal-combustion engine. A contrivance for injecting (e.g., water into the boiler of a steam engine or particles into an accelerator etc.),our next generation common rail system uses a hydraulically amplified diesel injector that can produce injection pressure of up to 2500 bar, says dohle. The unit features two injection units on the moving platen and a third injector on the stationary side. It has developed a specially-designed pesticide injector and the plans to showcase it on the road this year with as many as 12 field trials scheduled in them.

2.5 C-D nozzle

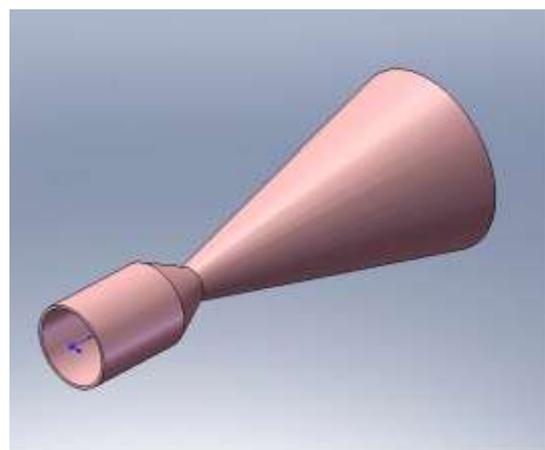


Fig.7 3-Dimensional model of C-D Nozzle

Figure 7 and 8 shows that 3 -D view of C-D nozzle and computational model of C-D nozzle. The 3-d model was generated from solid works software package. The dimension of C-D nozzle was taken from Liquid propulsion system centre (division of ISRO Thiruvananapuram]. It is a scale down model and was used for testing with air- kerosene

which has a maximum working pressure of 10 bar and therefore the secondary injection module was suitably altered in order to bypass the effects of flow separation. The secondary injection was initiated well before flow separation occurs.

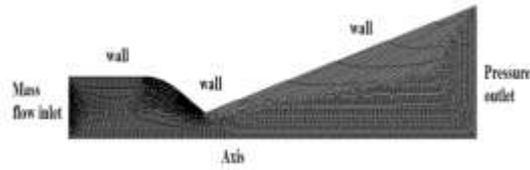


Fig.8 Computational Model of C-D nozzle

The 2- dimensional axisymmetric model was generated and meshed in Gambit pre-processor. Triangular elements and pave method of meshing was used for discretization. Totally 16616 elements and 8616 nodes were involved in discretization. Boundary types were mentioned in gambit pre-processor itself as shown in fig 3. For the purpose of decreasing solving time and cost, axi-symmetric model was given for this analysis.

A mathematical model comprises equations relating the dependent and the independent variables and the relevant parameters that describe some physical phenomenon. Typically, a mathematical model consists of differential equations that govern the behaviour of the physical system, and the associated boundary conditions. To start with fluent, it is necessary to know if the meshed geometry is correct, so is checked. To ensue with, we are to define the model, material, operating condition and boundary condition. Models are to be set in order to define if any energy equation is dealt with our study, if the flow is viscous.

3. RESULT AND DISCUSSION

This is the graph given by the following pressure(bar)

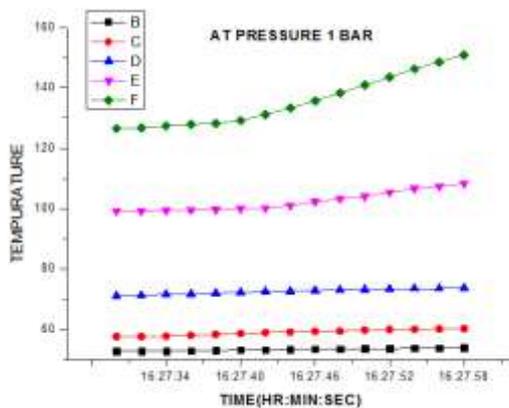


Fig.9 Temperature distribution at 1bar

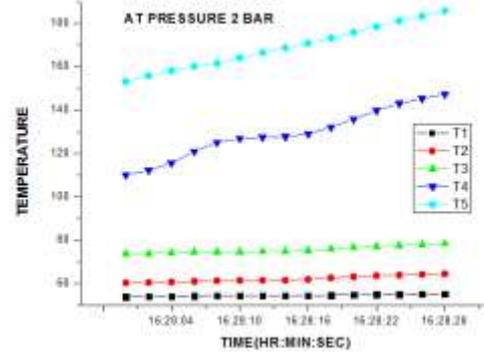


Fig.10 Temperature distribution at 2bar

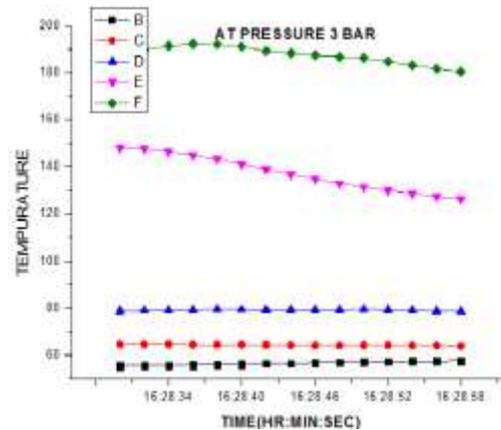


Fig.11 Temperature distribution at 3 bar

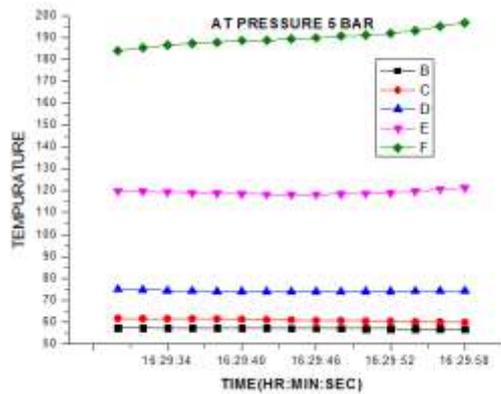


Fig.12 Temperature distribution at 4 bar

Form these plots, it has been found as temperature when pressure rises, temperature when fuel quantity increases, temperature distribution throughout the nozzle,

This is the experimental result analysis of the cd nozzle, the graph to be taken under the x-y axis at time vstemperature at the different level of bar (1,2,3,4)

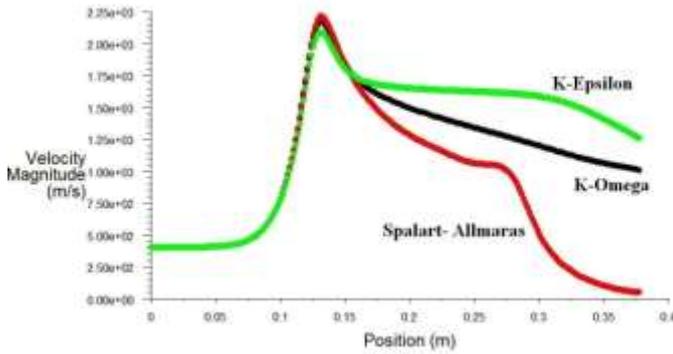


Fig.13 Velocity distribution plot at Mach no-1.2

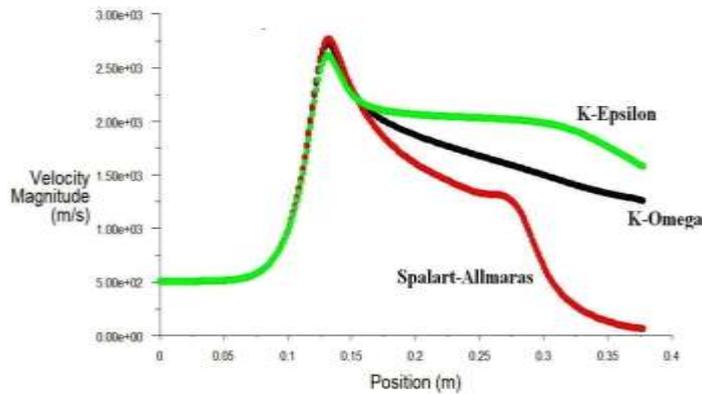


Fig.14 Velocity distribution plot at Mach no-1.5

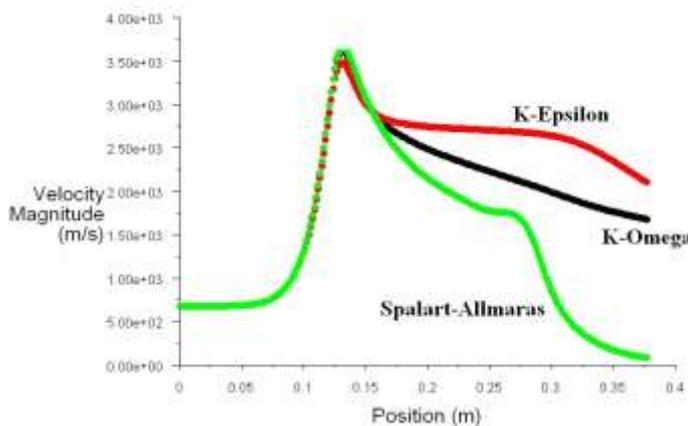


Fig.15 Velocity distribution plot at Mach no-2

Above all the plots are describing the velocity distribution of C-D nozzle for various Mach number and different turbulence model. All the distribution was quite similar to each other with different Mach number. The differences are very small. It reveals that, not much relation between Mach number and turbulence model. Variation was found between various turbulence models at same Mach number. K-epsilon and K-omega were had similar distribution except small deviation at tail of C-D nozzle. But Spalart allmaras model had immense digression form other two models. It might of large turbulence at tail section of C-D nozzle

4. CONCLUSION

Near the wall, the Mach number is decreasing for all the nozzles. This is due to the viscosity and turbulence in the fluid. Flow separation and formation of a vortex after the shock can be attributed to adverse pressure gradient following shocks. The turbulence intensity was high at tail of C-D nozzle in sparart allmaras model. But other two models had more intensity at throat. Flow was separated at near to throat (ahead of throat) in K-epsilon and K-omega models. But the flow was attached to the more length of divergent wall in spalart allmaras model. The standard K- epsilon turbulence model provided the accurate results as compared to Spalart-Allmaras model and K-omega for same set of conditions and discretization schemes. In this experimental method, we have to find out the temperature for given in the time of pressure, at that the stages temperature value of the thermocouple increases and the efficiency of the nozzle to be increases as well as pressure to be increased. So that the efficiency of this experimental studies will be very high temperature.

REFERENCES

- 1) P. Padmanathan, Dr. S. Vaidya Nathan, "Computational Analysis of Shockwave in Convergent Divergent Nozzle" International Journal of Engineering Research and Applications, Vol. 2, Issue 2, Mar-Apr 2012, pp.1597-1605
- 2) Ms.B.Krishna Prafulla, Dr. V. Chitti Babu 2 and Sri P. Govinda Rao, "Cfd Analysis of Convergent-Divergent Supersonic Nozzle" International Journal of Computational Engineering Research, Vol, 03, Issue 5.
- 3) Sibendu Soma, Anita I. Ramirez , Douglas E. Longman , Suresh K. Aggarwal, "Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions" journal homepage: www.elsevier.com/locate/fuel
- 4) Seyed Ehsan Rafiee, Masoud Rahimi, "Experimental study and three-dimensional (3D) computational fluid dynamics(CFD analysis) on the effect of the convergence ratio, pressure inlet and number of nozzle intake on vortex tube performance Validation and CFD optimization"Energy, Volume 63, 15 December2013, Pages195-204
- 5) A. Balabel, A.M. Hegab, M. Nasr, Samy M. El-Behery, "Assessment of turbulence modeling for gas flow in two-dimensional convergent divergent rocket nozzle"Applied Mathematical Modelling, Volume 35, Issue7, July2011, Pages3408-3422
- 6) K. Pougatch, M. Salcudean, E. Chan, B. Knapper, "Modelling of compressible gas-liquid flow in a convergent divergent nozzle" Chemical Engineering

Science, Volume 63, Issue 16, August 2008, Pages 4176-4188

- 7) Y. Bartosiewicz, Zine Aidoun, "CFD-Experiments Integration in the Evaluation of Six Turbulence Models for Supersonic Ejectors Modeling" CETC-Varenes, Natural Ressources Canada, ybartosi@RNCan.gc.ca
- 8) C.Mohanraj, Dr.P.K.Srividhya, "CFD Simulation of 20 kW Downdraft Gasifier"
- 9) International Journal of Current Engineering and Technology ISSN 2277 - 4106, Vol.3, No.1 (March 2013)
- 10) K.M. Pandey, Virendra Kumar, Prateek Srivastava, "CFD Analysis of Twin Jet Supersonic Flow with Fluent Software" Current Trends in Technology and Sciences Volume 1, Issu 2 , (Sept.-2012)
- 11) Biju Kuttan P, M Sajesh, "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" The International Journal Of Engineering And Science (Ijes), Volume2, Issue 2,Pages196-207,2013
- 12) Nadeem Akbar Najar, D Dandotiya, Farooq Ahmad Najar, "Comparative Analysis of K- ϵ and Spalart-Allmaras Turbulence Models for Compressible Flow through a Convergent-Divergent Nozzle" The International Journal Of Engineering And Science (IJES), Volume2, Issue8, Pages08-17, 2013
- 13) Pardhasaradhi Natta, V.Ranjith Kumar, Dr.Y.V.Hanumantha Rao, "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd)" Engineering Research and Applications (IJERA) ISSN 2248-9622 ,Vol. 2, Issue 5, September-October 2012, pp.1226-1235
- 14) D. Kornack and P. Rakic, "Cell Proliferation without Neurogenesis in Adult Primate Neocortex," Science, vol. 294, Dec. 2001, pp. 2127-2130, doi:10.1126/science.1065467.