A Review on use of Computational Fluid Dynamics in Gas Turbine Combustor Analysis and its Scope

H. A. Bhimgade[1]**, S. K. Bhele**[2]

[1, 2]Department of Mechanical Engineering, Kavikulguru Institute of Technology and Science, Ramtek, India

Abstract: ***Computational fluid dynamics (CFD) modeling is now widely applied as combustion optimization tool. The steady increase in computer power over recent years has enabled combustion engineers to model reacting multi-phase flows in a realistic geometry with good mesh resolution. As a result, the number of applications of CFD to industries and power generation are also growing rapidly and increasing in sophistication. This paper reviews some of the recent applications of the CFD in gas turbine combustor used in the power generation, aero-engines and combustion industries. The aim is to illustrate what can be done and also to identify trends and those areas where further work is needed.***

Keywords: CFD, Emissions, Gas Turbine Combustion chamber, Temperature, Turbulence.

1. Introduction

The gas turbine is a power plant, which produces a great amount of energy for its size and weight. The gas turbine has found increasing service in the past 40 years in the power industry both among utilities and merchant plants as well as the petrochemical industry, and utilities throughout the world. Its compactness, low weight, and multiple fuel application make it a natural power plant for offshore platforms. Today there are gas turbines, which run on natural gas, diesel fuel, naphtha, methane, crude, low-Btu gases, vaporized fuel oils, and biomass gases. Basically gas turbines are classified as stationary and non-stationary type, open and closed type, and sub divided accordingly. Major components of gas turbine are compressor, combustion chamber and turbine, an overall efficiency depends on the performance of these components. The last 20 years has seen a large growth in Gas Turbine Technology. The growth is spearheaded by the growth of materials technology, new coatings and new cooling schemes. This, with the conjunction of increase in compressor pressure ratio, has increased the gas turbine thermal efficiency from about 15 % to over 45 % [1]. The major losses occur in the combustion chamber and it is more responsible for the efficiency of gas turbine. Hence, in this paper the study is mainly focused on the gas turbine combustor performance and tried to evaluate the scope of the CFD use to increase the combustor performance.

1.1 Gas Turbine Combustor Performance

The aerospace engines have been the leaders in the most of the technology in the gas turbine. The design criteria for these engines was high reliability, high performance, with many starts and flexible operation throughout the flight envelop. The engine life of about 3500 hours between major overhauls was considered good. The aerospace engine performance has always been rated primarily on its thrust/weight ratio is achieved by the development of high aspect ratio blades in the compressor as well as optimizing the pressure ratio and firing temperature of the turbine for maximum work output per unit flow. The industrial Gas turbine has always emphasized long life and this conservative approach has resulted in the industrial gas turbine in many aspects giving up high performance for rugged operation. The industrial gas turbine has been conservative in the pressure ratio and the firing temperatures. This has all changed in the last 20 years; spurred on by the introduction of the “Aero-Derivative Gas Turbine” the industrial gas turbine has dramatically improved its performance in all operational aspects. As both the pressure ratio and firing temperature growths are necessary to achieving the optimum thermal efficiency. Increase in the pressure ratio increases the gas turbine thermal efficiency when accompanied with the increase in turbine firing order. Now a day’s mostly the study is focused on the combustor performance. Since past 40 years, combustor technology has developed gradually and continuously, rather than through dramatic change, which is why most of the aero-engine combustors now in service tend to resemble each other in size, shape, and general appearance. This close family resemblance stems from the fact that the basic geometry of a combustor is dictated largely by the need for its length and frontal area to remain within the limits set by other engine components, by the necessity for a diffuser to minimize pressure loss, and by the requirement of a liner (flame tube) to provide stable operation over a wide range of air/fuel ratios. During the past half century, combustion pressures have risen from 5 to 50 atmospheres, inlet air temperatures from 450 to 900 K, and outlet temperatures from 1100 to 1850 K. Despite the continually increasing severity of operating conditions, which are greatly exacerbated by the concomitant increases in compressor outlet velocity, today’s combustors exhibit close to 100% combustion efficiency over their normal operating range, including idling, and demonstrate substantial reductions in pollutant emissions. Furthermore, the life expectancy of aero-engine liners has risen from just a few hundred hours to many tens of thousands of hours. Although many formidable problems have been overcome, the challenge of ingenuity in design still remains. New concepts and technology are needed to further reduce pollutant emissions and to respond to the growing requirement of many industrial engines for multi-fuel capability. Gas turbines are “omnivorous” machines, capable of operating efficiently on a wide variety of cheap fuels, solid, liquid, and gaseous, with the exception of aircraft engines. Today, the ever-rising cost of petroleum fuel is prompting research into developing alternative liquid fuels and this is posing new combustor design challenges.

Another problem of increasing importance is that of acoustic resonance, which occurs when combustion instabilities become coupled with the acoustics of the combustor. This problem could be crucial to the future development of lean premixed combustors.

Gas turbine technology has developed rapidly and continuously over last 20 years and its use in various field has been increased. Despite the continued advances in the gas turbine combustor technology, the challenge which a combustion engineer faces today to ingenuity in design is greater than ever before. Interest in the environment by scientists and the general public has evolved from the gradual deterioration in our air quality and the discovery of the Antarctic ozone hole, in addition to hot summers, greenhouse effect and increased smog in cities. In addition, the energy conservation and higher conservation efficiencies remain on the priority list. In this paper the CFD application and its scope, is mainly focused on gas turbine combustor (generally can or tubular, annular and tuboannular type of combustor used in gas turbines for higher efficiency). Because such a type of combustors, now-a-days are being used widely in the fields like Jet engines, Turboprop engines, Aero derivative gas turbines, Auxiliary power units : Industrial gas turbines for power generation, Industrial gas turbines for mechanical drive, Compressed air energy storage, Micro turbines also known as: Turbo alternators, Turbo generator External combustion, Gas turbines in surface vehicles : Passenger road vehicles (cars, bikes, and buses), Racing cars, Buses, Motorcycles, Trains, Marine applications : Naval, Civilian maritime etc. and hence the fuel consumptions all over the world increased rapidly. Below figure shows the conventional gas turbine combustor where the fuel burns with different zones.

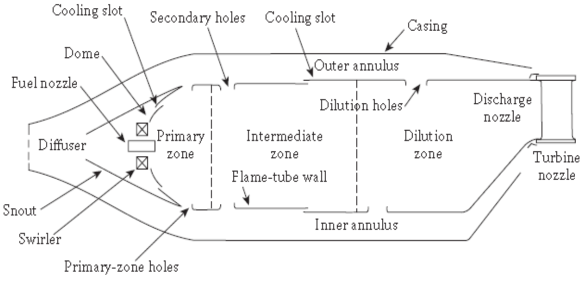

Figure 1: Main components of conventional gas turbine combustor

A Primary zone (PZ) or recirculation zone: It is the primary zone fuel/air ratio that, more than any other single factor governs the nature of the combustion process and the overall chamber characteristics. The advantage of a stoichiometric primary zone include a high volumetric heat release rate and relatively low rate of carbon formation, in consequence, the primary zone can be made small, and the combustion process is characterized by freedom from coke deposition and exhaust smoke. Its main disadvantages are a high rate of heat transfer to the liner walls and high emission of nitric oxides. A secondary zone, intermediate zone (IZ) or burning zone (with a recirculation zone which extends to the dilution region): The main function of the intermediate zone is to provide conditions that are conducive to recombination and thus to the elimination of dissociated products from the gases entering the dilution zone. Temperature plays a crucial role in this process: if it is too low, then recombination will proceed very slowly and will not be completed in the time available. If the temperature is too high, then the gases entering the dilution zone will contain dissociated species in equilibrium concentration. An intermediate zone temperature of around 1800 K offers a good compromise, although this value may have to be exceeded for high values of turbine inlet temperature to secure adequate air for the dilution zone. A dilution zone (DZ): Perhaps the most important and, at the same time , most difficult problem in the design and the development of gas turbine combustion chamber is that of achieving a satisfactory and consistent distribution of temperature efflux gases discharging in to the turbine. Thus, it is now generally accepted that satisfactory temperature profile is dependent upon both adequate penetration of the dilution jets and use of the correct number of jets to form sufficient localized mixing regions; however, the manner in which a given total dilution hole area is translated into a particular number and size of holes [2].

Combustion of fuel produces emission like CO_2, H_2O, hydrocarbons and soot particles, also several nitrogenous compounds, including NO_x (commonly grouped as NO, NO_2, N_2O), NO_3, N_2O_5, and HNO_3, oxides of sulfur (SO_x) and other sulfur compounds. The total unburned hydrocarbons (THC) and CO in the exhaust plume represent not only combustion inefficiency but also play an important role in stratospheric HO_x (OH and H_2O) chemistry. Nitrous oxides (N_2O) and nitrogen oxides (NO_2) diffusing upward from exhaust breaks into destructive nitric oxide (NO) in the atmosphere and accounts for about 50% of the ozone depletion and acid rain [3]. Aircraft and stationary gas turbine power plants contribute to the problem of ozone depletion and acid rain. Complete combustion of fuel in the combustor assures the no exhaust emissions and maximum overall efficiency of gas turbine (ranges from 35% - 65% depends on type of cycle used). Operating characteristics of the combustor, such as fuel efficiency, level of emissions and transient response (the response to changing conditions such as a fuel flow and air speed) inlet air temperature, inlet air pressure, equivalence ratio etc. determines the combustion process The desired performance requirements of combustors, in terms of higher engine thrust/weight ratio and lower specific fuel consumption call for higher turbine inlet temperatures and closer fit to the design temperature profile at inlet to the turbine. In addition, greater reliability, increased durability, lower manufacturing, development and maintenance costs, fuel economy, fuel flexibility and lower levels of pollutants emission adds importance. In order to reduce these problems now-a-days, advanced combustion diagnostics, using computational tools providing significant advances in combustor design and development. It has been found that the computational analyses are more advantageous over the experimental methods thus it includes the use of computers to analyze systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions based on numerical approach. Computational tool especially CFD is being used. However, there have been numerous computational techniques on this. In this field, there are many computational commercial (FLUENT, KIVA, VECTICS, STAR-CD and so on) or personal codes using finite volume, finite element methods C++ or FORTRAN. Some of the illustrations have been provided below which reveals that the application of CFD helps in rapid design and development of the gas turbine combustor as well as the scope of using CFD is also discussed.

B. Zamuner et al [4] carried out a study of numerical simulation of reactive flow in a tubular gas turbine combustor with detailed kinetic effects. In this study, the kinetic effects were incorporated in the turbulent CFD simulations of a methane/air tubular chamber. Main features of mean flow were measured with a classical numerical code DIAMANT which solves the Reynolds Navier Stokes (RANS) equations with an algebraic fast chemistry combustion model. Turbulent combustion model, called Probabilistic Eulerian-Lagrangian (PEUL+) model, which consists solving the joint probability density function (PDF) transport equation by a Monte Carlo technique was coupled with the RANS solver in order to account for detailed kinetic effects. In addition to this, to avoid the CPU expensive computation, chemical source terms were evaluated by tabulation method Intrinsic Low Dimensional Manifold (ILDM) which keeps the complex behavior of a detailed kinetic system, while describing its dynamics with only two or three progress variables. It was found that combustion took place approximately along the mean stiochiometric line and the temperature increased rapidly up to about 1750 K in the center of recirculation zone when fuel particles injected from the fuel inlet. It was concluded that, Flame structures, combustion regimes and pollutants emission are often difficult to predict in aircraft combustors because they are closely related to the turbulent nature of the flow and requires a complete iterative procedure to improve the predicted values. Sierra, J. Kubiak [5] et al carried out work upon the combustion chamber was part of a 70MW gas turbine used in an operating combined cycle power station. Numerical computation was applied to investigate the temperature front field in a gas turbine combustion chamber. In order to solve the turbulence the Reynolds Averaged Navier–Stokes (RANS) assumption were considered and the combustion was solved using the partially premixed combustion model (PPCM), which considers a partially premixed mixture before performing the combustion. The range of pressure imbalance of primary air applied in the refined model was adopted and this effect is interpreted as a flame expansion process caused by inlet air pressure variations. The temperature increments in the pipes were not linear with the pressure variations. The range of pressure variations for which the flame expansion effect was observed correspond to pressure fluctuations that can be registered in practice (around $3x10^3$ Pa). Results obtained by Sierra, of temperature variation at front of combustor with pressure variation are shown below.

Table 1: Variation conditions of inlet air pressure simplified and refined models

Condition	P_{airin}	P_{prim} *(Refined model)*
1	1	1
2	0.56982	0.994
3	0.08925	0.988
4	0.03639	0.982
5	0.00892	0.982
6	0.00364	0.97
7	8.92E-4	0.964
8	3.63E-4	
9	8.92E-5	

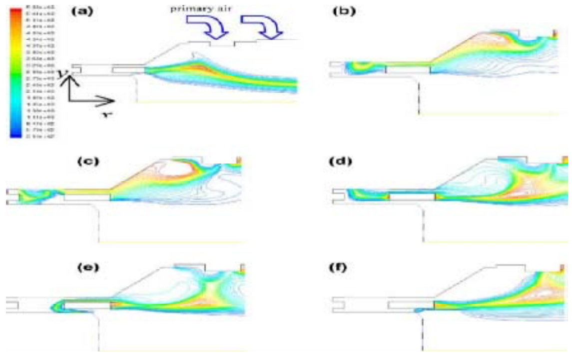

Figure 2: Contours of temperature in combustion chamber, as a function of P_{prim} pressure imbalance. (a) Condition 1 (see Table 1). (b) Condition 2 (see Table 1). (c) Condition 3 (see Table 1). (d) Condition 4 (see Table 1). (e) Condition 5 (see Table 1). (f) Condition 6 (see Table 1).

Fureby, Grinstein et al. [6] carried out an experimental and computational study of a multi-swirl gas turbine combustor. The computational approach pursued was large eddy simulation (LES), which provides a compromise between accuracy and cost. LES attempts to capture the dynamics and evolution of the large-scale flow, at the same time as it allows for inclusion of realistic flow and chemistry parameters. The flow was affected by the exothermicity through the volumetric expansion, increased molecular viscosity and baroclinic torque, resulting in flow acceleration downstream of the flame, and the development of wall jets. Di Martin, Cinque et al. [7] studied a reactive CFD analysis of complete annular gas turbine combustor module for aero engines application. All the complex combustor features like cooling holes, primary and dilution holes, swirler, fuel injector etc. were considered and fully coupled into the CFD calculation. The internal combustor flow field of mixture were studied and observed that effusion holes, which were very small in diameter and very large in number was not meshed but the effect of drilled liners was properly modeled by means of source terms and well tested correlations that link holes mass flow rate with the pressure drop across the liners. Amin Akbari et al [8] carried out an experimental and computational analysis of methane and hydrogen mixing in a model premixer and observed that, for fixed momentum fixed ratio, the penetration dispersion of both methane and hydrogen were similar (based on measurements, CFD, and empirical correlations), indicated that turbulent mixing was dominating the jet behavior in the region studied. Massimo Masi, Gobbato et al [9] studied the numerical and experimental analysis of the temperature distribution in a hydrogen fuelled combustor for a 10 MW gas turbine as part of a ministerial project coordinated by ENEL Ricerca (ENEL is the former national power company). The basic CFD approach was to predict the temperature field inside the combustor. Liner wall temperatures and turbine inlet temperatures measured during full scale full pressure experimental tests are employed to validate the numerical results. The computed temperature profiles showed acceptable matching with measurements in the liner wall and cost CFD model presented was able to capture the mean temperature field within the combustor, which was the fundamental requirement for the planned thermoacoustic characterization of the analyzed combustor. Recently Daero

Joung and Kang [10] designed and developed a small size gas turbine combustor of a reverse flow, semi-silo type for power generation operated in a lean premixed mode to achieve stable combustion. In this experiment, combustor with multi swirler (main swirler and pilot swirler at inlet of burner), annular nozzle having, pilot fuel and main fuel injector is simulated. The premixed coherent flame model (PCFM) is applied for partially premixed methane/air with an imposed downstream flame area density (FAD) to avoid flashback and incomplete combustion. The combustion efficiency reached about 99.9% with higher inlet temperature.

2. Discussion

Various literatures on CFD application in gas turbine combustor has been reviewed briefly and it is noticed that CFD results shown the good agreement with the experimental results. Particularly, use of CFD in combustion processes (as it is very complex phenomenon to solve numerically and experimentally) in the gas turbine combustor, has increased. Recently, Joung and Kang showed the combustion efficiency reached up to 99.9% which is higher than previous experimental results. Generally combustion in combustor is of diffusion (non-premixed) type or premixed type where reacting mixture like fuel and air, fuel mixes with hot incoming air and reaction of mixture releases heat which is used for power generation, high thrust etc. depend on utilization. Mostly the combustion efficiency is affected by the turbulence of air at the inlet of combustor as turbulence assures the proper mixing of fuel with the air, spray penetration as fuel injected in the combustor with fuel injector unlike petrol engines, higher temperature and pressure inside combustor, air/fuel ratio, liner temperature etc. Thus the study and analysis of combustion, experimentally becomes critical and consumes more time. In computational analyses governing equations (numerical co-relations) are solved by computer. The basic governing equations are mass conservation, momentum conservation, energy conservation and species conservation in case of reacting mixture (species of fuel which is comprise of many hydrogen and carbon atoms which again produces different species after reacting with the oxygen like nitrogen oxides, sulfur oxides etc.) The CFD is known as an indispensable tool and now it is capable of solving unsteady, reacting flows accurately with powerful computer. Recently, Ezhil Kumar [10] showed governing equations used in combustion modeling for unsteady, reacting flows assuming single step chemical reaction with negligible radiation which are shown below.

Continuity equation:-

$\partial\rho/\partial t + \partial\rho u_i/\partial x_i = 0$ (1)

Where, $\partial\rho/\partial t$ = rate of change in density
$\partial\rho u_i/\partial x_i$ = rate of mass efflux

Momentum equation:-

$\partial(\rho u_i)/\partial t + \partial(\rho u_i u_j)/\partial x_j = -dP/\partial x_i + \partial/\partial x_j\,[\mu\,(\partial u_i/\partial x_j + \partial\rho u_j/\partial x_i)]$ (2)

Where, $\partial(\rho u_i)/\partial t$ = unsteady term
$\partial(\rho u_i u_j)/\partial x_j$ = convective term
$-dP/\partial x_i$ = pressure term
$\partial/\partial x_j\,[\mu\,(\partial u_i/\partial x_j + \partial\rho u_j/\partial x_i)]$ = viscous term

Energy equation:-
$\partial(\rho T)/\partial t + \partial(\rho u_j T)/\partial x_i = \partial/\partial x_i\,(\rho D\,\partial T/\partial x_i) + \dot{R}_T$ (3)

Where, $\partial(\rho T)/\partial t)$ = unsteady term
$\partial(\rho u_j T)/\partial x_i$ = convective term
$\partial/\partial x_i\,(\rho D\,\partial T/\partial x_i)$ = diffusion term
$\dot{R}_T = -1/C_p \sum_{k=1}^{N} \Delta h_{f,k} \dot{R}_k$
$\dot{R}_k$ = source term = heat release due to combustion

Species conservation equation:-

$\partial(\rho Y_k)/\partial t + \partial(\rho u_j Y_k)/\partial x_i = \partial/\partial x_i\,(\rho D_k\,(\partial Y_k/\partial x_i) + \dot{R}_k$ (4)

Where, $\partial(\rho Y_k/\partial t)$ = rate of change of mass
$\partial(\rho u_j Y_k)/\partial x_i$ = convective term
$\partial/\partial x_i\,(\rho D_k\,(\partial Y_k/\partial x_i)$ = diffusion term
D_k = Diffusion coefficient of species k
$\dot{R}_k$ = mean reaction rate of the kth species

The above equations show the real approach to reach the real solution and hence there is ample scope for applied research on modeling techniques, ignition, fuel injection, fluid dynamic and heat transfer processes in the gas turbine combustor.

3. Future scope

Design of gas turbine combustor improving steadily as recently Joung shown that small size combustor and near about complete combustion of fuel (natural gas). Due to the scarcity of conventional fuels and concerning environmental pollution, the combustion engineers and scientists are working on the combustion of biofuels. Liquid biofuels are being researched mainly to replace conventional liquid fuels (diesel and petrol). A recently popularized classification for liquid biofuels includes “First-Generation” and “Second Generation” biofuels. The primary distinction between them is in the feedstock used. Research work is in progress for the “third-generation of biofuels” (third generation biofuels like biofuel from algae, bioethanol, butanol, biodiesel). Computational analyses especially CFD (CFX, FLUENT tools provides reach reaction library) has the greater scope to analyze combustor using biofuels as an alternate fuel in the gas turbine.

4. Conclusion

In many practical combustion applications like gas turbine and diesel engine, the combustion takes place in turbulent flow field. Therefore it is important to model the effects of turbulence and mixing interactions including all related processes either physical or chemical. In the present the emphasis is on how the turbulence leads to increased mixing in order to be used to compensate for the inaccurate prediction for the chemical reaction rate. However this has to be treated numerically and physically. Both ways are referring to the incomplete mixing process that may lead to ignite the fuel vapour before the auto-ignition delay time or out of the main reaction zone. Physically, the mixing process tends to speed up the overall reaction rate by stretching and wrinkling of the preheating zone. In addition the simulation of turbulent spray combustion remains quite a hard task because many problems may occur due to strong coupling that exists between predicted vapour mass fraction and the chemical reaction. Recently, advanced fuel spray and combustion modeling like Combined Combustion Modeling (CCM) and Conditional Moment Closure (CMC) model [11], a computational model provides real approach to solution and there is ample scope to study the different combustor geometries with this two combustion models. The study can be carry out to recognize the various combustor parameters like temperature front field, reacting fluid flows, turbulence inside combustor, influence of inlet swirl air, combustion pressure on important combustion and emission characteristics. The experimental results and the semi empirical correlations for calculating CO, UHC, NO_x, exhaust gases temperature and inner liner wall temperature as a function of different operating parameters are useful for design and further development of design, of gas turbine combustor is possible. Even with existing physical models, CFD can offer cost-effective solutions for many complex systems of interest to the power generation, aero-engines and process industries.

References

[1] Meherwan P. Boyce, Gas turbine engineering hand book, Gulf professional publishing, second edition, 2002.

[2] Arthur H. Lefebvre and Dilip R. Ballal, Gas turbine combustion (Alternative fuels and emissions), Taylor and Francis group, Third edition, 2010.

[3] P. S. Nigam, Anoop Sing, “Production of liquid biofuels from renewable sources,” Progress in energy and Combustion science, vol. 37, pp 52-68, 2011.

[4] B. Zamuner, B. Bourasseau, C. Berat, H. Niemann, “Gas Turbine Engine Combustion, Emissions and Alternative Fuels,” Paper presented at the RTO AVT Symposium held in Lisbon, Portugal, 12-16 October 1998, and published in RTO MP-14.

[5] F.Z. Sierra., J. Kubiak, G. Gonza´lez, G. Urquiza, “Prediction of temperature front in a gas turbine combustion chamber,” Applied thermal engineering, vol. 25, issues 8-9, pp 1127-1140, 2005.

[6] C. Fureby, F.F. Grinstein, G. Li , E.J. Gutmark, An experimental and computational study of a multi-swirl gas turbine combustor, Proceedings of the Combustion Institute , vol. 31, 3107–3114, 2007.

[7] A. K. Gupta, “Gas turbine combustion: prospect and challenges,” Energy Convers. Mgmt, Vol. 38, pp 1311-1318, 1997.

[8] Daero Joung, Kang Y. Huh, Yunho An, “Parametric simulation of turbulent reacting flow and emissions in a lean premixed reverse flow type gas turbine combustor,” Journal of engineering for gas turbines and powers, vol. 34, 021501, 2012.

[9] P. Di Martino, G. Cinque, A. Terlizzi, G. Mainiero, S. Colantuoni, “Reactive CFD Analysis in a Complete Combustor Module for Aero Engines Application,” Universita Degli Studi Di Napoli “Federico II”, April 26-28,2009.

[10] Ezhil Kumar P.K., “Turbulence and reacting flows (numerical modeling of TVC), short term course at combustion laboratory,” department of aerospace engineering, IIT, Kanpur, 2 MARCH 2012.

[11] Ahmed Abed Al-Kadhem Majhool, “Advanced Spray and Combustion Modeling,” University of Manchester, Faculty of Engineering and Physical Sciences, 2011.

Author Profile

H. A. Bhimgade is a student of final semester of P.G course in Engineering (Heat Power Engineering) of KITS at Ramtek, Nagpur University, India.

S. K. Bhele is an Associate Professor at KITS, Ramtek in the Mechanical Engineering Department. He has 15 years teaching experience. He has published 03 technical papers in International Conference and 05 technical papers in National conferences and is a Life Member of ISTE, SAE, and Associate Member of Institute Engineers. He is pursuing PhD from VNIT Nagpur, India.