MODELLING FOR CROSS IGNITION TIME OF A TURBULENT COLD MIXTURE IN A MULTI BURNER COMBUSTOR

Eng. Mazen Abu Amro¹, Eng. Ali Imara² and Samy S. Abu Naser³

¹Faculty of Engineering and Information Technology, Al Azhar University, Gaza

²Palestinian Airlines, Quality Insurance Department, Gaza

and

³Faculty of Engineering and Information Technology, Al Azhar University, Gaza

ABSTRACT

The impact of Cross Ignition process (CI) in the gas turbine operation and environmental issue is still investigated for extending the efficiency of gas turbine engines and meanwhile decreasing the environment pollution. This paper presents various constructive influential parameters and analysis of their related interaction during CI. A developed computational model for determination of cross-ignition time (CIT) is proposed, based on previous relevant models for thermal analysis and for distinguishing of heat fluxes in combustion processes.

Due to the first analysis of theoretical results, experimental investigation for various operating conditions were essential to validate the developed computational model of the CIT. Thus, a simple experimental test rig is designed for this purpose, and for validation of certain conditions of the computational model.

Meanwhile, for expanding the investigations in higher energy conversion and reducing expensive test-procedures, that are conducted during critical test running, a new strategy is proposed for simulating the thermal heat fluxes throughout the burners compartment model by implementation of Computational Fluid Dynamic (CFD).

Finally, new constructive criteria based on the validated investigations will enable the future generation of gas turbine combustors to operate in critical conditions.

KEYWORDS

Multi- Burners, Liquid fuel, Cross- Ignition, Heat Flux, CFD.

1.INTRODUCTION

The phenomenon of cross-ignition requires in practice hot combustion gases and emissions flowing sequentially to the adjacent burner for ignition purposes. A heat flux is then spreading into the targeted burner, until the cold mixture in this burner is appropriate for a cross-ignition to be carried out. Until now, only few combustion studies have been established about determination of CIT based on controlling the heat fluxes in turbulent mixing processes.

During this combustion process, turbulent energy transport and diffusion processes are being investigated, as heat energy is transferred from an active burner to a fresh mixture jet of an adjacent burner. Meanwhile, chemical reactions are running parallel, until a cross-ignition is observed at a particular CIT and ignition temperature (T_{ig}). In this case, the CIT and T_{ig} are

affected by several physical and constructive parameters. Namely, by fuel/air- mixture properties, and the implemented constructive conditions that affects the heat exchange process throughout liners and mixing zone of both burners.

Due to this methodology of influencing the heat exchange process in the combustion system through constructive parameters, theoretical models can be derived for calculating the CIT of the cross ignition process throughout heat balance equations, and simultaneously identify the interaction between these influence parameters for the entire process. Due to limited facilities, the established experiment in this study is conducted in targeted industrial operation conditions [1], and low turbulent gradients [2], considering the CIT as the main evaluation measure during the experimental procedure.

Thus, a 2-D schematic compartment model of both burners (Fig. 1) was first proposed for initial configuration of the test- rig, and meanwhile for distinguishing the heat fluxes throughout the entire system.

For enhanced investigations in high gradient regions and substituted for high cost test running, it is essential to implement a CFD simulation.

Fig. 1: Schematic of the heat balance between flame and adjacent mixture jet illustrated on a control volume ($V_{contr.}$) of the adjacent mixture.

The following influential parameters are entitled in the balance equations of a proposed theoretical model, as for construction of the desired experimental test- rig. They are also adopted as the proper varied operation parameters:

- the contact area.
- the speed and swirl intensity.
- the distance between each two burners.

Within this schematic model, several concepts of operation conditions can be illustrated and investigated. Specially, variation of the control volume of the mixing zone. Following analysis of the validated results, certain criteria can be developed to control the entire cross-ignition process.

2.LITERATURE REVIEW

There are few studies that have targeted the problem of cross-Ignition between turbulent mixture jets and an ignition resource. Very little of these studies have examined this problem experimentally throughout mutual heat fluxes, and according to relative operation conditions.

M. Boileau [3] has studied the ignition sequence at an annular combustion chamber as well as the flame propagation from one burner to the next adjacent one. He presented a simulation for the flame topology in 3D, using the Large Eddy Simulation (LES) at a combustion chamber with full contact burners built by VESTA Turbomeca with 18 burners (See Fig. 2).

Fig. 2: Sketch of the annular combustor showing the positions of the two igniters (I1 and I2), the eighteen sectors (S1 to S18), the four quadrants (Q1 to Q4) and their four corresponding azimuthal angles ($\Theta 1$ to $\Theta 4$). $\Theta 4 = 50$ and 70 deg.

As he had showed, that the injection rate has a strong influence on the CIT. And that the transmission rate of a burner flame to the next one is higher than the flame speed due to the effect of thermal expansion, he divided the chamber into sectors and calculate the CIT from a sector to the next one. He could modify the flame speed by the main burners aerodynamics due to the swirled injection, computing the ignition time and thermal fluxes of every sector, but not for individual burner, as the present study is focusing on.

A previous study of M. Jordan [4] has implemented a simple experimental test rig consisted of three compartments to examine the ignition of a turbulent mixture in the adjacent (the middle)

compartment (Fig. 3). By a tow ventilators supported on the separation wall, he could varying the expanding turbulent mass flow of hot gases resulting from an ignition in the first compartment.

Fig. 3: Experimental test rig for ignition transition at Pisa University.

The pressure and temperature in the middle compartment were consequently increased gradually until an ignition occurred. M. Jordan has found that fuel/air ratio and the velocity of hot gas flux play a major role in heat and mass transfer to the fresh mixture, and meanwhile to the CIT of this mixture. In Jordan's study, the heat fluxes through the liners and the variation of the pass aria were not investigated.

An enhanced study of C. Hirsch [5], has examined the impact of cross-ignition for liquid fuel experimentally by using a three- burner experimental setup (Fig. 4). He depended on the mixture ratio of individual burners (Φ) and on the variation of the pilot fuel ratios of every adjacent burner to identify CI- Limits (Φ ignition).

Fig. 4: A three- burner test rig cross section

He used a general quantitative methodology to describe the combustion properties of the mixture in different operating conditions, where a mixing model was represented to control the cross-ignition limits in a combustion chamber relative to the mixing behavior of the burner, without determination of the CIT. Whereas, in the present study, it is possible to vary the contact area on the burner liners and the distance between individual burners in the simple proposed model (Fig. 5 and 6), to examine the cross ignition and computing the CIT of an individual burner (see next section).

A recent study of Yasamin Khazraii [6] have implemented a CFD- simulations for computing the concentrations of NO_x emission during combustion processes of a liquid fuel spray flames, investigated at an experimental furnace for liquid fuel. The concentrations had been simulated exclusively, due to the influence of the fuel spray angel and inlet air temperature. Their results had showed, the determination of temperature profiles in high gradient regions relative to the fuel spray angel only.

However, it is possible to enhance this determination, by varying several constructive parameters, that are in concern of the current case study. Since the investigated CI is carried out in a very complex area with high velocities, a numerical simulation (based on CFD program) of the proposed experimental model will be essential for the current study, (see the section of Numerical Analysis).

3.DESCRPTION OF THE EXPERIMENTAL TEST RIG

A simplified experimental rig is built up, consisting of two adjacent combustion chambers in an open housing, separated by a movable partition wall to vary the flow area between flame and mixture jet. And getting meanwhile, a varying mixing zone, so that the impact of the varied contact area on cross-ignition can be examined.

A cross- section of this model is shown in Fig. 5 and 6. Two burners of the company [RIELLO (P/ N 40) F20] are installed parallel in each chamber individually, and are driven by controlled and metered fuel supply.

Fig. 5: cross section of the test- set up

Fig. 6: cross section of the combustor housing

The two burner jets are flowing parallel in the Combustion Chamber Compartment (CCC). A special glass window is flanged at outlet of the chamber, allowing to view the propagation of the flame. At foreground of the window, a high-speed video camera (Fig. 5) with a high image frequency is installed, through which a CI can be observed between the burners, and also the expansion of flame and ignition of combustible mixture can be seen in the mixing zone.

An electrical schematic is installed at the experimental setup, which mainly consists of an ignition switch and a photo-diode with a digital light timer mounted at the outside wall of the CCC to monitor the process, and for determination of the total CIT.

The individual burner air/fuel mass flow ($\dot{m}_{air,i}$) and the total air mass flow ($\dot{m}_{air,tot}$), which enters the CCC are controlled and metered by a special installed flow meters.

The distance (b_i) between centre lines of the burners are varied by moving one of the two burners on the horizontal axis (Fig. 8).

4.IMPLEMENTED OPTICAL MEASUREMENTS

With the high speed video camera, used for recording of flame propagation and CI, the recorded images are saved. The entire cross-ignition process is then progressively classified and manipulated for each experimental procedure. Namely, for the total CIT, that entitled firstly the time from injection- start, up to the first appearance of small intermediate flash lights of a cross-ignition in the adjacent jet (Fig. 9). This time includes the physical reactions time ($CIT_{phys.}$),

Fig. 8: The distance b_i between the two burners (left), the variable flow area (right)

flash light up to the last big flash is related to the very fast chemical reactions (CIT_{chem}). Since the heat flux starts heating the injected cold mixture as soon as the injection process starts, until a big flash appears. The computation of CIT in the mathematical model are more concerned with CIT_{phys} , rather than the very small CIT_{chem} , which will not be in concern comparing to the main targeted time in this paper CIT_{phys} .

Fig. 9 shows high-speed images of a local cross-ignition, from the first small light of a cross-ignition in the CCC up to the last big flash light.

Fig. 9: Cross ignition steps show local ignitions through a flow area (A_i) = 20cm x 20cm and $S_i = 0.5$

These images show, how an ignition process starts from a low temperature, accumulating the transferred heat partially at tiny flammable parts of the turbulent cold mixture. Little flames appear at a time of ($t = 2.35$ s) and extinguished after a very small time ($t = 2.45$ s), losing their heat energy for activating and heating the surrounding mixture droplets, leading during a very tiny period of time to a temperature decrease, distinguished in the literature as the cool flame, see more in [7]. This internal accumulated heat exchange is normally continued due to the circulating effect in the mixing zone until the real ignition temperature is reached, and a big flash is occurred at a time of ($t = 2.58$ s).

5. GOVERNING MATHEMATICAL MODEL

Based on the above mentioned heat flux spreading into the second burner, the following heat balance [8] is applied on a determined control volume ($V_{cont.}$) along the mixing zone:

$$\dot{I}_{tot.} = \dot{I}_{trans.} + \dot{I}_{chem} - \dot{I}_{loss.}$$

According to Newton's law of heat transfer and the theory of the Semenov explosion [9], the amount of the generated heat is then proportional to the temperature difference between ignition temperature (T_{ig}) and initial temperature (T_0) of the fresh mixture:

$$\dot{I}_{trans.} = \alpha_{tot.} A_{w.tot.} (T_{ig} - T_0)$$

$$\dot{I}_{los.} = \dot{m}_{exh.2} * C_{p_{exh.2}} * (T_{exh.2} - T_{amb.})$$

$$\dot{I}_{chem.} = (h_E - h_P) \rho_2 A \exp\left(-\frac{E}{RT_0}\right)$$

$$\Delta h = (h_E - h_P) = C_{p_{fuel}} * \dot{m}_{fuel.burned} * \Delta T$$

$$\int \rho_2 C_{p_2} dT / dt = V_{cont.} (\dot{I}_{rad.} + \dot{I}_{conv.} + \dot{I}_{diff.} + \dot{I}_{Chem.} - \dot{I}_{los.})$$

which results after the integration in the following main- equation:

$$\rho_2 C p_2 dT / \Delta t = V_{cont.} (\dot{I}_{rad.} + \dot{I}_{conv.} + \dot{I}_{diff.} + \dot{I}_{Chem.} - \dot{I}_{los.})$$

Whereas the term Δt is corresponded to the calculated CIT.

6.DISCUSSION OF RESULTS

Primary results were obtained, describing the interacting of the influence parameters on CIT in preliminary low velocities producing almost linier relations Fig. 10, 11 and 12.

The effect of the flow area between the two burners is in Fig. 10 illustrated. It shows, that by larger A_i , the exhaust gas mass flow rate will increase creating wider flame spreading, that minimizes the distance between flame- and mixture contours, where faster cross-ignition could be conducted in smaller period of time. this phenomena is obvious in the current case study, after the flow area A_i is opened wider than 0.07 m^2 .

This diagram shows also in the current operational condition a pretty linear relation, where the CIT is directly proportional to the size of the flow area A_i .

Fig. 11 shows the effect of different flame ϕ or (distances) on a cross-ignition. It can be seen, that by increasing the flame ϕ , the speed of exchange mass flow with the mixture in the adjacent burner is then increased due to the direct closer contact surfaces. Thus, the speed of the cross-ignition becomes very fast by wider contact of both surfaces, specially, after the flame ϕ is wider than 0.3 m .

Fig. 10: Effect of different flow areas (A_i) on a cross-ignition process at ($\delta_{oi} = 0.5$, $\Phi_i = 1:15$)

Fig. 11: Effect of different distances or (flame ϕ) on a cross-ignition at ($\delta_{oi} = 0.5$, $A_i = 20 \times 20\text{cm}$, $\Phi_i = 1:15$)

Fig. 12 shows that, while increasing of Reynolds numbers, an interaction between several parameters is occurred. Namely, the residence time is decreased due to the high axial velocity of the mixture, and hence the heat flux into this mixture is disaffected. Accordingly, it is noted in the diagram below, that the distance between the two lines is getting gradually more diversion by higher Re. Thus, a post investigations in this high turbulent region is very necessary.

Fig. 12: Effect of various Reynolds numbers on a cross-ignition at ($\delta_o = 0.5$, $A_i = 20 \times 20 \text{ cm}$, $\Phi = 1:15$)

These diagrams show that, the influence of the distance between adjacent burners, and also the flame ϕ , on the determination of CIT greater than the effect of Re and A_i . Moreover, they illustrate a reasonable coincidence between experimental and theoretical results, that entitled almost linear relations, where the CIT is directly proportional to the magnitude of the investigated influence parameters. This direct proportionality, is considered as a determined factor, while establishing of constructive criteria in certain velocity region. For simulating more critical

operational conditions, the computational model should be enhanced for nonlinear high gradient regions, where the implementation of CFD simulation [10] would be very beneficial and less dangerous.

7. NUMERICAL ANALYSIS

For solving the balance equations of the mathematical model in this study, a grid generation for the domain of the CCC by using a commercial CFD- Software is prerequisite. Since the test rig consists of two adjacent combustion chambers, separated by a movable partition wall, the entire test rig (Fig. 5), was divided into two domains, where the first mesh grid is generated for the lighted burner domain with several mesh configurations.

However, after analyzing of initial computations, it was clear that for more accurate results, the area near the entrance of the CCC, in the turbulent zones where the flow properties are changing rapidly and at outlets of the CCC, finer meshing with high number of nodes are selected as shown in Fig. 13.

Fig. 13: Computational mesh for the first burner domain

The generated mesh model is computed with the CFD- Program for getting the temperature profiles at the significant parts of the computed domain, for distinguishing the heat flux passing to the next adjacent burner, and to be compared with the measured experimental results. Specially, at the varied pass area A_i between the two burners, at the relevant walls and at the flame perimeter of the first burner domain.

Fig. 14, 15 and 16 show a computed temperature profile for the combustion of a kerosene - air - mixture at certain operational conditions (different inlet air velocities ($C_{air\ inlet}$)) in the first burner, illustrating the effect of mixture velocity increase on the temperature profile of the combustion process. The resulting relative temperatures of the computed profile are entitled in the balance equations of the mathematical model.

Thus, the numerical results of the CFD simulated domain, are considered as data inputs for computing the CIT in the second burner domain, through the above mentioned mathematical model. Whereas, these data inputs could be directly transferred by extending the numerical Analysis for the CI-Process in the next domain of the adjacent burner, see outlook.

Fig. 14: Computed $T_{\max} = 899$ k at ($C_{\text{air inlet}} = 2.5$ m/s, $\Phi = 1:15$, $A_i = 20 \times 20$ cm)

Fig. 15: Computed $T_{\max} = 918$ k at ($C_{\text{air inlet}} = 3$ m/s, $\Phi = 1:15$, $A_i = 20 \times 20$ cm)

Fig. 16: Computed $T_{\max} = 938$ k at ($C_{\text{air inlet}} = 3.5$ m/s, $\Phi = 1:15$, $A_i = 20 \times 20$ cm)

8.CONCLUSION AND OUTLOOK

For validation of the computational results, an experimental test rig was constructed for combustion of liquid fuel. Several operational tests have been carried out for a three of the influential parameters.

The experimental results obtained in this paper at certain regions, were in coincide with the corves of the computational model. These results have opened the door wide for implementing a CFD simulation for this study, separating the test rig into tow domains, starting a grid generation and simulation the process on the lightened burner, where the computational results are considered to be as data input for the adjacent domain.

For enhancing the study, a subroutine program should be developed to work out the data in/out put between the two domains of the adjacent burners in the test rig, where a direct comprehensive computation of the CIT can be conducted.

Due to analysis of the above mentioned three diagrams, a design concept of a movable device connecting between the three investigated parameters, can be constructed for controlling the CI during operation.

REFERENCES

- [1] Heinrich Ostenrath, Gasturbinen – Treibwerke, (1968), Verlag W. Girardet. Essen.
- [2] Jürgen Warnatz and Ulrich Maas, Verbrennung physikalischer- Chemiekalischer Grundlagen,(20011), Modellierung und Simulation, Experimente, Schadstoffentstehung, Springer Verlag.
- [3] M. Boileau and G. Staffelbach, (2008), LES of an ignition sequence in a gas turbine engine.
- [4] Martin Jordan, (1999), Zündung und Verbrennung Vorgemischtenturbulenten Freistrahles, Dissertation.
- [5] C. Hirich, (2002), An Annular Combustor Natural Gas Ignition Model Derived from Atmospheric Sector Experiments. ASME 2002.
- [6] Yasamin Khazraii, (2011), Numerical Simulation on NO_x Emission in Liquid Fuel Spray Flames.
- [7] Stefan Schnaubelt, (2005), Numerische Analyse des Selbstzündverhaltens einzelner Brennstofftropfen, Universität Bremen, Desertation.
- [8] M. Abu Amro1, A. Imara and S. Abu Naser. (2015), Design Criteria For Optimization of the Cross Ignition Process In Gas-Turbine-Engines, IJSCMC Vol 4, No 4.
- [9] Prof. Alfonsleben, Wärmetransport, Vorlesungsskript (I, II) – TU-Berlin, 1988.
- [10] Günther Clauss and Mazen Abu Amro, (2004), Two and Three-Phase Flow Computation For The Optimization Of Oil Skimming Systems, 3rd International Symposium on two-phase flow modeling and experimentation, Pisa, 2004.