Conference Paper

Comparison of Pollutants Formulation Prediction Using Several Turbulence Models for a CFM56-3 Combustor

Harsh Hansraj and Francisco Brójo

Universidade da Beira Interior

Abstract

Nowadays the CFM56 engine is one of the most widely used engine models in the aviation industry. With this work it is intended to analyse several turbulence models during the combustion, allowing a better understanding of some problems and their possible resolution. It was used a STL file of the combustion chamber digitization, from the work of Oliveira. In the numerical case only a quarter of the combustion chamber is used due to its symmetry allowing a less computational effort during the simulations and the fuel used in combustion is Jet-A. The mesh used was designed in Helyx OS software and numerical simulations were performed in ANSYS Fluent 16.2. The models k-ϵ, k-ω, RSM and LES are analysed and in the latter, the initial conditions resulting from the k-ϵ model are used. The results obtained show reasonable agreement with some experimental reference data present in the ICAO Emissions Data Base. Among the analysed models it was observed that in general, despite its high computational cost, the LES model is the one that best identifies the various zones of the combustion chamber. However, the RSM and k-ϵ models proved to be very useful in observing the emission distribution of some gases during combustion. It is concluded that the LES model gives the best results, but the choice of the most suitable model may vary depending on the boundary conditions and flow type of the case study to be analysed.

Keywords: CFM, Combustion chamber, Turbulence models

Corresponding Author:
Harsh Hansraj
harshansraj_12@hotmail.com

Received: 26 November 2019
Accepted: 13 May 2020
Published: 2 June 2020

Publishing services provided by Knowledge E

Selection and Peer-review under the responsibility of the ICEUBI2019 Conference Committee.

1. Introduction

The engine of an aircraft is one of the fundamental components that moves the aircraft through the production of propulsive power. Since the first attempts to build a combustor chamber by Frank Whittle in the decade of 1950 [1], the combustion chambers have been developing gradually and efficiently. One of the great achievements in history happened when General Electrics and Safran Aircraft Engines developed in partnership the CFM engine company. The engines produced by CFM revealed to be very efficient in terms of emissions and had a very desirable performance for that time, which made aircraft producers like Boeing and Airbus to invest in this engine for their aircrafts. The first derivative of the CFM56 series, the CFM56-3 was designed for Boeing 737 Classic

How to cite this article: Harsh Hansraj and Francisco Brójo, (2020), “Comparison of Pollutants Formulation Prediction Using Several Turbulence Models for a CFM56-3 Combustor” in *International Congress on Engineering — Engineering for Evolution*, KnE Engineering, pages 465–475. DOI 10.18502/keg.v5i6.7062

series (737-300/-400/-500), with static thrust ratings from 18,500 to 23,500 lbf (82.3 to 105 kN). This model is a dual rotor, axial flow, high bypass ratio turbofan engine which has the following characteristics [2]:

- single stage fan, 3-stage low pressure compressor (LPC), 9-stage high pressure compressor (HPC);
- annular combustion chamber;
- single stage high pressure turbine (HPT), 4-stage low pressure turbine (LPT);
- hydro-mechanical main engine control (MEC) with limited authority electronic power management control (PMC).

Combustor chamber is where the combustion takes place and generates the propulsive power for the aircraft. The combustor should be capable to sustain a stable burn for long periods of time during the lifetime of the engine and requires high values of combustion efficiency. They also need to assure the re-ignition of the flame in case of a flame extinction during flight. Combustion can be described as the exothermic reaction of a fuel and an oxidant [1]. In other words, the goal of the combustion is to convert chemical energy in thermal energy leading to a temperature increment through the efficient burn of fuel. The liquid fuel is separated, by a spray, in small drops that mix with air, then the heavier hydrocarbons are divided in lighter ones in order to react with the oxygen triggering the chemical reaction. For a rapid combustion it is necessary a through mixture of fuel and air in the combustor. The combustion has 2 important regimes: deflagration and detonation.

Computational Fluid Dynamics (CFD) solves fluid flow problems coupled with heat and mass transfers in a given geometry by means of a mesh where all the Navier-Stokes transport equations are solved [3]. It is a flow analysis tool that allows the study of various flows and phenomena applied to diverse areas of engineering as aeronautical, physics, biomedical, etc. The work principle of CFD is to solve systems of differential equations, as the Navier-Stokes equations in complex geometries. The turbulence models used in CFD in the past years are Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds Average Navier- Stokes (RANS). The models used in this study were: k-ϵ, k-ω, Reynolds Stress Model (RSM) and Large Eddy Simulation (LES). The k-ϵ model is the simplest 2 equation model used in turbulent flow analysis. This model solves the turbulent kinetic energy, k and the kinetic energy dissipation rate, e equations which are:

- Turbulent kinetic energy, k

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\epsilon-Y_M+S_k \tag{1}$$

- Dissipation rate, ϵ

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_i}(\rho\epsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+c_{1\epsilon}\frac{\epsilon}{k}\left(G_k+C_{3\epsilon}G_b\right)-C_{2\epsilon}\rho\frac{\epsilon^2}{k}+S_\epsilon \tag{2}$$

Where G_k is the turbulent kinetic energy related to the velocity gradients and G_b refers to the turbulent kinetic energy due to bouyancy. Y_M is the compressible dilatation turbulence variation and the constants $C_{1\epsilon}, C_{2\epsilon}, C_{3\epsilon}, \sigma_k$ e σ_ϵ are 1.44, 1.92, 0.09, 1.0 e 1.3, respectively. The turbulent viscosity presented in the above expressions is the relation between k and ε given by:

$$\mu_t=\rho C_\mu\frac{k^2}{\epsilon} \tag{3}$$

Where C$_\mu$ is a constant.

The k-w model solves the same equation for k replacing the equation for kinetic energy dissipation by the turbulent kinetic energy specific dissipation rate, ω which is [4]:

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_i}(\rho\omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega\frac{\partial \omega}{\partial x_j}\right)+G_\omega-Y_\omega+S_\omega \tag{4}$$

Where G_ω represents the generation of ω, Γ_ω the effective diffusivity of ω, Y_ω the dissipation of ω due to turbulence and S_ω is the user-defined source term.

The RSM model is more complex than the 2 models above as it solves 7 transport equations along with dissipation rate equation. The Reynolds Stress transport equation includes turbulent diffusion, molecular diffusion, shear stress production, buoyancy production, pressure tension, production by rotating system and dissipation as described by H. K. Versteeg and W. Malalasekera [3].

The LES model has been used to better understand some properties of turbulent flows and provide more details about this type of flow. In this model the large eddies are solved numerically, and the small eddies are modelled by the Sub-Grid Scales (SGS) which increases the computational power required for the simulation. The main equations that are used to solve this model are described by H. K. Versteeg and W. Malalasekera [3].

2. Case Study

The CAD model of the combustion chamber used in this study was from Oliveira work [5]. The combustor has a length of 2430 mm, it is 2000 mm wide and has 2160 mm depth [2]. A good meshing process is fundamental to obtain a successful converged solution during the simulations and for that was used the HelyxOS software which is an Open-Source of graphical user interface (GUI) developed by ENGYS to run over OpenFOAM code. This is a user-friendly software which simplifies the meshing process and turbulence models resolution. Firstly, the CAD components of the combustor were converted in.STL files with the Blender program in order to import geometry to HelyxOS. After importing the geometry and setting the units to millimetres, it was defined the "base mesh spacing" to 0.009 as it was verified that decreasing this value it took longer to generate the and its characteristics would remain unchanged. The next step was to define the refinement levels, the zone type, the number of layers and its thickness. For the stretching rate and the minimum layer thickness there were used the default values. Both the refinement level and the number of layers was defined as 6 for the swirlers and the fuel injectors and for the rest of components the refinement level was 4 and 3 layers. The final layer thickness was in the range of 0.04 to 0.06 according to the complexity of the component. Finally, the material point needs to be located inside the geometry and it was defined as -0.1306;0.0911;0.0253. The mesh generation process took around 1 hour and the mesh created had 2353474 cells, 8739826 faces and 4093842 points.

The simulation setup and configuration were made in ANSYS Fluent 16.2 and was selected the 3D option along with the double precision and parallel processing with 8 processors. After uploading the mesh and through the "Report Mesh" tool it was possible to know some mesh characteristics such as minimum orthogonal quality, maximum orthogonal skewness and maximum aspect ratio which were 0.8165, 0.1835 and 1.732 respectively. Following the mesh upload, was necessary to select the adequate models for the simulation desired as there is a great diversity of models for various types of problems. The models used in this study were:

- Energy Model – defines the energy and heat transfer parameters in the turbulence model;
- Viscous Model – it is selected the turbulence model used in the analysis (k-ϵ, k-ω, RSM, LES) and the constant values are kept the pre-defined;
- Species – It was selected non-premixed combustion, it was selected non-adiabatic energy treatment, the values of the pressure and the Fuel Stream Rich Flammability Limit for an engine working at 100% were inserted (2343346Pa and 0.0748 respectively), the concentration of the oxidant, which was considered air composed by oxygen and nitrogen, was set as 0.78992 and 0.21008 and

the Jet-A fuel Flash Point temperature was inserted (312K). After this setup the Probability Density Function (PDF) table could be created;

- NO$_x$ prediction –was added the Jet-A fuel and then activated the Thermal and Prompt NO$_x$ parameters in the Formation section. In the thermal tab was selected partial equilibrium regarding the O model. Then, was defined the Fuel Carbon Number and selected Temperature in turbulence interaction method.

The main section of the case setup is the “Boundary Conditions” as it is fundamental to define the mass flow inlets, pressure outlets and the walls. Moreira [5] has calculated in his work the total air flow rate (AFR), $\dot{m}_a$ needed for the CFM56-3 engine combustion as well as the $\dot{m}_f$ and these values were divided by 4 and used in the present work. The AFR obtained by Moreira [6] was 10.36 kg/s. In the Solution Methods was selected the Coupled scheme as this is a robust method for this kind of problems. The spatial discretization for the gradient selected was the Least Squares Cell Based, PRESTO! for pressure and Second Order Upwind for the rest of the parameters. There were defined monitors for mass imbalance and fuel mass fraction in order to verify the solution convergence. Finally, in the initialization was defined 5000 iterations for the k-ϵ, k-ω and RSM models and for the LES 1800 time steps with 0.00823 as a time step size.

3. Results

3.1. Validation

The results achieved in the simulations executed are compared to reference values which were experimentally measured and presented by ICAO in the emissions data sheet of the CFM56-3 engine (Table 1)[7]. The results obtained are in agreement with the reference values presented in table 2, except for the LES model which can be explained by the fact that the fuel atomization is not being considered, allowing its validation and further parameters analysis. Another reference value used was from Moreira work [5] related with average exit temperature from the combustor presented in table 3. The results presented in this study can be validated to Moreira results that reported a value of, approximately,1650K. The predicted temperature distribution and contours by the models analysed are presented in Figure 1.

Table 1: ICAO Emission data sheet for CFM56-3 engine.

Mode	Power setting	Time	Fuel flow		EI (g/kg)		SN
	(%F00)	(mins.)	(kg/s)	UHC	CO	NO_x	
Take-of	100	0,7	0,946	0,04	0,9	17,3	4
Climb out	85	2,2	0,792	0,05	0,95	15,5	2,5

Table 2: NO_x Emission Index results and reference value.

Model	NO_x [g/kg]
ICAO	17,3
k-ϵ	16,91
k-ω	13,53
RSM	16,9
LES	6,8

Table 3: Average temperature at the combustor exit zone.

Model	Average temperature at the combustor exit zone [K]
Pedro Ribeiro	1650
k-ϵ	1705
k-ω	1659
RSM	1664
LES	1650

3.2. Velocity magnitude analysis

The velocity magnitude contours are presented in Figure 2. In order to make this analysis more understandable the original scale values were reduced to a maximum of 400m/s since higher velocities are verified in the injector zone which is not the main point in this study. It is observed that the k-ϵ model does not display the higher velocities right after the injector while in the LES model distribution they are perfectly visible presenting a larger velocity zone above 400m/s. Comparing the 4 models it is noted that the LES study identifies, as expected, more clearly and accurately the different velocity variations in the combustion chamber.

3.3. Emissions analysis

Here were analysed the models' behaviour concerning NO_x, CO and CO_2 emissions during the combustion as these are the main pollutant gases produced. This a very important study since the pollutant gases produced are regulated by ICAO and that it is necessary to certify that the emissions do not exceed the maximum limit. In Figure 3 is observed the distribution contours of NO_x mass fraction in a plane normal to the injector. In the k-ϵ model the far field zones from the wall are presented precisely and accurately giving a good prediction of the NO_x distribution with a maximum of 0.00095 that is represented in red. On the other hand, the k-ω and the RSM models only present some zones of NO_x and the LES model is almost incapable of capture the higher density zones of this pollutant.

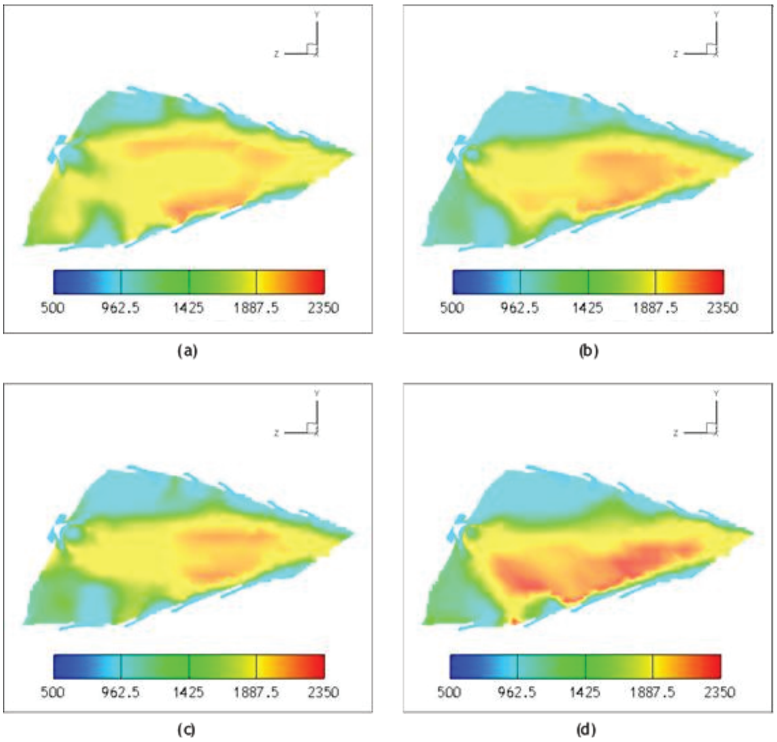


Figure 1: Contours of temperature distribution in a plane normal to the injector of the combustion chamber for a) k-ϵ, b) k-ω, c) RSM, d) LES.

Regarding the CO emissions presented in Figure 4 the results does not agree with the reference values. After searching for similar cases and errors it was found a project from the European commission [8] with difficulties in identifying the CO emissions in their engine. It was concluded in the project that although these models predict in a good way the NO_x emissions, they are not capable to identify the exchanges between NO_x and other pollutants like CO and UHC. In terms of the quality of the results presented, the LES model present better results, as expected, comparing to the k-ϵ model where

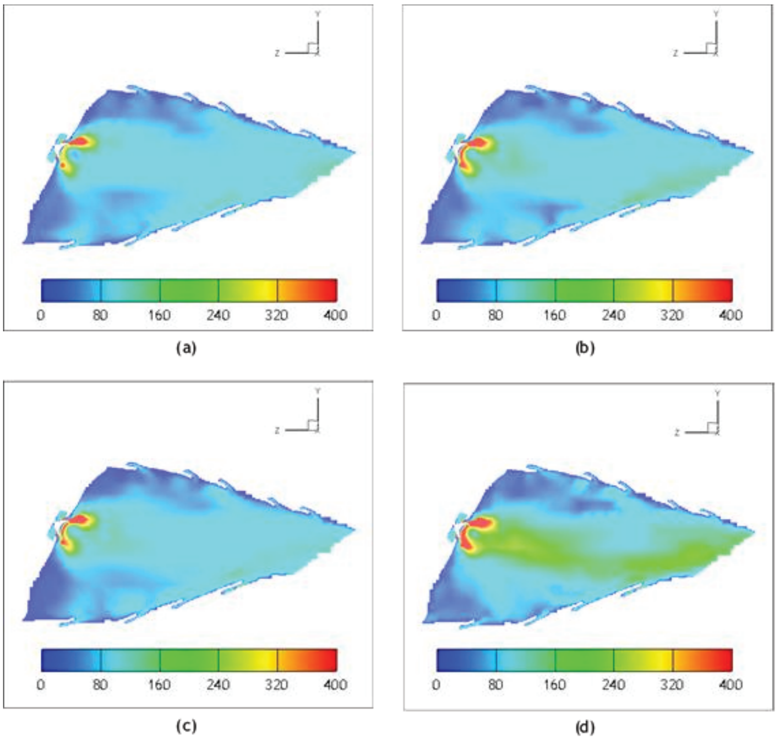


Figure 2: Contours of velocity magnitude distribution in a plane normal to the injector of the combustion chamber for a) k-ϵ, b) k-ω, c) RSM, d) LES.

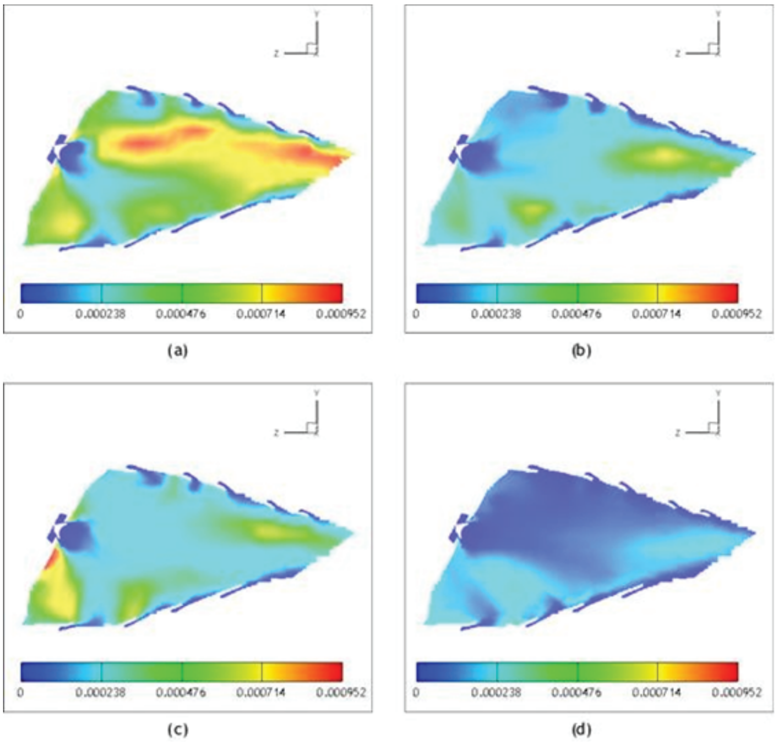


Figure 3: Contours of NO_x mass fraction distribution in a plane normal to the injector of the combustion chamber for a) k-ϵ, b) k-ω, c) RSM, d) LES.

the mass fraction zones are partially shown. The RSM and k-ω present less detailed results.

The CO_2 analysis has great importance as it is a greenhouse gas and nowadays it is necessary to minimize its emissions. Figure 5 shows the mass fraction distribution of CO_2 in a plane normal to the combustion chamber's injector. Comparing all the models it can be concluded that the LES model is the only one that identifies a red spot of CO_2 concentration while the other models could not.

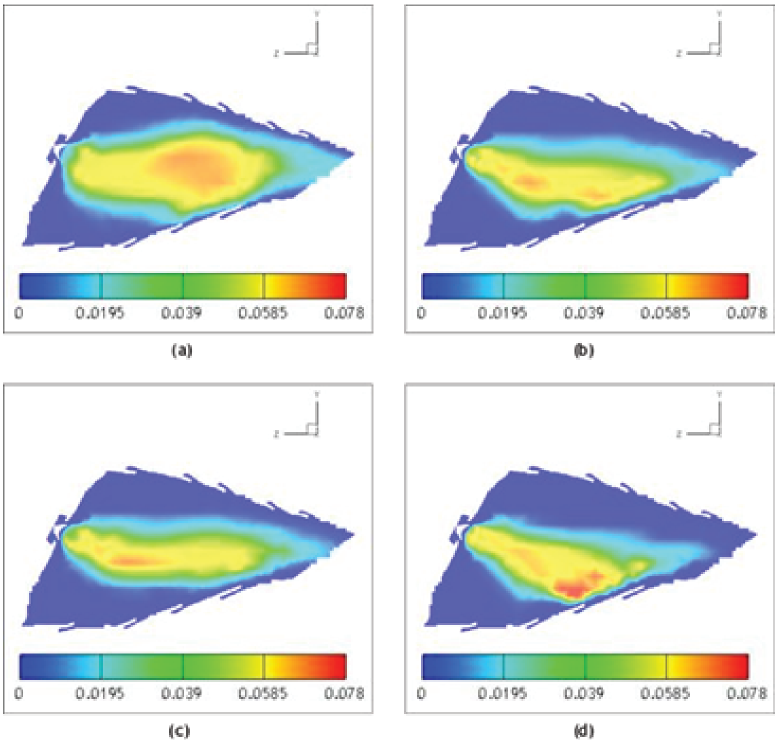
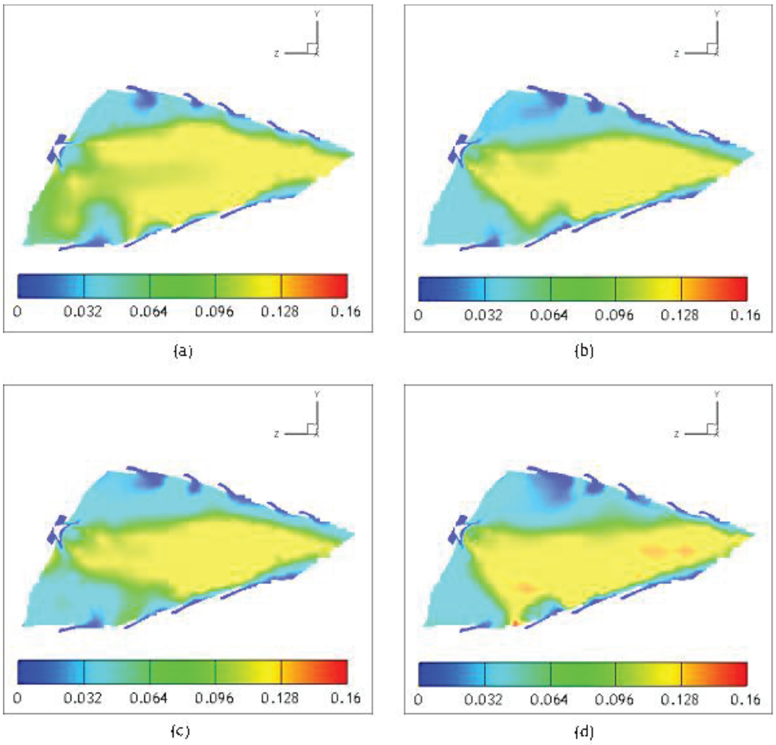


Figure 4: Contours of CO mass fraction distribution in a plane normal to the injector of the combustion chamber for a) k-e, b) k-w, c) RSM, d) LES.

4. Conclusion

With this study it can be concluded that the LES model predicts, with good accuracy, various parameters during the combustion of the CFM56-3 engine. It was possible to understand some differences between the 4 models analysed like there are some situations in which is preferable to use a simpler model that can give good results using less computational power than the LES model.



(a) (b) (c) (d)

Figure 5: Contours of CO2 mass fraction distribution in a plane normal to the injector of the combustion chamber for a) k-ϵ, b) k-ω, c) RSM, d) LES.

According to the validation of ICAO's reference values and Moreira work, it was proved that the LES model gives good predictions for temperature, velocity, CO and CO_2 parameters while the k-ϵ model makes a good estimation of the NO_x parameter.

Acknowledgments

The current study was funded in part by *Fundação para a Ciência e Tecnologia* (FCT), under project UID/EMS/00151/2013 C-MAST, with reference POCI-01-0145-FEDER-007718.

References

[1] A. H. Lefebvre and D. R. Ballal, *Gas Turbine Combustion: Alternative Fuels and Emissions, Third Edition*, Third Edit. 2010.

[2] E. A. S. Agency, "EASA TYPE-CERTIFICATE DATA SHEET," pp. 1–9, 2008.

[3] W. Malalasekera and H. K. Versteeg, *An Introduction to Computational Fluid Dynamics*, vol. M. 2007.

[4] A. F. ANSYS, *ANSYS Fluent Theory Guide*, no. November. 2013.

[5] J. Oliveira, “CFD Analysis of the Combustion of Bio-Derived Fuels in the CFM56-3 Combustor, MSc Thesis,” Universidade da Beira Interior, 2016.

[6] P. Coelho and M. Costa, *Combustão*. Lisboa: Orion, 2007.

[7] ICAO, “ENGINE EXHAUST EMISSIONS DATA BANK SUBSONIC ENGINES - CFM56-3,” 2015.

[8] T. Faravelli, A. Frassoldati, and A. Cuoci, “Final Report Summary - EMICOPTER (Emission analysis. Tools required to perform the emission analysis and evaluation methodology).”