International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438

Design and CFD Analysis of Gas Turbine Engine Chamber

Raminder Singh[1], B. Dinesh Kumar[2], Sai Kumar[3]

[1, 2] PG. Scholar (Aerospace), MLR Institute of Technology, Hyderabad, India

[3] Assistant Professor Aerospace Engineering, MLR Institute of Technology, Hyderabad, India

Abstract: ***An engine is a propulsive device used by missiles, rockets and satellites in the reaction control system. These engines are used in reaction control system for station keeping and attitude control. High temperature gases are produced on combustion or decomposition of the propellant. The products produced are discharged through nozzle to achieve high gas velocity and thereby desired thrust. Suitable design of chamber, nozzle plays vital role for effective utilisation of propellant energy. Chamber with one side convergent divergent nozzle and the other side flange model was considered as model. New computational methods are continuously developed in order to solve problems in different engineering fields. One of these fields is gas turbines, where the challenge is to make gas turbines more efficient and to reduce emissions that are bad for the environment. One of the main parts of a gas turbine that can be improved is the combustion chamber. In order to optimize the combustion chamber, both experimental and numerical methods are called for. Numerical optimization implies the necessity to model the most important phenomena in combustion chambers such as turbulent swirling flow, chemical reactions, heat transfer, and so on. In this project we try to design a simple yet accurate model, for a generic combustor of industrial interest, that may be tested in a relatively short time and that yields reliable results. An important topic is here to perform grid sensitivity studies to make sure that the model yields mesh independent results. Another topic of interest is the choice of turbulence model and how this choice affects the grid sensitivity. Heat transfer models are also important to evaluate.***

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

1. Introduction

Energy is needed in order to make machines work. One of the best forms of energy is electrical energy. There are several devices that produce electrical energy such as solar panels, wind turbines and gas turbines. In this project we will focus on gas turbines. Gas turbines produce electrical energy from burning a combustible mixture of fuel (e.g. natural gas or evaporated hydrocarbons) and air. When the gas mixture burns, the volume of the gas will increase. This expansion in gas volume makes a rotor of a turbine rotate and this rotation may then be converted to electricalenergy.

Stationary gas turbine

Objectives

1) To determine the efficiency of the gas turbine combustion chamber.
2) To study the flow of fuel-air mixture in the gas turbine combustion chamber by experiment and CFD analysis.

Scopes

Running a significance experiment to get the efficiency of the combustion process. Design a gas turbine combustor using design software as the model will be Cusson gas turbine combustor unit. Simulate and analyse the design using CFD software.

Software

1) ICEMCFD: This software is used to draw the surface geometry. Then it used again in order to mesh the computational domain which is bounded by the surface geometry.
2) CFX: This is the solver software. This software is used to simulate the flow in the computational domain. Also some part of the post processing is carried with CFXpost

A **rocket engine** is a type of jet engine that uses only stored rocket propellant mass for forming its high speed propulsive jet. Rocket engines are reaction engines, obtaining thrust in accordance with Newton's third law. Most rocket engines are internal combustion engines, although non-combusting forms also exist. Vehicles propelled by rocket engines are commonly called rockets. Since they need no external material to form their jet, rocket engines can perform in a vacuum and thus can be used to propel spacecraft and ballistic missiles.

Rocket engines as a group have the highest thrust, are by far the lightest, but are the least propellant efficient (have the lowest specific impulse) of all types of jet engines. The ideal exhaust is hydrogen, the lightest of all gasses, but chemical rockets produce a mix of heavier species, reducing the exhaust velocity. Since they do not benefit from air, they are best suited for uses in space and the high atmosphere.

Thermal rockets use an inert propellant, heated by a power source such as solar or nuclear power.

Solid-fuel (or **solid-propellant rockets** or **motors**) are chemical rockets which use propellant in a solid state.

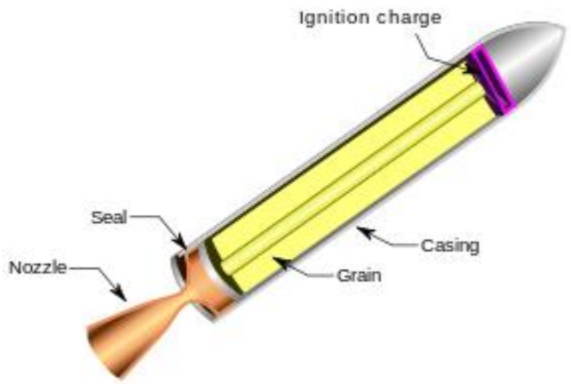
Liquid-propellant rockets use one or more liquid propellants fed from tanks.

Hybrid rockets use a solid propellant in the combustion chamber, to which a second liquid or gas oxidiser or propellant is added to permit combustion.

Monopropellant rockets use a single propellant decomposed by a catalyst. The most common monopropellants are hydrazine and hydrogen peroxide.Rocket engines produce thrust by the expulsion of exhaust which has been accelerated to a high-speed.
The exhaust must be a fluid, usually a gas created by high pressure (10-200 bar) combustion of solid or liquid propellants, consisting of fueland oxidizer components, within a combustion chamber. (An exception is water rockets, which use water pressurized by compressed air, carbon dioxide, nitrogen, or manual pumping.) In rocket engines, high temperatures and pressures are highly desirable for good performance as this permits a longer nozzle to be fitted to the engine, which gives higher exhaust speeds, as well as giving better thermodynamic efficiency.

2. Introducing propellant into a combustion chamber

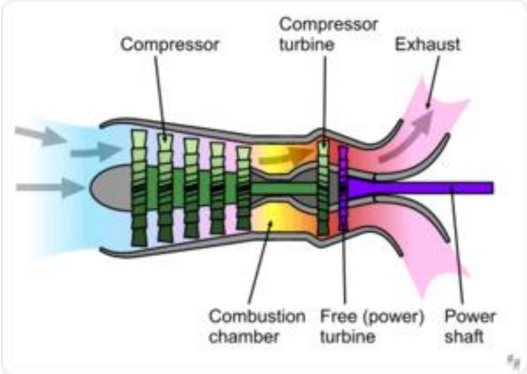
Rocket propellant is mass that is stored, usually in some form of propellant tank, prior to being ejected from a rocket engine in the form of a fluid jet to produce thrust. Chemical rocket propellants are most commonly used, which undergo exothermic chemical reactions which produce hot gas which is used by a rocket for propulsive purposes. Alternatively, a chemically inert reaction mas can be heated using a high-energy power source via a heat exchanger, and then no combustion chamber is used.
Figure



A solid rocket motor

3. Methodology

Figure



These parts are:

Inlet: A gas turbine can have one or several inlets, based on their design and usage. Inlets are used to send fuel and air into the gas turbine. The main inlet in front of the gas turbine is used to suck air in; while there are several other small inlets existing further downstream in order to inject fuel.

Compressor: Compressors are used to increase the pressure of the inlet air, in order to increase the efficiency of the turbine. The effect of compressor, as well as other parts, can be described by using Brayton cycle, as shown in the Figureit will raise pressure from point 1 to point 2. From the diagram one can expect that output work will rise with the raise of pressure in the point 2. On the other hand pressure at point 2 is limited by several parameters such as material constraints, 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 turbine or a turbojet to produce work or thrust. This is an isobaric process.

Turbine: Its job is to drive the compressor shaft and, in the case of a stationary gas turbine, to provide useful mechanical work to drive for example an electrical generator. In ideal cycle, this process is isentropic.

Outlet: This section is designed based on gas turbine usage; for stationary gas turbine
the outlet is a low speed exhaust, which will guide combustion products out of system, either to the environment or to other cycles. For the turbofan gas turbine the outlet is a jet nozzle, which will increase velocity to produce thrust.2.3

4. Combustion

Combustion is a chemical process that burning fuel. Gas turbine engine use internal combustion system to generate thrust. It is all depend on the burning of fuel to produce power.The original substance is called the fuel, and the source of oxygen is called the oxidizer. The fuel can be a solid, liquid, or gas, although for airplane propulsion the fuel is 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 created from the fuel and the oxidizer. These substances are called

International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438

exhaust. Most of the exhaust comes from chemical combinations of the fuel and oxygen. When a fuel bums, the exhaust includes water (hydrogen and oxygen) and carbon dioxide (carbon and oxygen). The exhaust can also include chemical combinations from the oxidizer alone. If the fuel is burned 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 high because of the heat that is transferred to the exhaust during combustion. Because of the high temperatures, exhaust usually occurs as a gas, but there can be liquid or solid exhaust products as well. Soot, for example, is a form of solid exhaust that occurs in some combustion processes.

During the combustion process, as the fuel and oxidizer are turned 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 a product of combustion. To summarize, for combustion to occur three things must be present, a fuel to be burned, a source of oxygen, and a source of heat. As a result of combustion, exhausts are created and heat is released. The combustion process can be controlled or stopped by controlling the amount of the fuel available, the amount of oxygen available, or the source of heat.

5. Combustion Chamber

The combustion chamber is the place where two major events take place; at the inlet fuel will mix completely, or to a sufficient degree, with air. In some combustors fuel mixes with air before combustors, however, in order to achieve a smooth burning, air and fuel should be mixed before burning. There are number of facts that make this part of gas turbine important. In order to make this clear, we will address 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, it can burn incompletely which results in increased levels of CO, soot, NOx and unburned hydrocarbons (UHC).
2) Uneven combustion: This happens when temperature of a section goes high but the neighbouring sections are colder, thus this can result in extra thermal stresses. Thermal stresses may in time lead to material fatigue and failure.
3) Environment: incompletely burned gases or unburned hydrocarbons (UHC) can poison the environment. UHC, NOx and soot are important factors for each burning device. The design should lower them as much as possible.
4) Economy: With increasing price of oil, it is important that gas turbines have high efficiency and therefore low fuel consumption. One of the most important parts, in order to achieve high efficiency, is the combustion chamber.
5) Above factors shows the importance of combustion chambers in gas turbines.

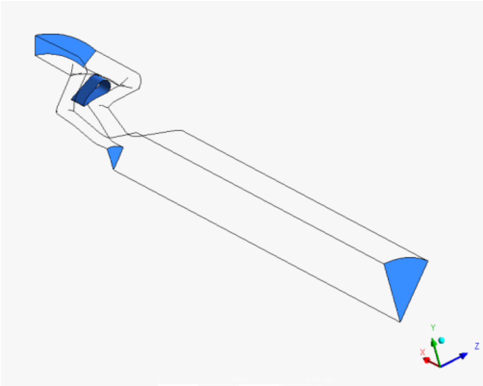
6. Calculation methodology

Geometry Simplifications

The simplifications that were done are the following:

The most common combustors have no symmetry in the domain usually coming from the locations of the burner inlets. The first simplification was to omit these inlets so the geometry becomes symmetric. This implied that only 45° (1/8)th of the full geometry were modelled, shown in Figure.

Figure



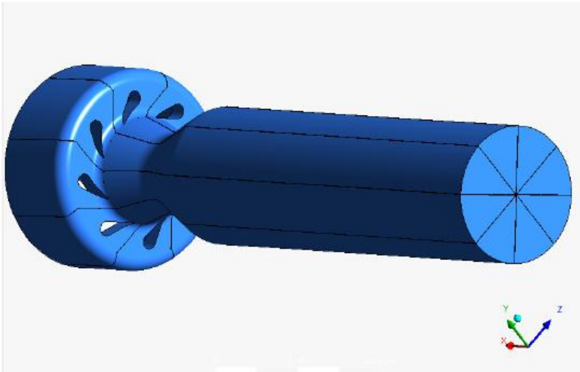
This is one section that has been modelled.

1) We assume one inlet for the fuel and the air. The most common combustors have separate inlets for fuel and air. Both the fuel and the air are assumed to be perfectly mixed at the inlet.
2) NOx formation was neglected and assumed that the fuel will be burn completely.

Modelled geometry

The simplified geometry consists of an inlet, a guide vane and bottom faces are set to walls, while the side faces are axial symmetric shown in Figure 5. There is a secondary inlet in the beginning of the iteration process the mass flow rate is set to zero The full geometry is shown in which consists of 8 sectors.

Figure



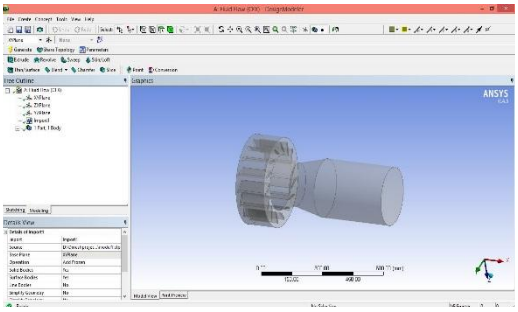
Full modelled geometry

PROBLEM SET UP IN CFD

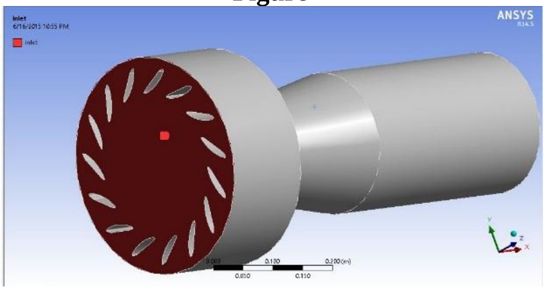
IMPORTING OF COMBUSTION CHAMBER TO ANSYS

Figure

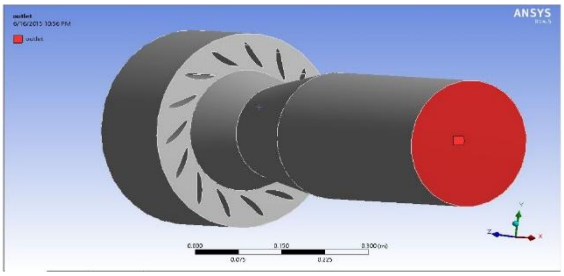
International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438



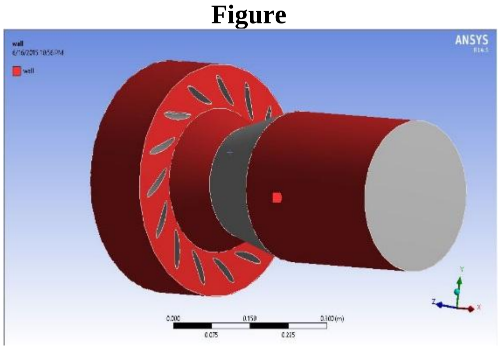
IMPORTING OF COMBUSTION CHAMBER TO ANSYS AND INLET
Figure



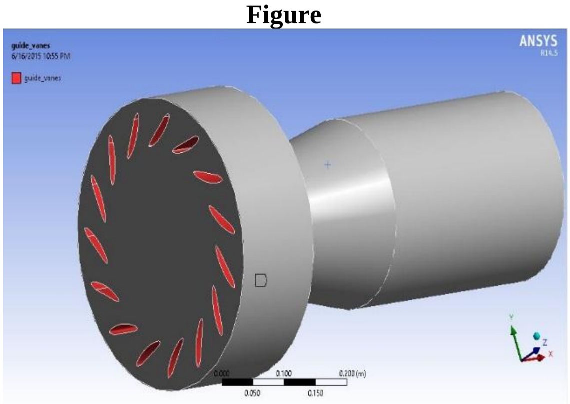
OUT LET OF THE COMBUSTION CHAMBER
Figure



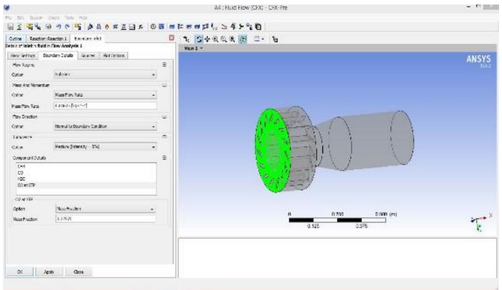
WALL OF THE COMBUSTION CHAMBER
Figure



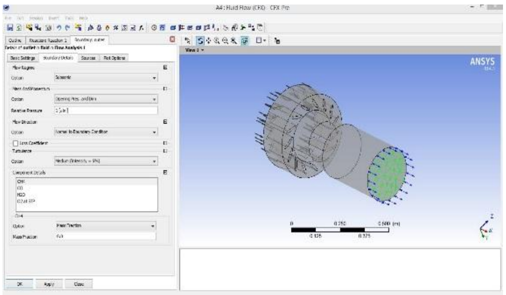
GUIDED VANES OF THE COMBUSTION CHAMBER
Figure



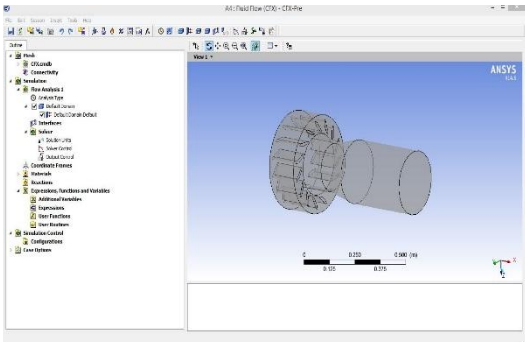
INLET BOUNDARY CONDITIONS
Figure



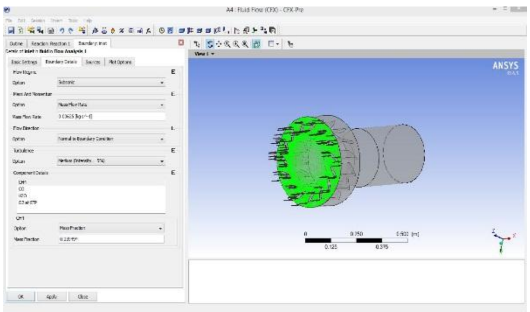
OUT LET BOUNDARY CONDITIONS
Figure



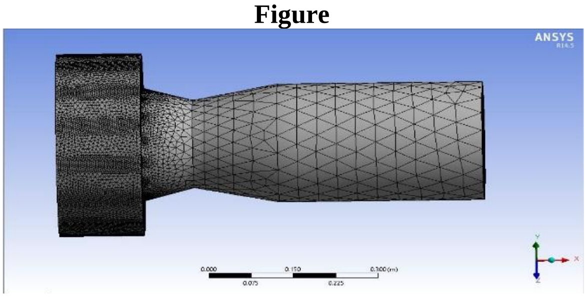
CFX PRE PROCESSOR BEFORE MESHING TOTAL VIEW
Figure



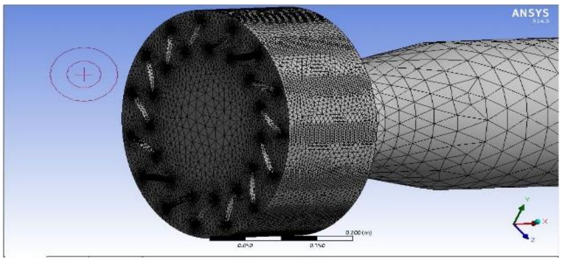
Details of the Fluid Analysis of the Inlet
Figure



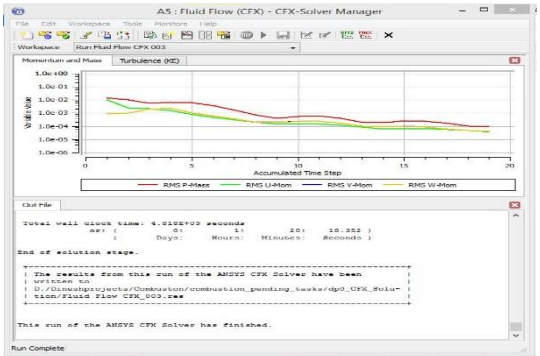
Total Mesh of the Combustion Chamber
Figure



Mesh of the Guided Vanes
Figure

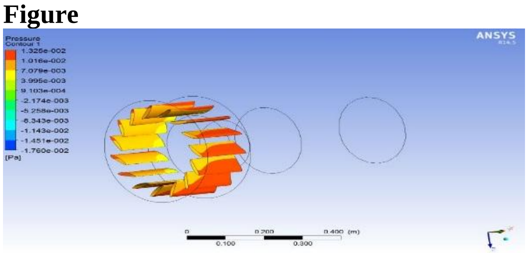


End of the Solution after the Problem Set Up
Figure

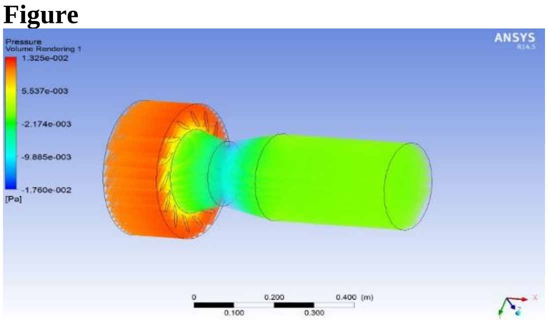


CFD Results of the Combustion Chamber

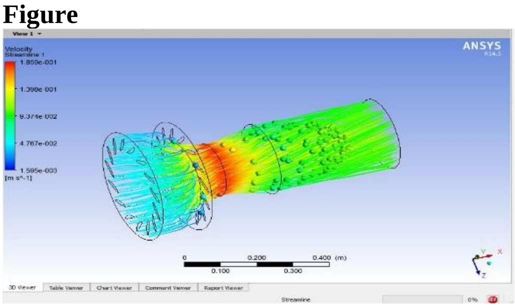
Pressure Counters
Figure



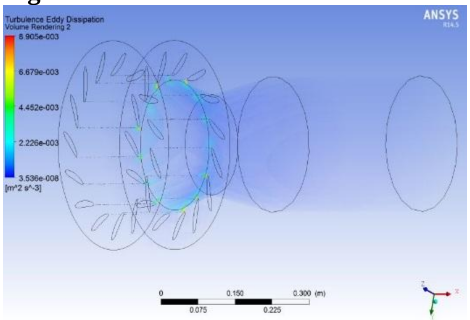
Pressure Volume Rendering
Figure



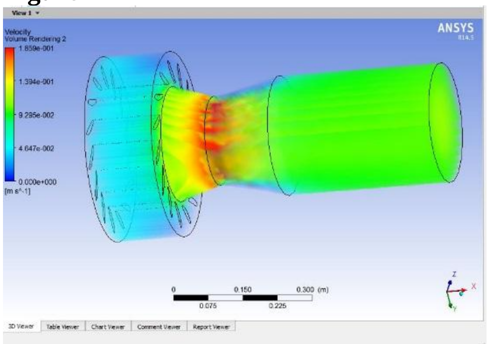
Velocity Streamlines
Figure



Turbulence Eddy Dissipations Volume Rendering
Figure



Velocity Inside The Chamber
Figure



7. Results

Flow solution
ANSYS CFX v14.5 was used as solver. The numerical settings for the solver are described below.

Time stepping
The problem is solved as a steady state flow problem, consistent with the RANS turbulence modelling used, which means that relatively large time steps are used in order to achieve a converged solution as quickly as possible. In spite of the turbulence model the flame itself is slightly unsteady, but the oscillations are negligible.

Heat Transfer
"Total energy including viscous work terms" model is used, which means that the total energy models the transport of enthalpy including the kinetic energy effects. This model should be used where there is change in density or the Mach number exceeds 0.2; in both of these cases kinetic energy effects are significant. In ANSYS CFX, when one chooses total energy the fluid is modelled as compressible, regardless of the original fluid condition, i.e. gases with Mach number less than 0.2. One should know that incompressible fluid does not exists in reality but for the gases with Mach number less than 0.2 the compressible effects are in general negligible.

Turbulence
For the turbulence both the k- **ε** SST and the k- **ε** turbulence models are used. The k- **ε** model is one of the most common turbulence models. It is a two equation model that includes two extra transport equations to represent the turbulent properties of the flow. This allows the model to account for history effects like convection and diffusion of turbulent

energy. The k- ε model has a good prediction in the free stream, but near the walls, the prediction is poor since adverse pressure gradient is presented. For wall treatment scalable wall function is used. Standard wall functions are based on the assumption that the first grid point off the wall (or the first integration point) is located in the universal law-of-the-wall or logarithmic region.

We choose further on 500K also not the lowest mesh density. There exists a circulation zone in front of ignition inlet; this region is one of the most important regions in order to mix flow and better flame. The combustion – turbulence interaction can results in unrealistic combustion model. In further test two other methods will be used, finite rate and eddy dissipation

8. Conclusion

It should be mentioned in the pressure plots and pressure profile of cases, pressure distribution may vary with number of iteration and that is the reason the plots and charts may vary. The variation of the pressure fall into acceptable error range for numerical error, thus the results were accepted. The conclusion from the grid-study is that the mesh-size that is used for the 500K caseis enough, or in other words the results are grid-independent. These conclusions are based on steady-state simulations and were not tested on transient simulations due to limitations of time in the project. This is also important to check in the future work. The 500k mesh size would imply that the number of cells for a full 360o model would be approximately 16M cells. Because this model showed stable convergence and also it predicts flow field better than the other cases.

A new functional expression for such a model parameter, which represents extinction of the flame brush by turbulent eddies, was proposed based on laminar flames. Distribution of air, flow recirculation, jet penetration and mixing are achieved in all the zones of the combustion chamber. The temperature levels near the wall region of the dilution zone suggest some lacuna in the design of this zone.

9. Future Works

The suggestion for future work is to test more models. Also, the simulations done in this work uses CH_4 as fuel, while there are varieties of fuels available to use.
Suggestion for future work:

- Test the open source software Open FOAM
- Test different fuels
- Modelling the generic gas turbine combustor with different inlets for fuel and air and use pre heated air.
- Grid study for transient simulations

References

[1] Riccio G., Marini A., Martelli F., “Numerical investigation of gas turbine combustion chamber hydrogen fired”, 19th ISABE Conference, ISABE 2009-1112, Montréal, Canada.
[2] FLUENT 6.3 User’s Guide, 2006.
[3] Barlow R.S., Carter C.D., “Relationship among Nitric Oxide, Temperature and Mixture Fraction in Hydrogen Jet Flames”, Combustion and Flame 1996, Elsevier.
[4] Hsu A.T., Anand M.S., Razdan M.K., “An assessment of pdf versus finite-volume methods for turbulent reacting flow calculations”, 34th Aerospace Sciences Meeting & Exhibit, AIAA-96-0523, January 15–18. Reno, NV: AIAA, 1996.
[5] Pope S.B., “Computationally efficient implementation of combustion chemistry using in situ adaptive tabulation”, Comb Theory and Modelling 1997; 1:41–63.