Computational study of a micro-turbine engine combustor using large eddy simulation and Reynolds averaged turbulence models

C. A. Gonzalez[1] K. C. Wong[2] S. Armfield[3]

(Received 30 July 2007; revised 27 November 2007)

Abstract

A computational study of the combustion process inside micro-turbine engines is presented. Different turbulence models are assessed and results are compared against experimental data. Results indicate that RANS models and LES with dynamic Smagorinsky sub-grid model fail to achieve convergence or accurate solutions at the meshes employed in this study. Less accurate results are obtained for the DES when compared with LES-WALE. In terms of the combustion, the outlet flow presents temperature gradients that are likely to affect the turbine performance.

See http://anziamj.austms.org.au/ojs/index.php/ANZIAMJ/article/view/338 for this article, Published January 23, 2008. ISSN 1446-8735

Contents

1 Introduction

Micro-turbine engines have traditionally been developed for very specific flight applications such as hobby and target/military purposes. Due to the nature of such applications, fuel efficiency has been neglected in favour of affordability and ease of construction and running [4].

More recently, it has been recognized that these engines have applications in unmanned aerial vehicles or UAVs, small scale electricity generation and hybrid transport. All these three call for low fuel consumption, low pollutant emissions, high reliability and ease of manufacture.

One of the main components of these turbines is the combustion chamber. Several researchers studied the micro-turbine combustion [1, 2, 3, 4]; however, none of these studies is able to give fundamental data to develop new technologies for its optimisation.

The main contributions towards the understanding of the micro-turbine combustion come from combustors of full scale turbines and basic research in simplified burners. Very robust CFD and experimental methodologies led to more advanced research into alternative fuels [5, 6], high temperature materials that can withstand the increasingly adverse conditions [7, 8], and deeper understanding of the combustion behaviour and pollutant emissions [9, 10, 11, 12].

Computational fluid dynamics (CFD) is an attractive way to analyse this problem, not only for understanding the processes in the engine, but also for carrying out improvements and optimisations in terms of efficiency and pollutant emissions. However, no extensive work has been published using this tool.

The selection of the turbulence model is a primary task in every CFD computation. Traditionally, Reynolds averaged Navier–Stokes turbulence models (RANS) have been used as simplified engineering tools for combustion problems. Alternatively, as explained by Boudier et al. [9] and James et al. [13], large eddy simulation (LES) seems to be a most appropriate approach for modelling combustion inside complex geometries. This is because LES is able to reproduce more accurately the turbulence, which plays a major role in the combustion processes. The main drawback of LES, with respect to RANS, is the computational time since finer meshes and an unsteady solution are required. However, with the increase of computational power in the last few years, LES models have been more widely used in research applications.

Choosing a turbulence model not only depends on the relationship between computational expense and a better description of turbulence. In many cases, the use of a particular model may fail to converge, yielding unpredictable results. Therefore, it is of interest to determine which turbulence model can be useful for this specific application. Such a comparison has recently been performed for a full scale Turbomeca combustor [9], showing that LES outperforms RANS in terms of temperature profiles at the outlet when compared with the experimental results.

The combustion inside a micro-turbine combustor is simulated here with the main objective of assessing the performance of different turbulence models. Convergence, mesh independence convergence and comparison with experiments are carried out for the following models:

- Standard k-ϵ (RANS),
- Realisable k-ϵ (RANS),
- Shear stress transport (SST) k-ω (RANS),
- LES and SST k-ω, also called detached eddy simulation (DES),
- LES with dynamic Smagorinsky sub-grid model,
- LES with wall adapted local eddy viscosity (WALE) sub-grid model.

Using these results, the combustion behaviour inside the engine is analysed based on overall thermodynamic variables such as velocity and temperature. More complex data such as pollutant or noise emissions are not yet considered mainly due to the current limitation of experimental data for comparison.

2 Methodology

2.1 Turbine and combustor and mesh details

The case considered here is the KJ66 micro-turbine [14]. This turbine has been created for small aircraft propulsion and is specially designed for easy manufacture. It is readily available and there is plenty of empirical information, making it ideal for this research [4, 14].

The KJ66 combustor features direct injection of the fuel with six vaporising sticks for achieving complete combustion before the turbine stage. Typical Reynolds numbers at its inlet are around 54,000. A diagram of this component is shown in Figure 1. This is a 60 degrees section cut, just like that employed for the mesh as explained in the following paragraphs. The fuel injection is carried out through a 0.7 mm diameter nozzle using a standard 12 V pump.

The combustor was represented with a 3D, 60 degrees section non conformal mesh. It was meshed with a mixture of tetrahedral and hexahedral elements: Tetrahedra in the inner part of the combustor and hexahedra in the vaporiser and the outer sections. Only a sixth part of the engine was meshed to reduce computational cost. Several different meshes were evaluated from 130,000 to approximately 500,000 elements.

2.2 Turbulence, fuel injection and combustion models

Modelling turbulence means that the Navier–Stokes equations are filtered to reduce computational cost. In the RANS models a time average or an ensemble average is performed and turbulent structures are modelled. In LES models only the scales below a specified filter length (sub-grid filter) are modelled. As a consequence, a better description of the flow is achieved using LES. A detailed explanation of these models is beyond the scope of this paper but plenty of information about them is available [15, 16].

As explained in Section 1, this study comprises six different turbulence models. The steady formulation was applied for the RANS models, while for the DES and LES were solved with the unsteady formulation. Since the interaction with the walls is of importance, enhanced wall functions were used in all the RANS models.

The spray injection was described with a Lagrangian model using an unsteady stochastic discrete particle tracking approach [16, 17]. The aero-

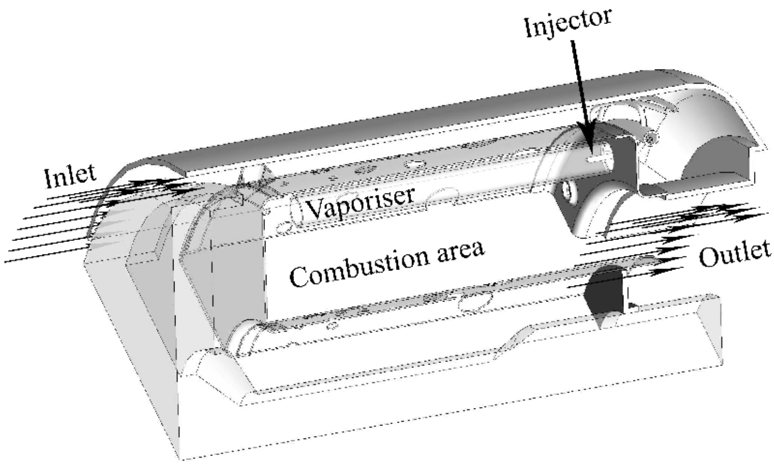

Figure 1: 60 degrees section cut of the combustor.

dynamic interaction of the droplets was computed with the dynamic drag model [18]. Droplet break up was modelled using a hybrid Kelvin–Helmholtz, Rayleigh–Taylor (KHRT) breakup model [19]. The model variables and constants were adjusted to match experimental results of Yang and Chin [20, 21] and those illustrated by Lefevbre [22].

The combustion was simulated with the steady flamelet model [23, 24] together with a reduced mechanism of 63 species and 167 reactions [25]. Kerosene was represented with a surrogate fuel consisting of 80% n-decane and 20% toluene.

2.3 Operating parameters and boundary conditions

A high load condition was evaluated in this study. The total air flow is 0.22 kg/s, the inlet pressure is 2.2 bar and global air fuel ratio is 65 [14]. The air inlet temperature is 400 K while that of the fuel is 300 K.

The experimental data available at these conditions is given in terms of mean values at the outlet of the combustor [4]. A range of temperatures from 920 to 980 K has been reported. The pressure drop in this combustor is reported to be approximately 12%, which yields a pressure at the outlet of 2.04 bar. Using this experimental data, a velocity of 130 to 137 m/s is expected.

The commercial software FLUENT 6.3 solved the governing equations and other scalars in the turbulent flow. Second order discretisation was used for all equations. The second order implicit in time approach was employed for the unsteady formulation. Pressure and velocity were coupled via the Pressure-Implicit with Splitting of Operators (PISO) algorithm. Residual convergence for continuity, velocities, mixture fraction was considered acceptable below 10^{-3}, while that of the energy equation was 10^{-5}. The time step for unsteady cases was 2×10^{-5} s.

3 Grid dependence and model evaluation

Time averaged temperature and velocity at the outlet were used for determining the performance of the different models. A fully developed solution was used for unsteady cases. In Figure 2 a plot for these two variables against the different mesh sizes is displayed for the six turbulence models considered in this paper. Results for the finest grids for the standard k-ϵ and k-ω fall outside the plotting range because their iteration residuals did not converge.

Of all the RANS models, only the realizable k-ϵ achieved acceptable resid-

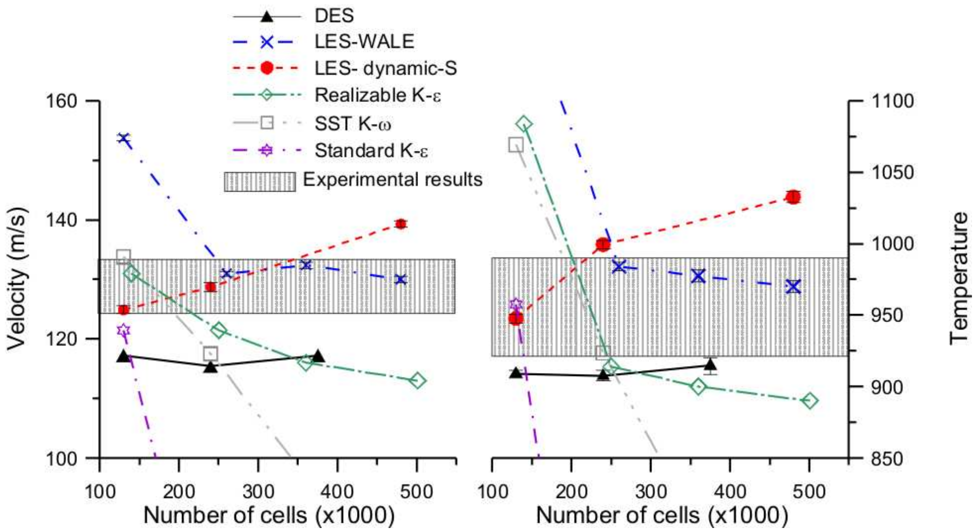

Figure 2: Mean mass flow rate and temperature at the outlet of the combustor for different turbulence models and different mesh sizes.

ual convergence of 10^{-3} or better in all grids tested. The remaining RANS models achieved at best 10^{-2} on the finest grid, which was not considered acceptable. The realizable k-ϵ achieved reasonable grid convergence but the results on the finest grid are far from the experimental results. As a result, none of the RANS models results are considered acceptable.

LES-WALE and DES models, achieved satisfactory residual convergence at each time step, and showed little grid dependence on the finest two grids. DES results were in poor agreement with the experimental results, whereas LES-WALE showed reasonable agreement.

LES with dynamic Smagorinsky sub-grid model had poor residual convergence at each time step (and therefore at some time steps the computed variables reported results far from reality) and was considered unsuitable for this flow.

As seen, LES-WALE is the most useful model for this problem and set-up showing good grid convergence and more accurate solutions. This is most likely because it features a specific sub-grid model that improves the predictions near the wall. Thanks to this sub-grid model, the results show that there is no need in this problem to use a finer discretisation at the wall boundaries (as normally required with other LES sub-grid models) in order to obtain relative good accuracy of the measured variables. Therefore, LES-WALE will be used for the results displayed in the following section.

4 Combustion results

The Reynolds averaged temperature and mixture fraction contours at the vaporiser-cut view are displayed in Figure 3.

In agreement with the theory, as observed in Figure 3(b), the section around the stoichiometric mixture fraction (obtained at 0.066) coincides appropriately with the high temperature region displayed in Figure 3(a). This high temperature area is also located behind the diffusion holes. This agrees with the experimental data in the sense that in these turbines conditions are regulated for the flame not to reach the turbine stage. The combustion is then carried out at stoichiometric or nearly stoichiometric mixtures. This explains the high temperatures in the combustion zone. They are likely to create high levels of nitrogen oxides and favour the production and destruction of soot.

A high level of turbulent mixing is carried out in the diffusion area, towards the outlet of the combustor. This makes it possible to achieve the relatively low temperatures obtained at the outlet (Figure 4). However, the temperature distribution at the combustor exit is far from homogeneous. As explained by Boudier et al. [9], radial and tangential temperature gradients at the outlet of a combustor are likely to negatively affect the performance of the turbine.

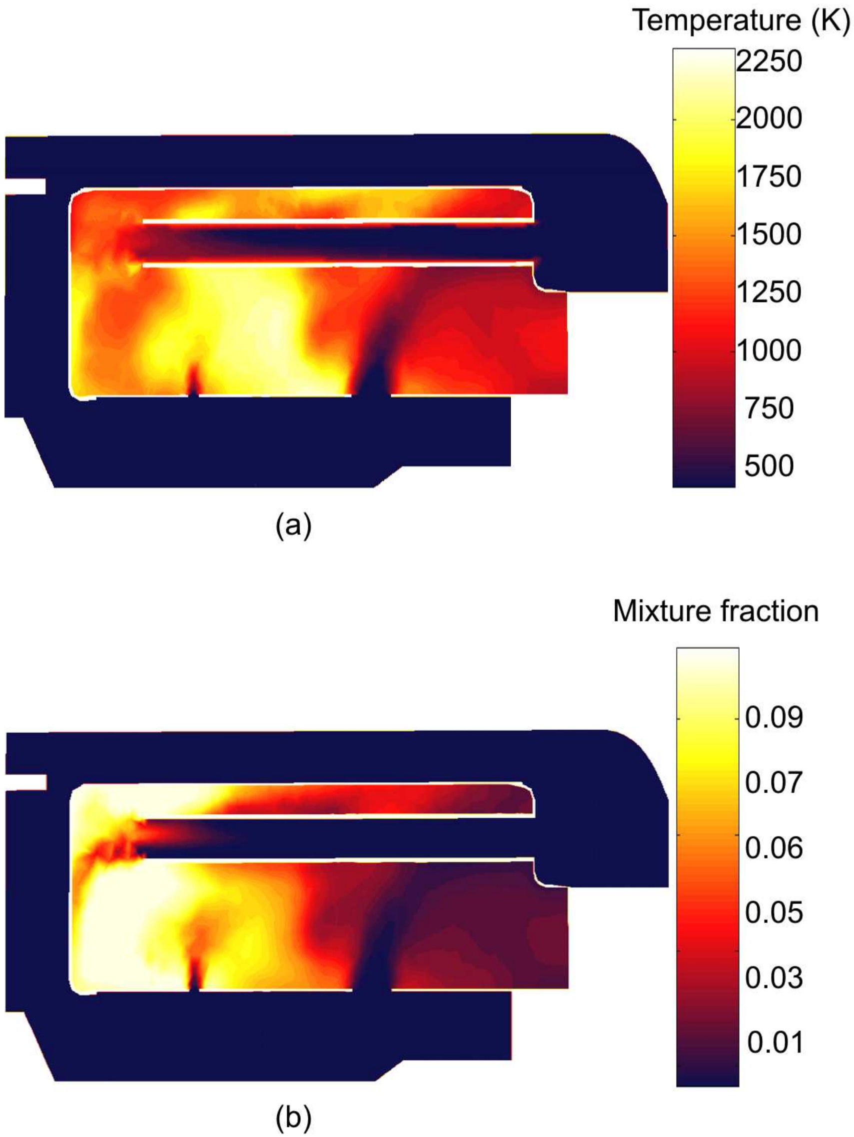

Figure 3: Temperature (a) and mixture fraction (b) results at the mid-section cut of the combustor.

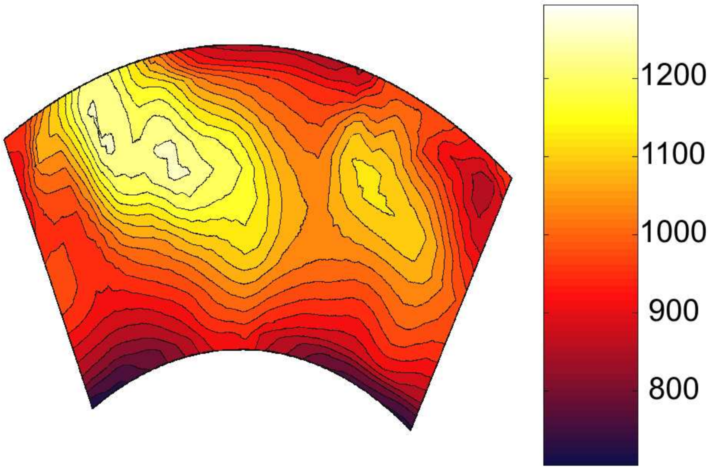

Figure 4: Reynolds averaged temperature contours at the outlet of the combustor.

The location of the diffusion holes so close to the outlet makes makes it impossible for the mixing process to produce a uniform temperature profile at the outlet. Increasing the primary and secondary air for the combustion to be carried out at leaner conditions can reduce the maximum temperatures in the combustion chamber and increase the mean global temperature and homogeneity at the outlet.

5 Conclusions and future work

Simulations with RANS models and LES with the dynamic Smagorinsky subgrid model fail to achieve a grid independent solution in the micro-turbine combustor simulation problem for the meshes employed in this study. Less accurate results were obtained for the DES when compared with LES-WALE. LES-WALE was found to be the most useful model for analysing this problem.

In terms of the combustion, in agreement with the experimental results at these conditions, the flame was found to be confined in the combustion chamber, rather than carried along to its outlet. However, temperature gradients were obtained at the outlet. These are likely to negatively affect the performance of the turbine.

Future work will focus on the the implementation of a radiation and soot model that, at lighter loads, can yield more realistic flame temperatures and the use of different meshing strategies in order to make it possible to achieve better convergence and more accurate results for RANS models.

Acknowledgements: The authors thank the Conselleria de Empresa, Universidad y Ciencia of the Generalitat Valenciana, Spain, for its support through a post doctoral fellowship.

References

[1] M. D. Agrawal and S. Bharani. Performance Evaluation of a Reverse-flow Gas Turbine Combustor using Modified Hydraulic Analogy. *The institute of Engineers India Journal MC*, April 2004:34–44, 2004. http://www.ieindia.org/publish/mc/0404/apr04mc7.pdf C408

[2] HeonSeok Lee and JeongJung Yoon. The Study on Development of Low NOx Combustor with Lean Burn Characteristics for 20kW class Microturbine. *Proceedings of ASME Turbo Expo*, 14–17 June, Viena, Austria, 2004. C408

[3] R. Tuccillo and M. C. Cameretti. Comparing different solutions for the micro-gas turbine combustor. *Proceedings of ASME Turbo Expo*. 14–17 June, Viena, Austria, 2004. C408

[4] T. Kamps. *Model Jet Engines*. Traplet Publications Ltd. 3rd Edition. 2005. C408, C410, C413

[5] S. Adachia, A. Iwamotoa, S. Hayashib, H. Yamadab and S. Kaneko Emissions in combustion of lean methane-air and biomass-air mixtures supported by primary hot burned gas in a multi-stage gas turbine combustor. *Proceedings of the Combustion Institute*. Volume 31, Number 2, pages 3131–3138, 2007. doi:10.1016/j.proci.2006.07.239 C409

[6] C. Syred, W. Fick, A. J. Griffiths and N. Syred. Cyclone gasifier and cyclone combustor for the use of biomass derived gas in the operation of a small gas turbine in cogeneration plants. *Fuel*. Volume 83, Issues 17–18, pages 2381–2392, 2004. doi:10.1016/j.fuel.2004.01.013 C409

[7] I. Gurrappa and A. S. Rao. Thermal barrier coatings for enhanced efficiency of gas turbine engines. *Surface and Coatings Technology*. Volume 201, Number 6, pages 3016–3029, 2006. doi:10.1016/j.surfcoat.2006.06.026 C409

[8] A. Portinhaa, V. Teixeiraa, J. Carneiroa, J. Martinsb, M. F. Costac, R. Vassend and D. Stoeverd. Characterization of thermal barrier coatings with a gradient in porosity. *Surface and Coatings Technology*, Volume 195, Issues 2–3, pages 245–251, 2005. doi:10.1016/j.surfcoat.2004.07.094 C409

[9] G. Boudier, L. Y. M. Gicquel, T. Poinsot, D. Bissieres and C. Berat Comparison of LES, RANS and experiments in an aeronautical gas turbine combustion chamber. *Proceedings of the Combustion Institute*. Volume 31, Number 2, pages 3075–3082, 2007.

http://www.cerfacs.fr/~cfdbib/repository/TR_CFD_06_25.pdf C409, C415

[10] K.-U. Schildmacher, A. Hoffmann A, L. Selle, R. Koch, C. Schulz, H.-H. Bauer, T. Poinsot, W. Krebs and B. Prade. Unsteady flame and flow field interaction of a premixed model gas turbine burner. *Proceedings of the Combustion Institute*, Volume 31, Number 2, pages 3197–3205, 2007. doi:10.1016/j.proci.2006.07.081 C409

[11] F. Ham, S. Apte, G. Iaccarino, X. Wu, M. Herrmann, G. Constantinescuy, K. Maheshz and P. Moin. Unstructured LES of reacting Multiphase flows in realistic gas turbine combustors. In Center for Turbulence Research. *Annual Research Briefs* 2003. http://ctr.stanford.edu/ResBriefs03/asci_combustor_group.pdf C409

[12] A. Datta and S. K. Som. Combustion and emission characteristics in a gas turbine combustor at different pressure and swirl conditions. *Applied Thermal Engineering*. Volume 19, Number 9, pages 949–967. 1999. doi:10.1016/S1359-4311(98)00102-1 C409

[13] S. James, J. Zhu and M. S. Anand. Large-Eddy Simulations as a Design Tool for Gas Turbine Combustion Systems. *AIAA JOURNAL*. Volume 44, Number 4, April 2006. C409

[14] J. Artes and K. Schreckling. *Building instructions and plans for the KJ-66 turbojet engine*. Available at http://www.artesjet.com/. C410, C412

[15] J. H. Ferziger and M. Peric. *Computational methods for fluid dynamics*. Springer. 3d edition, 2002. C412

[16] *FLUENT user manuals*. Fluent Inc. v. 6.3.23. USA. C412

[17] Q. Zhou and M. A. Leschziner. A time-correlated stochastic model for particle dispersion in anisotropic turbulence. *Proceedings of the 8th Turbulent Shear Flows Symposium*. Munich, 1991. C412

[18] A. B. Liu, D. Mather and R. D. Reitz. Modeling the Effects of Drop Drag and Breakup on Fuel Sprays. *SAE Technical Paper*. 930072, SAE, 1993. C412

[19] R. D. Reitz. Modeling atomization processes in high-pressure vaporizing sprays. *Atomization and Spray Technology*. 3:309–337, 1987. C412

[20] G. X. Yang and J. S.Chin. Experimental study on atomization of plain jet injector under high pressure co-axial air flow. *ASME, 32nd International Gas Turbine Conference and Exhibition*. Anaheim, CA, May 31-June 4, 1987. C412

[21] J. S. Chin. Atomization study in Jet Propulsion Lab. BIAA—A survey report. *International Journal of Turbo and Jet-Engines*. Volume 6, Number 3–4, pages 205–219, 1989. C412

[22] A. H. Lefevbre. Airblast atomization. *Progress in Energy Combustion Science*. Volume 6, pages 233–261. 1980. C412

[23] K. N. Bray and N. Peters. Laminar Flamelets in Turbulent Flames. In P. A. Libby and F. A. Williams, editors. *Turbulent Reacting Flows*. pages 63–114. Academic Press, 1994 C412

[24] N. Peters. Laminar Diffusion Flamelet Models in Non Premixed Combustion. *Prog. Energy Combust. Sci.*. 10:319–339, 1984. C412

[25] L. Elliott, D. B. Ingham, A. G. Kyne, N. S. Mera, M. Pourkashanian, and C. W. Wilson. A novel reduced reaction mechanism for kerosene combustion generated using genetic algorithms. *Proceedings of ASME Turbo Expo 2004: Land, sea and air*. Vienna, Austria, GT-2004-53053, 2004. C412

Author addresses

1. **C. A. Gonzalez**, School of Aerospace, Mechanical & Mechatronic Engineering, The University of Sydney Australia.
   mailto:carlos.gonzalez@aeromech.usyd.edu.au
2. **K. C. Wong**, School of Aerospace, Mechanical & Mechatronic Engineering, The University of Sydney Australia.
3. **S. Armfield**, School of Aerospace, Mechanical & Mechatronic Engineering, The University of Sydney Australia.