International Journal of Mechanical Engineering
ISSN : 2277-7059 Volume 2 Issue 2
http://www.ijmejournal.com/ https://sites.google.com/site/journalijme/

LARGE EDDY SIMULATION (LES) OF EFFECT IN SWIRL NUMBER TO THE EFFICIENCY OF GAS TURBINE COMBUSTION

Farid Viko Bhaskarra[1], Dr. Gunawan Nugroho[2].
Department of Engineering Physics, Faculty of Industrial Technology
Sepuluh Nopember Institute of Technology (ITS) Surabaya Indonesia 60111, email: [1]faridviko@ep.its.ac.id ; [2]gunawanzz@yahoo.com

Abstract—A good mixing process are required in designing gas turbine combustor. Numerical simulations using Large Eddy Simulations are well suited to address these issues. In this study, a numerical simulation of non reacting flow in gas turbine combustor was performed. There were 5 variations of swirler's angles (5^0, 15^0, 20^0, 25^0 and 30^0). Perfomances of these new swirler were investigated. The main target of this investigation is to get the effect of swirler's angle to combustion recirculation zone. The results show that the longest flame stagnation point of 45,26022 mm was obtained at 25^0 of swirler's angle.

Keywords : Gas turbine, Large Eddy Simulation, Swirler

I. INTRODUCTION

In the various ways in the generation of mechanical power turbine is best. Absence of piston engines and wrapping rubber, this means that problems in equilibrium condition during the process is reduced. The requirement for lubricating oil is also getting smaller and reliability of a system can be high. The advantage of using this first in an effort to realize an electric power generation with water as the working fluid from the system. The system is better known as the Steam Turbine. The use of steam turbines are the first time in the early 20th century and has become the primary tool used in the generation of electrical power source. The power generated can reach 1000MW with efficiency reaching 40 percent at this time. Although there are advantages in this steam turbines, but there are also some lacks. Production of high-pressure steam and high temperature are very high costs involved in purchasing equipment and installation of boiler systems either conventional or nuclear reactors. In addition the use of gas fuel in the boiler is not used directly by the gas turbine fuel but is used as fuel to heat water to produce a fluid that is in the form of steam, based on these two ideas for direct use of fuel gas is constantly being developed. The development of this new system begins in the moments before the second world war, called the Gas Turbine.

To generate an expansion through a turbine, the pressure ratio should be provided, and the first thing which has to be considered in the gas turbine cycle is a compression of the working fluid in the cycle. When the compressed working fluid is expanded through a turbine, and in this process heat loss does not happen so output power from gas turbine will be equal with required power at compressor to compress the working fluid. Cycle power can be improved by the addition of energy to raise the temperature of the working fluid. The addition of this energy happens after the working fluid compressed in the compressor section. The addition of this energy not only increases the power of this cycle, but also can be used to drive the compressor by a shaft that is connected between the compressor and turbine.

The performance of gas turbines can be known through two main factors i.e : efficiency and working temperature turbine components. The combustion chamber is one component of the gas turbine that is the most influential in the system performance. On this components there is an addition of energy to raise the temperature of working fluid that is compressed air. The addition of energy in the gas turbine is done by burning the compressed working fluid in the combustion chamber system [1]. To determine the performance of the combustion chamber, there are a variety of ways i.e: by direct experiments on the gas turbine system and numerical study of the combustion. Direct experiments on gas turbines require a very large fee to do so and how this is rarely used to analyze the combustion in gas turbines and has some limitations. Therefore numerical studies are required. CFD (Computational Fluid Dynamics) was used in this numerical study. In this CFD, flow of the fluid plays an important role. CFD is essentially a replacement science of governing equations that describe the basic principles of physics with discretization form of algebra, which in turn is solved to obtain numerical values for the flow field at discrete points of space and time domains.

There are various methods in CFD, one of them is the LES (Large Eddy Simulation). LES is a standard method to study the dynamics of turbulent combustion. For example, LES emerged as one of the tools to predict and study of combustion instabilities found in the gas turbine. Large Eddy simulations (LES) of turbulent flow is a powerful technique which consists of the elimination of the scale. The scale is smaller than scale Δx with a appropriate low-pass filtering to allow the development of equations that are suitable for large scale to be written. LES is a tool to translate the structure of turbulence swirling, because LES allows for us to capture the deterministic formation and progression is not yet known of the relationship between the vortices and structures. LES also allow a prediction of the many statistics associated with turbulence and a mixture of induction (forced). LES has contributed the development in the aerodynamics of vehicle, train and airplane [2].

In addition, the LES approach is intermediate between the RAN (Reynolds Average Modeling) and DNS (Direct Numerical Simulation) while the RAN is the only approach to a model based on the collection of statistics, DNS is an approach that can resolve the entire flow of the fluid that moves against the direction of mainstream especially in a circular motion. Consequently, all sorts of aspects included in this method and sub-grid scale used is very detailed. RAN is typically used in steady state simulation and this CFD methods can not be used to explain the finer detail, while the use of DNS in this simulation is not needed because the combustion simulation has a low Reynolds number flow and simple form refers to the need of resolution. It is not worth the effort and funds that have been issued. To meet the challenges of cost and time effectiveness, LES is used in combustion simulations. In this method, the turbulent movement of large-scale computed numerically (without the required physical models) and only small-scale sub-grid [3].

The advantages of LES is capable of providing three-dimensional simulation of the movement depends on the time scale for turbulent mixture with a much smaller scale than the modeled grid (RAN) computing [4]. Non-premixed combustion use a conserved scalar formalism to avoid the problem of evaluation of sources of chemical problems. In this approach, chemical reactions are assumed to "fast" domains so thermochemical of the mixture can be determined only in the form of conserved quantity of the mixture fraction.

LES is also based on the theory of Kolmogorov (1941) which are very well known about turbulence. The theory assumes that a large Eddy of the flow does not depend on the geometry of the flow, where the smaller Eddy are same with itself (self-similar) and have a universal character. This become an application to complete only on large eddy explicitly. In LES, the movement of the flow is calculated based on the universal influential of smaller scale (sub-grid) which is modeled using a SGS model (sub-grid scale). SGS is required to complete a filter based on the Navier-Stokes equations with additional constraints SGS stress. LES filtering is more widely used than averaging as was done in RANS, small parts on the LES is no longer zero $U \neq 0$ (u is the flow velocity vector). In the LES filtering (Filter) is spatially applied to the equations of motion. This spatial filter is a function of spatial convolution filter with a filter function of the kernel, the kernel function is a local function [5]. Fil ter kernel that is widely used in LES is Gaussian, box and cutoff [6].

Swirl number of a particular design is obtained when the average speed of axial, radial and tangential known as a function of radial location in combustion chamber. The following expression can be applied if only the axial and tangential velocity are known:

$$\boldsymbol{S} = \frac{\int_{R_i}^{R_o} vwr^2\,dr}{\int_{R_i}^{R_o} v^2 r\,dr} \qquad \textbf{(1)}$$

V is a component of the average axial velocity, W is the component of the average tangential velocity and R is the radial loc ation. The density is assumed not to vary, and the radial pressure gradient generated by the swirling motion is ignored in the above expression. This simplification causes some error. Swirl Numbers of rotating flow is a conserved quantity. If the pressure is negligible, swirl numbers obtained will not be seen as a conserved quantity on stream. Measurement of static pressure on the flow is hard to do. However, and a modified form of the equation above that contribute to the pressure variations are based on the axial and tangential velocity relationship is not suitable for high swirl numbers (where the swirl number, S> 1).

Giving the effect of swirl on the flow can be done in various ways. Practical applications, in general swirl is generated by the design Swirler, which is a package of blades mounted on the nozzle air. Swirler blades are generally parallel to the direction of airflow. The degree of swirl blade reaches a maximum around 70^0. Value greater than 70 usually does not happen Swirl effect, because these blades have a thickness of a limited (finite thickness). There is a pressure drop associated with blade shape. Pressure drop parameters are intimately affected the efficiency of the combustion chamber, it became necessary to a calculation in whi ch the swirl design will produce a low pressure drop. A rotating flow is also quite sensitive to input conditions. Swirler design is not perfectly symmetrical and would not produce effects that are not symmetrical swirl on the velocity field. The result, the sha pe of the resulting fire would not symmetrical, and this will affect combustion characteristics of fire.

There are 2 types of turbine blade i.e : straight and curvy. Swirler which has a straight blade, the degree of the blade is not affected by the radial distance from the blade. This type is more easily made but is less efficient in pressure drop problem. Curvy Swirl has a small swirl angles close to the center line and swirl angle the greater the greater the radius of the swirl. Numbers swirl to swirl blade is straight blade types which can be defined as follows :

$$\boldsymbol{S} = \frac{2}{3}\left(\frac{1-\left(\frac{d_h}{d_o}\right)^3}{1-\left(\frac{d_h}{d_o}\right)^2}\right)\tan\emptyset \qquad \textbf{(2)}$$

d_h is diameter inside of the swirler, while d_o is the diameter outside of the swirler. In (2) the value of numbers swirl swirl angle depends on the value $\emptyset$.

The phenomenon of vortex breakdown is characterized by circulated toroidal area which is located on the area after the swirl generator. Typically, the shape of the flow in this region will sustain an enlargement (expansion), due to the combustion chamber shape is greater than the swirl shape. In the room with a swirl stabilized combustion, swirl motion produces a low pressure area in the middle of the stream. If the swirl number is quite high, a low pressure area will be enlarged so that the movement will slow the flow, high pressure gas is pushed back from the downstream to upstream. Phenomenon in quite useful in stabilizing the combustion, because this creates a good mechanism that high temperature gas product can be contacted directly (reacted) with air, unburned fuel and oxidizer.

II. GOVERNING EQUATION

Governing equations for fluid flow can be obtained from the three basic equations of physics are the conservation of mass, momentum, and energy. These three basic equations can be used in theory and in CFD-based simulation. But for the process involve a change of chemical compounds, it is required species conservation equations. In using the approach of an infinite fluid element moving along the current lines here are the basic equations for steady flow, three-dimensional, capable of incompressible, and viscous flow for LES method [6].

Mass Conservation

In determining the flow through a control volume, if the amount of mass flowing into the control volume is not equal to the amount of mass that flows out of control volume, there would be changes in the amount of mass that flows inside the control volume. If the outflow is greater than the incoming flow, then a decrease in the amount of mass within the control volume, and vice versa [7]. Based on the above phenomenon, then the mass conservation can be expressed as, [7]:

$$\frac{\partial}{\partial x_i}(\rho \bar{u}_i) + \frac{\partial \rho}{\partial t} = 0 \qquad \textbf{(3)}$$

Energy Conservation

The first law of thermodynamics provide an understanding of the total energy. The total energy consists of kinetic energy, potential energy and internal energy. Internal energy can be divided into thermal energy and other forms of energy such as chemical energy and nuclear energy. For the study of heat transfer, it is focused on mechanical and thermal forms of energy. Addition of thermal energy and mechanical energy is not conserved, because there can be conversion in forms of energy between thermal and mechanical energy. For example, if a chemical reaction occurs that decrease the amount of chemical energy in the system, it will result in an increase in thermal energy in the system. Based on this, it can be expressed as :

$$\frac{\partial \bar{h}}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{h} \bar{u}_j) = \frac{\partial}{\partial x_j}\left(\frac{\mu}{Pr}\frac{\partial \bar{h}}{\partial x_j}\right) - \frac{\partial q_{jsgs}}{\partial x_j} \qquad \textbf{(4)}$$

Species Conservation

Species conservation equation can be analogous to the same as the equation of energy conservation in a control volume, so that:

$$\dot{M_{st}} \equiv \frac{\partial M_{st}}{\partial t} = \dot{M_{in}} - \dot{M_{out}} + \dot{M_g} \qquad \textbf{(5)}$$

Above equation can be expressed as

$$\frac{\partial}{\partial t}(\rho \bar{u}_i) + \frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j) = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \bar{u}_i}{\partial x_j}\right) - \frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial \tau_j} \qquad \textbf{(6)}$$

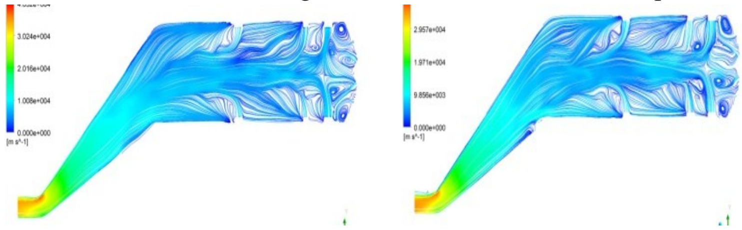
In (2.16) species equation is convoluted with LES Sub grid scale, LES sub grid scale can be expressed as :

$$\tau_{ij} \equiv \rho \overline{u_i u_j} - \rho \bar{u}_i \bar{u}_j$$

III. RESULTS AND DISCUSSION

Velocity field

Simulations are very useful as a qualitative consideration of the nature of which is shown by the velocity field in the case of combustion, as shown in Figure 1 is streamline in a vertical plane.



a **b**

Figure 1 Streamline at vertical plane

In Figure 1, it is clearly that the difference in the flow topology will be predicted. This difference occurs due to interaction of the vortex core with a flow distribution that occur due to the flow stream is passed through a nozzle (jet stream), where the flow is moving the back-flow in the primary area. Vorticity is formed on the state of the input ratio of air and fuel are the same, but it is difference with Swirler's opening. The differences flow structure is closely related to differences in opening angle swirler where for each case given the difference in angle of 5^0. The existence of an additional flow structure effects the configuration of the flow on the primary area that is also changing the distribution of temperature and species.

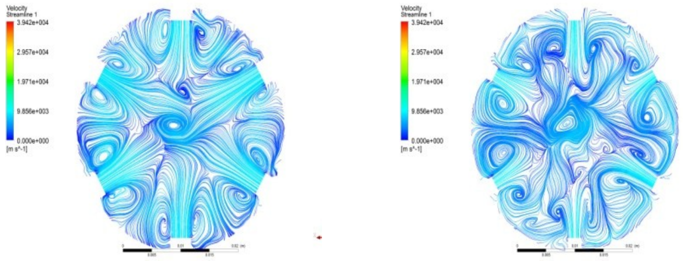


Figure 2. flowfield at primary section (left) and secondary section (right).

In Figure 2, it is clear that the flow of air from the hole primary and secondary holes collide with each other, and it produces a recirculation flow in the middle of the engine. Besides this, the phenomenon is also influenced by air flow through the combustion chamber of the head. It has been happened in experiment which has been done by Bicen and Palma [15]. Rotating core in the middle of the combustion chamber is an evidance to the process of averaging in the region where the flow of the jets impinge with each other and vary with time. In fact, Figure 2, it is a suitable representation and better to the behavior of the flow in the combustion chamber, on the other hand in Figure 1 shows the flow behavior to the "individuality", which meant individuality here is the flow of formation that occurred did not affect each condition in the combustion chamber so it is not sufficient in representing the behaviour of flow in combustion chamber. In Figure 2 it is possible to identify asymmetries in the jets penetration and this may be related to primary and secondary forms of the hole.

Temperature field

As previously described, the flow pattern can be directly known in the time change of the temperature field as shown in Figure 4.3 that displays six different states.

Previously described flow pattern can be directly known in the time change of the temperature field as shown in Figure 4.3. There is a hot area that is visible on the primary combustion region, regional and local secondary dilution. Combustion does not occur again after the dilution and flame stabilized in the recirculation region at the head of the combustion chamber. In the area around the combustion chamber wall reactions do not occur and the combustion process because the area around the walls have been isolated by the flow of air that is released by the air that surrounds the combustion chamber walls, it can be provided in each image. Flow visualization studies showed that some previous events that have been exposed closely related to the lack steady (unsteadiness) around the point of impingement (impingement region) and their jets is an important influence on the temperature distribution on the output of the combustion chamber.

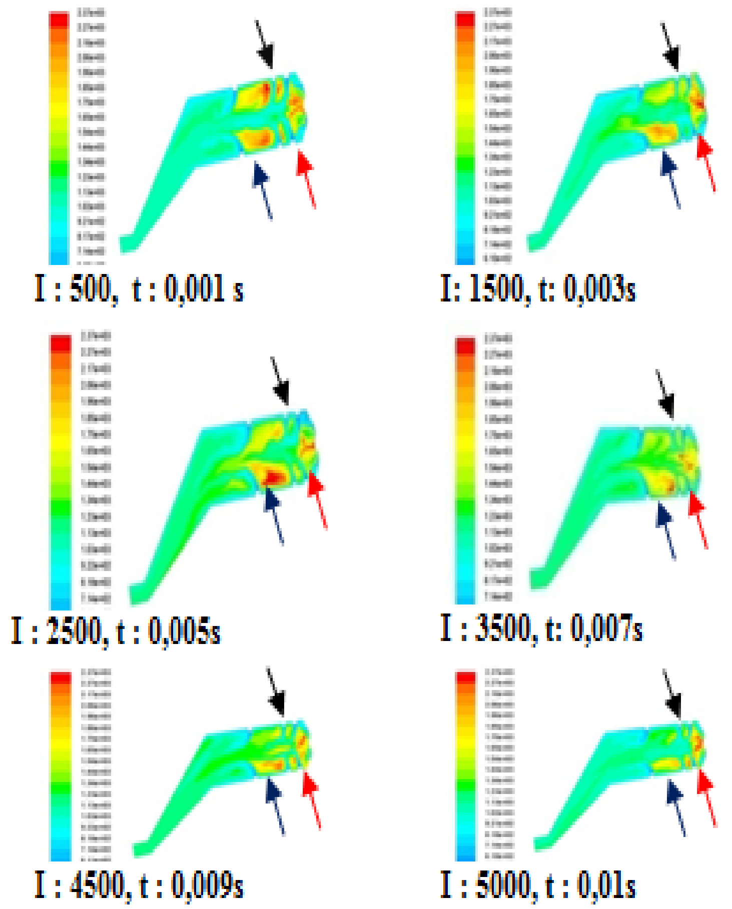


Figure 3 Temperature field every time step. Red arrow, black arrow and blue arrow represents primary, secondary and dilution zone respectively. I and t represent time step and time simulation

Flame Stagnation Point

Vortex breakdown phenomena in a vortex streamline is related with Reynold number of inlet stream and Swirl number (S_n). It is caused by boundary condition at downstream. Effect of this vortex is investigated furthermore with proposing a new design. The new design has difference swirl's angle each condition. In this part, it will be discussed the CRZ form and additional form from recirculation zone. Swirl number is calculated from 5 difference conditions and it is calculated when the air flow coming out from the Swirler.

Table 4.1 Swirl number at 5 conditions

Angles	5^0	15^0	20^0	25^0	30^0
swirl number	0,115	0,352	0,478	0,612	0,758
Flame stagnation point (mm)	39,13892	39,69542	43,0343	45,2602	39,3244

From table1, it can be seen, the increasing of swirl number cause the flame stagnation point, except in the 5^{th} conditions. In that condition, swirl's angle is designed in 30^0 and for this combustor model it is not appropriate. The represented contour of flame stagnation point is at 2300 K each condition.

The contour of the back-flow on stream wise field is shown in Figure 4. In Fig 4 represents the dependence of the CRZ (combustion recirculation zone) and it is evidence that the CRZ can be controlled by applying a new design. In the picture are also explained by changing the value of Swirl number, with changing its dimensions and angles of the Swirl, it can reduce the influence of the dead zone. This area has a negative effect on the combustion process and the uniformity of wall temperature.

Therefore, relieve or at least reduce this area is vital to improve the combustion process. Reduction of this area can lead to increased pressure drop in the combustor.

Static pressure is plotted in Figure 5 for all circumstances. It is clear that the increase in static pressure in accordance with the increase in the value of Swirl numbers. This can be understood physically, namely the influence of shear stress CRZ area, which dissipate the energy flow, in this case the pressure.

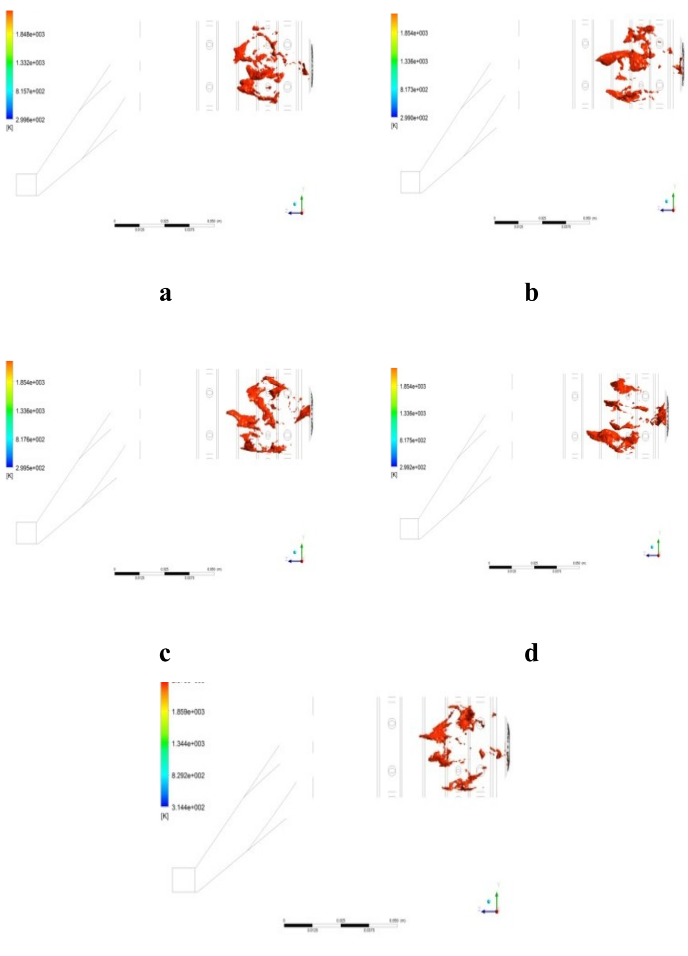


Figure 4 I*so surface* contour of flame at temperature 2300K for angle *swirl* : a. 5^0, b. 15^0, c. 20^0, d. 25^0 and e. 30^0

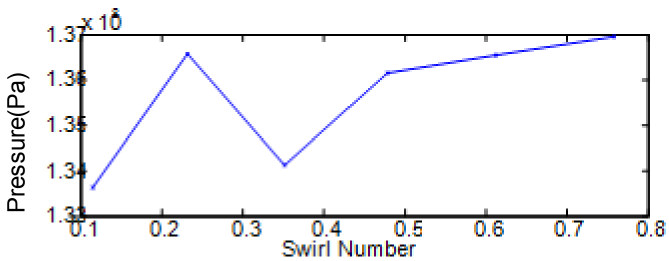


Figure 5 Static pressure in each condition

Primary Zone (Plane 1)

In addition to determining the design of a burner (burner), another thing to consider in research on combustion is the reduction of pollutant emissions. In a simulation study was conducted with the combustion gas turbine inlet fuel and the same air in each simulation condition. Different conditions ie on the value of Swirl numbers. Parameters used as consideration weather

International Journal of Mechanical Engineering
ISSN : 2277-7059 Volume 2 Issue 2
http://www.ijmejournal.com/ https://sites.google.com/site/journalijme/

good or poor combustion process that occurs is the mass fraction of CO and CO2. They are product of combustion and O2 mass fraction as a compound needed for combustion. The resulting temperature is also one consideration in this study.

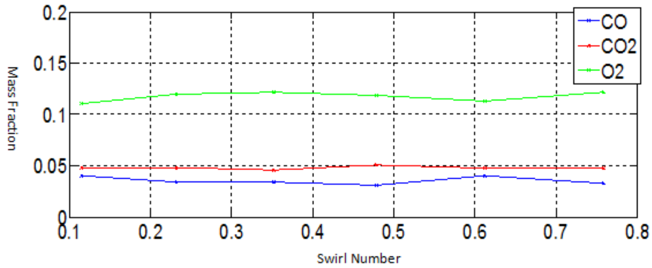
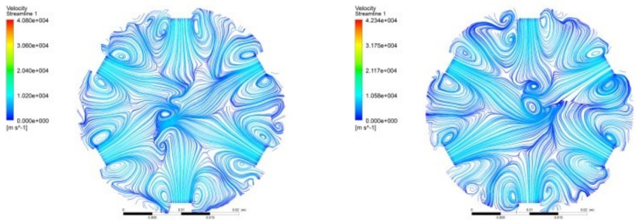


Figure 6 mass fraction in plane 1 (Primary zone)

In Figure 6 shows the changes in the mass fraction of CO, O2 and CO2 for every number in the Swirl at Field 1. Field 1 is the intersection of the transverse to the z axis, where the area of a cut cross section of the primary. In Figure 6, the mass fraction of oxygen tends to form the curve of the arch and at the Swirl number 0.7583, there is irregularities i.e the value of oxygen mass fraction rises. The maximum value on the curve arch occurs at 0.3519 Swirl numbers. Mixing process in this situation is less good, depicted from the flow field that forms a vortex at 0.3519 Swirl numbers are not quite perfect (Figure 7a), while in figure 7b more perfect mixing occurs.



a b

Figure 7 contour of velocity streamline plane 1 for swirl number a) 0,3519, b) 0,478

Velocity value is very high on the state than others, it is about 5589.56 m / s. In Figure 7b, good mixing process that occurs due to the vortex is very good so it can mix fuel (gas) and the air. Its mixing is very good. The temperature produced in this state around 1414.23 K.

In conditions of very high velocity (Figure 7a), the combustion process is less good. It can be seen in Figure 8. The resulting temperature is quite low in Figure 7a. In these conditions the highest temperature is obtained at the numbers 0.478 Swirl combustion and mixing process that occurs best among others, this can be seen the low content of CO and high temperatures in this state.

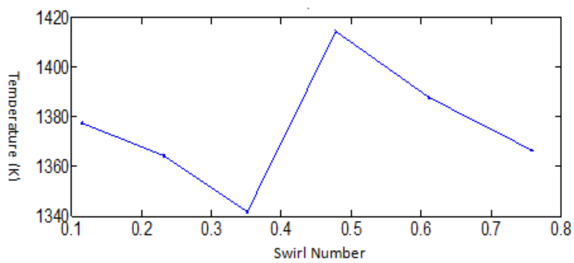


Figure 8 Temperature at plane 1

Secondary Zone (Plane 2)

Plane 2 is a cross-section of the secondary zone, in this part the fuel (gas) remaining from the process on the primary zone is burned again after this section, so the remaining fuel is not there anymore.

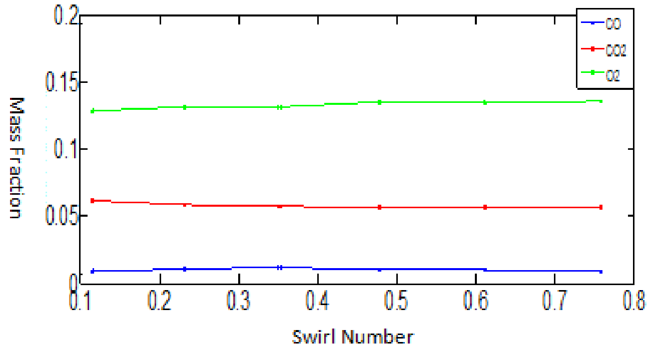


Figure 9 Mass fraction at plane 2

The concentration of carbon dioxide (CO_2) is greater than the concentration of carbon monoxide (CO) and oxygen (O_2) is seen in Figure 9, this is due to carbon dioxide in this section is the accumulation of carbon dioxide before combustion and in this section alone. CO mass fraction is zero for almost every situation, this means that, in this section the combustion process is complete.

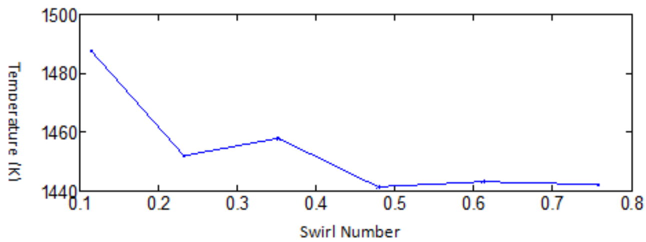


Figure 10 Temperature at plane 2

In Figure 10, the temperature for each state has declined. However, the temperature is higher than the field 1, because the combustion happen again, which means the addition of combustion energy followed by a rise in temperature.

After this section (secondary), there is the dilution zone, where the fire occurred this section is diluted, besides to dilute the fire, this section also serves as a stabilizer of the fire, so fire in this section will be stabilized. After the dilution zone, the combustion does not happen again. The purpose dilute the fire is to lower the temperature of the fire which is followed by a decrease in air temperature in the combustion chamber that will be used to rotate the turbine blades, turbine blades can only accept a certain temperature depending on the kind of the material forming a turbine blade.

The combustion is said to be perfect, if the used fuel is all burning and no left. In the combustion, it is not only fuel but also air and heat to make s a combustion.. In this process the reaction occurs between some species of the carbon (C), hydrogen (H) reacts with oxygen contained in air. This reaction will produce carbon dioxide (CO_2) and water vapor (H_2O) when the process is perfect (complete). This state that the required oxygen in the combustion is enough to burn the carbon and hydrogen. Combustion is said to be incomplete combustion (incomplete) if the amount of used oxygen is less, or the ratio of air and fuel that does not meet, so the fuel is partially not burnt (ignition temperature). Flue gas still contain components that can be easily burned, primarily carbon monoxide (CO). The results of the combustion reaction of the desired effect are heat and pressure.

IV. CONCLUSION

Novel swirler have been proposed with redesign the swirler's angle at gas turbine combustion chamber, this cause the change of swirl number. Based on discussion of simulation result, the variation of the change swirl number. Therefore, it can be concluded :

- The Change of swirl number value, resulting in the length of stagnation point flames (flames) were fluctuative. The longest stagnation point occurred at the swirl number with a length of 0.612 mm 45.02666 flames.
- In the primary area, the best mixing occurs at the swirl number 0.478, where the mass fraction of CO gas remaining 0.0313357. The resulting temperature at the swirl number is around 1414.23 K.
- In the secondary area, the remaining mass fraction of CO is almost close to zero for all conditions. Temperature for each state, its value is higher than state 1. The highest temperature in this region, occurring on the state of swirl number 0.06285.

V. REFERENCES

[1]. Levabre, Arthur H. "*Gas Turbine Combustion*" Mc Graw Hill 1983.
[2]. Jiang, Xi. Hong lai, Choi. "*Numerical Techniques for Direct and Large-Eddy Simulation*" Chapman & Hall/CRC.2009.
[3]. L.X. Zhou, L.Y. Hu, F. Wang."Large eddy simulations of turbulent combustion using different combustion models. *Fuel* 87 (2008). 3123-3131.
[4]. G Boudier, L.Y.M Gicquel. "Comparison of LES, RANS and Experiments in Aeronautical Gas Turbine Combustion Chamber". CERFACS Turbomeca (2006)
[5]. F. di Mare, W.P Jones, and K.R. Menzies "Large Eddy Simulations of a model gas turbin combustor", *Combustion and Flame* 137 (2004). 278-294.
[6]. Yan Ying-Wen, Zhao Jian-Xing, Zhang Jing-Zhou, Liu Yong."Large Eddy Simulation of two-phase spray combustion for gas turbine combustor" *Applied Thermal Engineering* 28 (2007). l 1365-1374.
[7]. Yehia A.Eldrainy, Hossam S. Ally, Khalid M. Saqr, Nazri Mohd Jafaar. "CFD Insight of the flow dynamics in a novel swirler gas turbine combustor" *International Communications in Heat and Mass Transfer 36(2009)* hal 936.