MODELLING THE VAPORIZATION OF A MULTICOMPONENT FUEL IN A T-56 COMBUSTOR AT AMBIENT TEMPERATURE

A. Benaissa, T. Pringuey and W.D. Allan
Royal Military College of Canada, Kingston (Ontario) Canada K7K 7B4
Corresponding author: benaissa-a@rmc.ca

Keywords: modeling, evaporation, CFD, gas turbine

Abstract

The present work consists of a modeling exercise of the flow and droplet evaporation in a combustion chamber of a gas turbine engine. A numerical model is used to compare simplified geometries representing the T-56 combustor. A multicomponent fuel vaporization model is incorporated into a CFD code. The aerodynamic and fuel vaporization simulations are performed for two fuels JP-4 and Jet A-1. To validate the results of the aerodynamic simulation, experiments have been carried out to determine the velocity and turbulence intensity fields in the combustion chamber.

1 Introduction

The amount of energy necessary to ignite or reignite depends totally on the fuel vapour fraction existing in the combustion chamber near the ignitors. Hence, the understanding of the fuel vaporization [1,2] and the distribution of the vapour in the combustion chamber is critical. Thus, it is very important to model the combustor’s geometry accurately, in such a way as to predict the aerodynamic characteristics on which the vaporization rate of fuel droplets depends significantly. Moreover, modeling the fuel behaviour precisely is also essential.

To solve ignition or re-ignition problems, it is mandatory to improve the understanding of fuel volatility to enhance it. This can be done by introducing additives that extend the ignition temperature range of the air/fuel mixture. Since these complex fuels are made of blends of hundreds components, they cannot be considered as a simple component fuel. For a multicomponent fuels, the physical and thermal properties change continuously during fuel droplet vaporization. For instance, the vapour pressure depends not only on the temperature as for a single component fuel, but also on the mass fraction of the fuel vaporized.

The present work consisted firstly of an accurate modeling of the combustion chamber’s geometry. Then numerical calculations were performed using the CFD code FLUENT 6.1 to calculate the flow in the combustor without combustion. The calculations were carried out in two geometries to help validate a combustor with transparent walls for optical measurements. To validate the aerodynamic simulations, an experimental rig was used to carry out measurements of velocity magnitude and turbulence intensity in the combustor. The multicomponent fuel vaporization model was then implemented in the CFD code in C language. The results of evaporation simulations are presented together with a comparison with a single component model.

2 Modeling

2.1 Evaporation

The evaporation of a liquid spray is a common phenomenon in industrial applications. The evaporation process is always conditioned by the dynamics and the thermal conditions and in many cases, by fuel composition as well. In liquid-fuelled gas turbine engines, the fuel vapour present near the ignitor is critical to successful light-off. Correct fuel spray evaporation modeling is essential for the
accurate prediction of cold start performance or any other marginal ignition conditions. The complex evaporation behaviour of commonly used multicomponent fuel is still poorly understood, and the complex interaction of various parameters makes evaporation difficult to predict.

A major problem in predicting evaporation behaviour of multicomponent fuels is specifying the state of the liquid phase (mixing, temperature, species distribution and velocity field). Another major difficulty is the characterization of the gas phase around the droplets; this depends on the quantity and nature of the components already evaporated. In most situations at ambient temperature, transient effects due to rapid changes in droplet conditions (velocity and temperature) are potentially important for mass diffusion processes. Hence, because a complex fuel is composed from numerous chemicals with different volatilities (light and heavy components), it is important to pay more attention to the evaporation of multicomponent fuels. This is very important during the transient mode while velocity, temperature and physical properties are changing. The transient mode is critical because of the presence of light components in the fuels and the existence of a high forced convection transfer due to changes in the droplet velocities from injection values until they reach the flow velocity.

For typical multicomponent fuels, the boiling temperature increases as vaporization proceeds. Vaporization of the lighter components occurs first, followed by heavier components. The lighter the compound, the lower is its individual boiling temperature.

2.2 Multicomponent Evaporation Approach

In addition to the changes in overall mixture accounting for vapour pressure as evaporation proceeds, the multicomponent volatility approach also provides instantaneous enthalpy of vaporization and vapour-phase molecular weight as an explicit algebraic functions of the mass fraction of the fuel vaporized.

The main objective of this approach is to simulate the evaporation of a multicomponent fuel like JP-4 or Jet A-1. It is about monitoring the changes in the physical properties of the fuel as the vaporization process occurs. A relatively simple model is used based on spherical symmetrical droplet vaporization with an infinite diffusion in the droplet. The model assumes a good mixing of the liquid phase with a uniform temperature, thus the effect of the concentration gradients in the liquid phase is neglected. The droplet temperature and composition are maintained spatially uniform, although they may vary temporally. This infinite-diffusion model has been tested on a two component fuel droplet evaporating in a heated air flow. Good agreement with experimental data for relatively low temperature conditions was observed.

Equation (1) is known as the Clausius-Clapeyron equation.

\[ p_{sat}(T_w) = f \exp \left( - \frac{E}{T_w} \right) \quad (1) \]

For single-component fuels, variables \( f_1 \) and \( f_2 \) are constant as evaporation proceeds. For the present purpose of a multicomponent fuel evaporation model, the variables, \( f_1 \) and \( f_2 \) depend on the mass fraction of fuel vaporized. The approach of the multicomponent fuel volatility has been validated against experimental data \[3\]. Polynomial equations and coefficients are used as parameters for the varying physical properties of JP-4 and Jet A-1 in the multicomponent fuel evaporation model. Functions for these parameters for JP-4 and Jet A-1 have been approximated using Legendre polynomials. This approach was used in the CFD code as in \[4\].

2.3 Flow Modelling

The T56 Combustor: The combustor modeled is on the Allison T-56 turboprop engine manufactured by Rolls Royce. This engine powers the military transport aircraft Hercules C-130 and CP-140 Aurora Maritime Patrol aircraft, both manufactured by Lockheed Martin.
The experimental test section in Figure 1 shows a transparent annular section housing a can-style combustion chamber (a) and a view looking upstream from the combustion chamber exit towards (b) with the swirler. The configuration represents the real engine and will be compared with simpler geometry where the real liner is replaced with a Pyrex tube for optical access. The idea is to model the main characteristics of the flow.

![Fig. 1: Pictures of the combustor side view (a) and face view (b)](image)

### 2.3.1 Modeling Geometry

This modeling was realised with the software GAMBIT 2.1. Figure 2 shows the model from upstream. It represents the external domain of calculations. The internal can was modeled with two configurations; the first is called the “real combustor model” (Figure 3(a)) which reproduces the real size of the perforations in the combustion chamber walls. The second one, called the “Pyrex can” has smaller radial holes in the walls (Figure 3(b)).

![Fig. 2: Geometry of the combustor](image)

Both geometries were modeled to study the influence of the size of the side holes on the flow field characteristics.

![Real combustor model (a)](image)

![Pyrex combustor model (b)](image)

![Fig. 3. Comparison between the two geometries](image)
2.3.2 Modeling Swirl

Figure 4(a) shows the inside face of the swirler looking upstream. Figure 4(b) shows how the eight internal sectors were reproduced to generate swirl at the entrance of the can. Both cans were modeled similarly.

![Real Model Comparison](image)

3. Results

3.1 Aero-simulations:

Figures 5 and 6 allow one to compare velocity magnitude for the two configurations modeled. The recirculation zone seems to be larger in the real combustor that in the Pyrex one; this is due to the higher flow rate from the side holes that block the main flow creating recirculation.

![Velocity Magnitude in the Pyrex Combustor](image)

On the axial component of velocity represented in Figures 7 and 8, a higher velocity near the side holes for the real can can be seen. Recirculation zones were apparent in both cans with a larger zone for the real can. Also, for this can, the recirculation zone had more influence on the central part of the flow producing more mixing in the primary zone of the combustor.

![Axial Velocity in the Pyrex Combustor](image)
On the tangential component of velocity represented for both cans in Figure 9 (Pyrex) and Figure 10 (Real), one can notice presence of swirl after the primary zone in the real can. For the Pyrex can, swirl is present just in the primary zone. Since the primary zone is the area where the spray and evaporation occur, one can consider that the two geometries gave similar swirl.

Another parameter to be considered in comparing the two models is turbulence intensity. Figures 11 and 12 represent turbulence intensity for the two configurations displaying a similar overall behaviour.

The radial representations of the tangential velocity (Figure 13) for both cans, confirmed the previous observations about the presence and the intensity of the swirl further downstream for the real can. With these views one can see also that the swirl occupied almost all the flow domain, whereas for the Pyrex can the swirl was mainly concentrated near the wall.
As a result of the higher mixing in the real combustor, turbulence intensity was well distributed after 0.7 diameters downstream (Figure 14). Compared with the Pyrex combustor, these figures confirm that the higher side flow contributed to a better mixing in the primary zone of the combustor.

As a conclusion of this study, it is clear that there was a difference in the flow patterns between the two configurations and it will be necessary to increase the size of the side holes in order for the Pyrex can to adequately represent the flow in the combustor. The study will continue with the real can and experimental validations will follow.

3.2 Experimental Validation

Since the calculations were to be carried out in the simplified model of the engine can-style combustion chamber, experimental validation was necessary. A rig was built using the real can of a T-56 engine and a measurement system was used to carry out a grid of measurements on different radial plans of the can. A blower was used upstream the can to generate airflow at 0.13kg/s at ambient temperature (300K). The flow that entered the real combustor simulated the real flow situation inside the combustor (Figure 15).
Velocity and turbulence intensity were measured with hot wire anemometry 56C17 and 55P11 probes from DANTEC. Two axial sections were considered, one at \( x = 165 \) mm from the can inlet, the other at \( x = 200 \) mm. The hot wire probe was moved in the radial plan using two translations, in two orthogonal directions. The probe was moved manually in the axial direction. LABVIEW was used to realize the probe motion and data acquisition was performed by an ADC 488/8SA system from IOTECH.

The measurements were carried out on the central part of the can on a 80 mm square centered on the axial axis. A grid step of 20 mm was used in both directions. Experimental results were compared with the calculations shown in Figure 16. On the calculation plot results, the square shows the experimental domain.

This figure compares the velocity magnitude at \( x = 200 \) mm downstream. The calculations represent the entire radial domain while the experiments are centered on the central part where all the mixing occurs. Similarly Figure 17 compares turbulence intensity predicted with the measured ones at the same location.
3.3 Evaporation Results

For evaporation, simulations were carried out only for JP-4 fuel. Calculations were conducted in a 2-way coupling; mass and heat transfer between the droplets and the air flow were considered. The minimum and maximum diameter of droplets were 5 µm and 152 µm, respectively. Fuel temperature at the inlet was 300 K and its flow rate of 5 g/s was comparable with the real case. The droplets were injected at a frequency of 7.5 Hz and at a speed of 1 m/s.

No interaction between the droplets was considered. To compare the evaporation model inserted in the CFD code with the standard one supplied with the code, iso-surfaces of mass fraction of C\textsubscript{12}H\textsubscript{23} were calculated with the two models. At t = 30s, FLUENT’s vaporization model gave a maximal mass fraction of C\textsubscript{12}H\textsubscript{23} equal to unity indicating full evaporation, whereas the multicomponent model predicts a maximal mass fraction of 0.00294. To compare the iso-surfaces from both models, it was decided to display only one quarter of the domain. The quarter, the half and the three-quarter of the maximal mass fraction for each model are displayed below at t = 30s from the injection time.
These results show that the standard vaporization model overestimated the evaporation compared with the multi-component model.

The simulations can follow a value of mass fraction of fuel in time. Iso-surfaces of C_{12}H_{23} can be plotted at different instants. Since the maximal value of the mass fraction at t = 30s was 0.00294. The mass fraction iso-surfaces of C_{12}H_{23} were studied at three values: 0.000735, 0.00147, 0.00225, corresponding respectively to the quarter, the half, and three-quarter of the maximal mass fraction at t = 30s. In this paper only 0.00225 mass fraction will be presented.

These time evolutions represent the spatial distribution of fuel vapour in the combustor. This gives an approximate idea on how the fuel is mixed in the flow. At this time studies are being carried out to link these views with the time steps of the flow. More in depth analysis of vapour mixing will be presented at the conference.
4 Conclusions

The present work is part of an overall effort to explore benefits and disadvantages of fuel additives for JP-8. The objective is to improve engine ignition at low temperature and reignition at high altitude. The objective of the research was to improve evaporation modeling by using an approach that takes into account its multicomponent nature. To enable this work, modeling of a T-56 combustor geometry was realized with a CFD code. This allowed the study of the internal aerodynamics of the combustor and the construction of a Pyrex test section to reproduce the main dynamic features of a real can-style combustion chamber. Experimental validation of the aerodynamic calculation were carried out and the Pyrex can was built. Spray studies with and without combustion were completed [5]. A multicomponent vaporization model was implemented in the commercial code, FLUENT, and first simulations for JP-4 showed that the model improved the standard evaporation calculation in the CFD code. By analyzing distributions of mass fraction of fuel, the importance of an accurate modeling of the entire flow pattern in the combustor was highlighted. To enhance this work, experimental validation of the vaporization simulation will follow and JP-8 fuel will be used and compared to JP-4. The final step of this project will include the addition of an ignition model simulation, and comparison of the results with experiments. This will contribute to the design of an additive for JP-8 to enable improved relight performance.

References