International Journal of Research
Available at https://edupediapublications.org/journals

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue-17
December 2017

An Effective Design and CFD Analysis of Combustion Chamber in IC Engine: A Study

B.Radhika & Dr.V S Hariharan
[1]M.Tech(CAD/CAM), Dept. of Mechanical Engineering, Balaji Institute of Technology & Science, Warangal Rural, Telangana, India
[2]Principal, Dept. of Mechanical Engineering, Balaji Institute of Technology & Science, Warangal Rural, Telangana, India

Abstract:*High-temperature gases are produced via combustion ordecomposition of the propellant. The products produced is discharged via a nozzle to achieve excessive gas pace and thereby preferredthrust. Suitable design of chamber, nozzle performs the essential function for powerful utilization of propellant strength. Chamber with one aspectconvergent-divergent nozzle and the opposite side extension model changed into considered because of the version. New computational strategies are continuouslyevolved so that we can solve issues in distinctive engineering fields. One of these fields is gas turbines, wherein the project is to make gasturbinesmorenormal and to reduce emissions which are awful for the environment. One of the principal elements of a gas turbine that may beadvanced is the combustion chamber. In order to optimize the combustion chamber, both experimental and numerical methods arereferred to as for. Numerical optimization implies the need to model the most vital phenomena in combustion chambers which includeturbulent spinning go with the flow, chemical reactions, heat transfer, and so forth. In this contest, we try to design a simple but correct version, for aregular combustor of the commercialinterest, that can be examined in a quite quick time and that yields reliable effects.*

Keywords-Gas turbineEngine, Combustion chamber, Heat transfer Ansys, Computational Fluid Dynamics

I. INTRODUCTION

As the reserves of fossil fuels have decreased, alternate fuels as well as alternative power sources havebecome more and more important. The fueling of classical IC engines with biogas represents the otherpossibility of biogas utilization as the alternate renewable fuel. It is a very convenient fuel for reciprocatingengines, which is able to fulfill all future requirements concerning emission formation and engine efficiency.An optimization of performance could be achieved if the diesel engines are supplied with premixed biogas gasand is ignited by the combustion flame initiated by a diesel spray, and subsequently supplied with thecontrolled supply of diesel. Pure biogas fueled engine performance is better at very high compression ratios,which are practically difficult, and the complete engine needs to be re designed. Instead if biogas is supplied indiesel engine in dual fuel mode with the controlled supply of diesel, the existing diesel engine could beused without any modifications and with considerable reduction in pollution.

Light load operation of dual fuel engines, associated with the use of very lean gaseous fuel-air mixturesproduces relatively significant exhaust concentrations of unconverted methane and carbon monoxide, especiallywhen small pilot liquid fuel injection is involved [3, 12]. Natural gas in combination with diesel was tried andfound to be very effective in NOx reduction but engine operation can suffer from high hydrocarbons(HC) emissions and poor performance, especially at high loads [1,9]. The auto ignition of methane wasstudied experimentally to obtain ignition delay data as a function of engine cylinder pressure andtemperature by Sandia Group [2, 6]. Polasek, et al. [8] has developed an application of advanced simulationand modeling of biogas-fueled engine. Two models have been applied i.e. 0-D algorithm and CFD. The 0-Dmodel has been based on GT-Power code. The CFD model has been based on advanced MultizoneEulerianModel representing general method of finite volume. The influence of main engine parameters e.g. excess air,spark timing, compression ratio, on NOx formation and engine efficiency has been investigated and reported.Experimental investigation on a LPG- diesel dual fuel engine by Poonia, et.al, revealed that at lowloads, the brake thermal efficiency is always lower than diesel values but is better at high loads. Also,at low outputs increasing the pilot quantity and intake temperature improves the thermal efficiency. TheHC and CO emissions were found to increase in the dual fuel mode. The objective of the present work was tocarry out the CFD analysis to analyze the combustion and emissions for Biogas-Diesel combination in dual fuelengine by (1) varying the compression ratio (2) by varying the biogas substitution over a wide range and to (3)Experimentally verify the results.

II. RELATED WORKS

An I.C Engine is one of the best available reliable sources of energy in the field of agriculture. Majorissue arises on performance of diesel engine are enhanced by proper design of combustion chamber. Flow andcombustion chemistry which effect swirl induced by re-entrant piston crown on pollution emission from a singlecylinder diesel engine. For more efficient in combustion, less emission and soot, High Carbon formation isrequired. It is observed that from the literature several types studies and methods that have been reported in toincrease the performance of engine such as injection pressure, injection timing, exhaust gas recirculation, swirlratio, multi injection spray angle, nozzle diameter etc,.

Inlet: A gas turbine can have one or several inlets, based ontheir design and usage. Inlets are used to send fuel and airinto the gas turbine. The main inlet in front of the gasturbine is used to suck air in; while there are several othersmall inlets existing further downstream in order to injectfuel.

Compressor: Compressors are used to increase the pressureof the inlet air, in order to increase the efficiency of theturbine. The effect of compressor, as well as other parts, canbe described by using Brayton cycle, as shown in theFigureit will raise pressure from point 1 to point 2. From thediagram one can expect that output work will rise with theraise of pressure in the point 2. On the other hand pressure atpoint 2 is limited by several parameters such as materialconstraints, temperature raise and etc.

Combustor: Here, fuel is mixed with the air and then burns.This reaction results in increasing temperature and volume.Volumetric expansion can drive the rotor blades of a turbineor a turbojet to produce work or thrust. This is an isobaricprocess.

Turbine: Its job is to drive the compressor shaft and, in thecase of a stationary gas turbine, to provide useful mechanicalwork to drive for example an electrical generator. In idealcycle, this process is isentropic.

Outlet: This section is designed based on gas turbine usage;for stationary gas turbinethe outlet is a low speed exhaust, which will guidecombustion products out of system, either to theenvironment or to other cycles. For the turbofan gas turbinethe outlet is a jet nozzle, which will increase velocity to produce thrust.

Combustion is a chemical process that burning fuel. Gasturbine engine use internal combustion system to generatethrust. It is all depend on the burning of fuel to producepower.The original substance is called the fuel, and thesource of oxygen is called the oxidizer. The fuel can be asolid, liquid, or gas, although for airplane propulsion the fuelis usually a liquid. The oxidizer, likewise, could be a solid,liquid, or gas, but is usually a gas (air) for airplanes.During combustion, new chemical substances are createdfrom the fuel and the oxidizer. These substances are calledexhaust. Most of the exhaust comes from chemicalcombinations of the fuel and oxygen. When a fuel bums, theexhaust includes water (hydrogen and oxygen) and carbondioxide (carbon and oxygen). The exhaust can also includechemical combinations from the oxidizer alone. If the fuel isburned in air, which contains 21% oxygen and 78%nitrogen, the exhaust can also include nitrous oxides (NOX,nitrogen and oxygen). The temperature of the exhaust is highbecause of the heat that is transferred to the exhaust duringcombustion. Because of the high temperatures, exhaustusually occurs as a gas, but there can be liquid or solidexhaust products as well. Soot, for example, is a form ofsolid exhaust that occurs in some combustion processes.

During the combustion process, as the fuel and oxidizer areturned into exhaust products, heat is generated. Interestingly,some source of heat is also necessary to start combustion.Heat is both required to start combustion and is itself aproduct of combustion. To summarize, for combustion tooccur three things must be present, a fuel to be burned, asource of oxygen, and a source of heat. As a result ofcombustion, exhausts are created and heat is released. Thecombustion process can be controlled or stopped bycontrolling the amount of the fuel available, the amount ofoxygen available, or the source of heat.

III. COMBUSTION CHAMBER

The combustion chamber is the place where two majorevents take place; at the inlet fuel will mix completely, or toa sufficient degree, with air. In some combustors fuel mixeswith air before combustors, however, in order to achieve asmooth burning, air and fuel should be mixed beforeburning. There are number of facts that make this part of gasturbine important. In order to make this clear, we willaddress problems in a poorly designed combustion chamber.There are several problems that can occur:

1) Poor mixing: When fuel is not mixed enough with air, itcan burn incompletely which results in increased levelsof CO, soot, NOx and unburned hydrocarbons (UHC).

2) Uneven combustion: This happens when temperature ofa section goes high but the neighbouring sections arecolder, thus this can result in extra thermal stresses.

Thermal stresses may in time lead to material fatigueand failure.

3) Environment: incompletely burned gases or unburnedhydrocarbons (UHC) can poison the environment.UHC, NOx and soot are important factors for eachburning device. The design should lower them as muchas possible.

4) Economy: With increasing price of oil, it is importantthat gas turbines have high efficiency and therefore lowfuel consumption. One of the most important parts, inorder to achieve high efficiency, is the combustionchamber.

Above factors shows the importance of combustionchambers in gas turbine. Geometry SimplificationsThe simplifications that were done are the following:The most common combustors have no symmetry in thedomain usually coming from the locations of the burnerinlets. The first simplification was to omit these inlets so thegeometry becomes symmetric. This implied that only 45°(1/8)th of the full geometry were modelled, shown in Figure.This is one section that has been modelled.

1)We assume one inlet for the fuel and the air. The mostcommon combustors have separate inlets for fuel and air.Both the fuel and the air are assumed to be perfectlymixed at the inlet.

2)NOx formation was neglected and assumed that the fuelwill be burn completely.

Modelled geometry: The simplified geometry consists of an inlet, a guide vaneand bottom faces are set to walls, while the side faces areaxial symmetric shown in Figure.1. There is a secondaryinlet in the beginning of the iteration process the mass flowrate is set to zero. The full geometry is shown in whichconsists of 8 sectors.

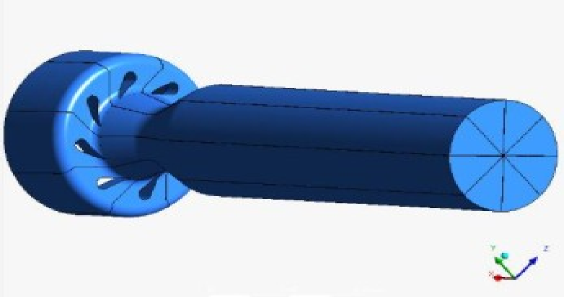


Fig.1 Full modelled geometry

IV. PERFORMANCE ANALYSIS

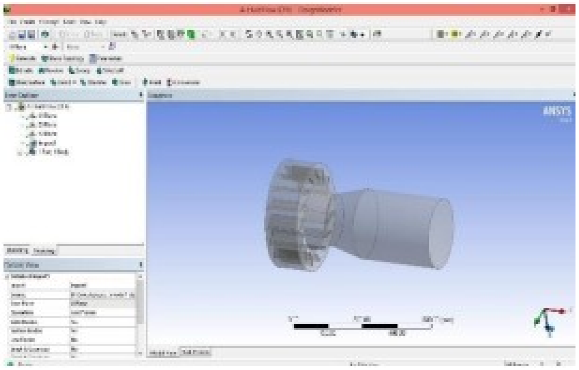


Fig.2Importing of Combustion chamber to Ansysand inlet

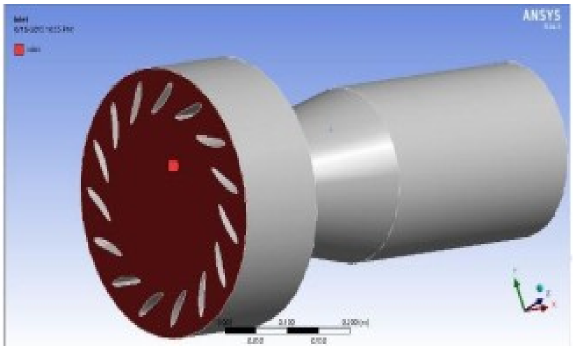


Fig.3Outlet of the Combustion chamber

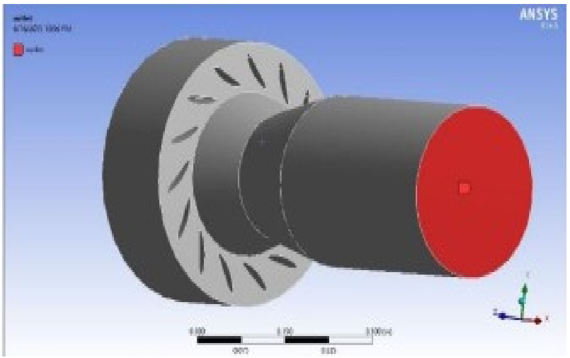


Fig.4Wall of the Combustion chamber

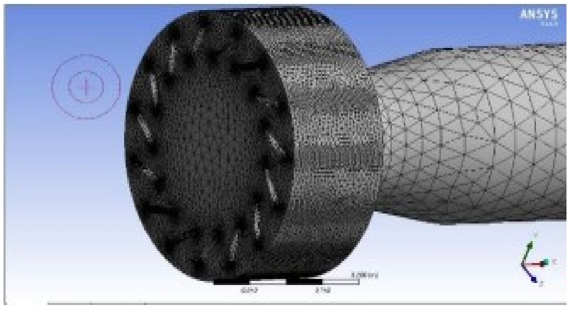
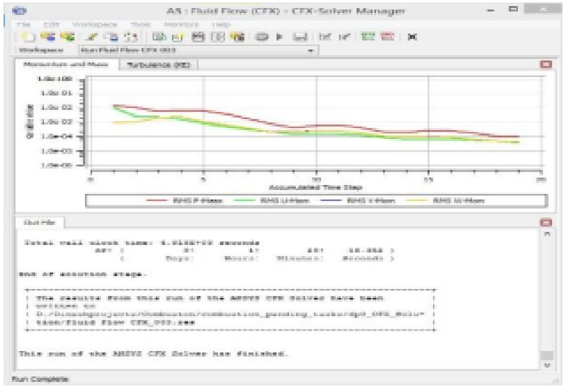


Fig.5End of the Solution after the Problem Set Up



```
End of execution stage.

This run of the ANSYS CFX Solver has finished.
```

Fig.6CFD Results of the Combustion Chamber

ANSYS CFX v14.5 was used as solver. The numericalsettings for the solver are described below.The problem is solved as a steady state flow problem,consistent with the RANS turbulence modelling used, whichmeans that relatively large time steps are used in order toachieve a converged solution as quickly as possible. In spiteof the turbulence model the flame itself is slightly unsteady,but the oscillations are negligible.Total energy including viscous work terms" model is used,which means that the total energy models the transport ofenthalpy including the kinetic energy effects. This modelshould be used where there is change in density or the Machnumber exceeds 0.2; in both of these cases kinetic energyeffects are significant. In ANSYS CFX, when one choosestotal energy the fluid is modelled as compressible, regardlessof the original fluid condition, i.e. gases with Mach numberless than 0.2. One should know that incompressible fluiddoes not exists in reality but for the gases with Mach numberless than 0.2 the compressible effects are in generalnegligible.

Turbulence: For the turbulence both the k-ε SST and the k-ε turbulencemodels are used. The k-ε model is one of the most commonturbulence models. It is a two equation model that includestwo extra transport equations to represent the turbulentproperties of the flow. This allows the model to account forhistory effects like convection and diffusion of turbulentenergy. The k-ε model has a good prediction in the freestream, but near the walls, the prediction is poor sinceadverse pressure gradient is presented. For wall treatmentscalable wall function is used. Standard wall functions arebased on the assumption that the first grid point off the wall(or the first integration point) is located in the universal lawof-the-wall or logarithmic region.We choose further on 500K also not the lowest meshdensity. There exists a circulation zone in front of ignitioninlet; this region is one of the most important regions inorder to mix flow and better flame.

V. CONCLUSION

The end of the grid-have a look at is that the mesh-length thatis used for the 500K case is enough, or in different phrases, theeffects are grid-unbiased. These conclusions are based totally onconstant-nation simulations and were no longer tested on transientsimulations because of obstacles of time inside the challenge. This isadditionally crucial to test the destiny work. The 500k meshlength could mean that the variety of cells for a full 360°the model would be approximately 16M cells. Because thismodel confirmed stable convergence and additionally it predicts flowsubject higher than the opposite cases.A new functional expression for the sort of version parameter,which represents extinction of the flame brush through turbulenteddies turned into proposed based on laminar flames. Distributionof air, waft recirculation, jet penetration and mixing arecarried out in all the zones of the combustion chamber.

REFERENCES

[1] Riccio G., Marini A., Martelli F., “Numericalinvestigation of gas turbine combustion chamberhydrogen fired”, 19th ISABE Conference, ISABE 2009-1112, Montréal, Canada.

[2] FLUENT 6.3 User’s Guide, 2006

[3] KARIM, G.A., LIU, T., and JONES, W., “Exhaust Emissions from Dual fuel Engines at Light Loads”. Presented at SAE. Paper SAE932822(1993).

[4] KONG, S.C., HAN, Z.W., and REITZ,R.D., “The Development and Application of a Diesel Ignition and Combustion Model forMultidimensional Engine simulation” SAE 950278, 1995.

[5] KONG, S.C., HAN, Z.W., and REITZ, R.D., “The Development and Application of a Diesel Ignition and Combustion Model forMultidimensional Engine Simulations,” SAE 950278, 1995.

[6] NABER, J.D., SIEBERS, D.L., CATON, J.A., WESTBROOK C.K., and DI JJULIO, S.S.,“Natural Gas Auto Ignition under DieselCondition: Experiments and Chemical Kinetics Modeling”. Presented at SAE. Paper SAE 942034(1994).

[7] PETERS N., “Turbulent Combustion”. Cambridge University Press UK 2000.

[8] POLASEK, MILOS, MACEK, JAN, TAKATS, MICHAL, “Biogas Fuelled Engine- Properties Of Working Cycle and EmissionPotentials” Report No. F02V102 Czech Technical University in Prague, Czech Republic, 2002.

[9] SINGH, S., “The effect of Fuel Injection Timing and Pilot Quantity on the Pollutant Emissions From a Pilot ignited Natural GasEngine”, Master’s Thesis The University Of Alabama, 2002.

[10] SING, S. LIANG, L., SONG-CHARNG K., and RIETZ R.D., “Development of a Flame Propagation Model for Dual – Fuel PartiallyPremixed Compression Ignition Engines” SAE May 2004.

[11] TAN, Z., AND REITZ, R.D., “Modeling in Spark – Ignition Engines Using a Level Set Method”. Presented at SAE. Paper SAE 2003– 01- 0722.

[12] YASHUHIRO, KOU, T., YUKI, SHIGEKI, N., RYOJI, K., and TAKESHI, S., “Controlling Combustion and exhaust Emissions in aDirect – injection Diesel Engine Dual Fueled with natural Gas”. Presented at SAE. Paper SAE 952436 (1995).