Large Eddy Simulation of Turbulent Flow in a Fan-Stirred Combustion Vessel

F. Zhang*, T. Zirwes, P. Habisreuther*, N. Zarzalis*, D. Trimis*, H. Bockhorn***

Feichi.Zhang@kit.edu

* Engler-Bunte-Institute, Division of Combustion Technology, Karlsruhe Institute of Technology, Engler-Bunte-Ring 1, 76131 Karlsruhe, Germany

** Steinbuch Centre for Computing (SCC), Karlsruhe Institute of Technology, Hermann-von-Helmholtz-Platz 1, Karlsruhe, Germany

Abstract

The turbulent flow field within a fan-stirred combustion vessel is numerically calculated by means of large eddy simulation (LES). Objective of the work is to explore application of high fidelity numerical methods for a more detailed insight into the turbulence characteristics prevailing prior to ignition of fuel/air mixtures, which lead to turbulent flame propagation due to flame-turbulence interactions. The LES have been preformed with the code OpenFOAM for a laboratory-scale test rig, along with the moving mesh technique for considering the rotational motion of fans. Totally 8 fans are considered at varying rotating speeds. The calculated root mean square flow velocities showed a reasonably good agreement with measured data and exhibit a linear increase with the rotating speed of the fans. The integral length scale evaluated from the LES agrees well with the measured value too, which increases slightly with the rotation speed of the fans. The proposed method in this work is able to reproduce the essential characteristics of the turbulent flow field prevailing in a fan-stirred bomb.

1. Introduction

The mutual interaction between flame propagation and turbulent flow plays a key role for the efficiency, stability and emission characteristics of energy conversion through combustion [1]. One of the commonly applied experimental configurations for studying this effect is the constant-volume, fan-stirred combustion bomb [2, 3, 4]. Homogeneous, isotropic turbulence conditions can be generated by applying a number of symmetrically arranged fans mounted within the vessel. Moreover, the setup can be operated at varying thermodynamic conditions like elevated pressure, temperature and mixture composition, which are of significant importance for industrial applications [5, 6, 7]. The overall consumption rate in terms of the turbulent burning velocity and the wrinkling rate of the flame surface depend on the underlying turbulence characteristics, such as the turbulence intensity and the turbulent length scales. The initial flame kernel is small compared to the integral length, and hence the initial flame kernel is not exposed to the full spectrum of turbulence in the early stage after ignition, and, therefore remains laminar. As the

flame surface grows, it is wrinkled by larger, energy-containing vortices. Thus, a thorough characterization of the turbulent flow represents a prerequisite for the subsequent analysis of the flame-turbulence interaction in this setup.
Although a number of experimental works have been performed to study the turbulent flow field in such explosion vessel, for example in [2, 8, 9], the flow velocities can only be measured on a 2D plane with limited ranges and resolutions, so that temporal and spatial fluctuations of small-scale motions cannot be captured correctly. The objective of the present work is the detailed analysis of the underlying turbulence characteristics by the application of high fidelity numerical simulations for the turbulent flow generated in a fan-stirred combustion vessel. Highlight of the work lies in considering the rotational movement of several fans for this commonly used experimental configuration by employing moving numerical grids for the fans. The reliability of the results are validated by comparison with corresponding measured data [2].

2. Numerical Setups

The test rig consists of an explosion vessel with 8 fans mounted on the inner wall of the vessel, which generate an almost homogeneous and isotropic turbulent flow field in the core region [2]. The fans are located at the corners of a cube inscribed into the spherical vessel; the axes of the fans are collinear to the diagonals of the cube. The vessel is assumed as a sphere with a diameter of 16 cm, the axial distance from one single fan to the opposite one is 133 mm. Each fan has a diameter of 45 mm and consists of 6 blades of 6 mm depth and 3.6 mm thickness; the surfaces of the blades are flat (without curvature), having a slanted angle of 22.5°. The computational domain covers the internal volume of the vessel along with the 8 fans. Tetrahedral elements have been used to build up the grid cells. As shown in Fig.1 on the left, the average cell length in the zones I, II, III are specified to 0.5 mm, 1 mm and 2 mm respectively. The regions enclosing the fans has an average resolution of 1 mm, see Fig.1 on the right. In total, the computational mesh consists of 8.6 million cells.

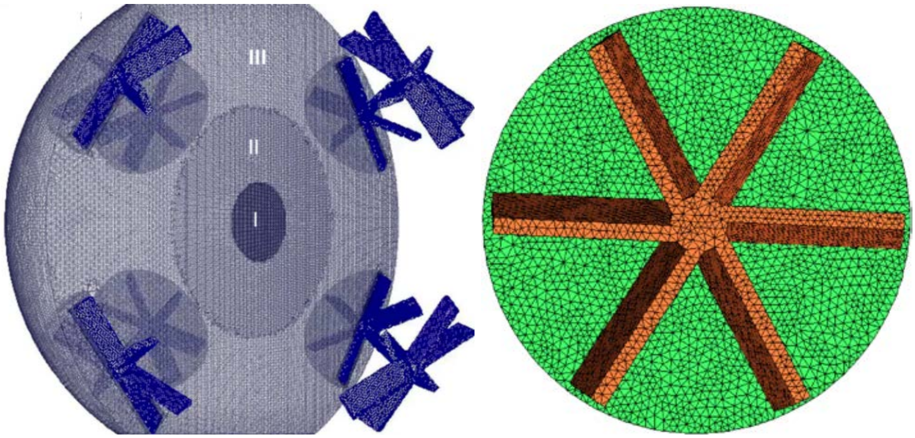

Figure 1. Surface meshes used for the LES of a fan-stirred bomb.

The open-source code OpenFOAM [11] has been used to solve the Navier-Stokes equations in the incompressible formulation by means of the finite volume method [12]. A constant kinematic viscosity of 1.58e-5 m^2/s for air at the condition of 298 K and 1 bar is assigned. The dynamic mesh implementation available in OpenFOAM has been applied to represent the rotational motion of the fans. The arbitrary mesh interface is used for exchanging data across adjacent mesh domains, in this case between the rotational zones including the fans and the remaining sphere volume. The motion of the mesh is accomplished by an automatic mesh motion solver, which is based on solving a mesh motion equation with prescribed boundary motion [12].

The solution procedure employs a fully implicit scheme of second order for the time derivative (three point backward) and a second order discretization scheme for the convective and diffusive terms [12]. A transport equation for the SGS turbulent kinetic energy [14] is applied in order to calculate the sub-grid scale (SGS) Reynolds stresses. The LES have been performed for four rotating speeds of the fans: ω = 2500, 5000, 7500, 10000 rpm (rounds per minute). The surface of the spherical vessel is set as no-slip wall. All simulations have been run for different physical times of 4.8 s, 2.4 s, 1.8 s, 1.2 s for ω = 2500, 5000, 7500, 10000 rpm (approx. 400,000 time steps) to gather time mean and rms (root mean square) statistics.

3. Simulation Results

3.1. Instantaneous Flow Field

Figure 2 shows instantaneous contours of the magnitude of flow velocity u for the cases with ω = 5000 rpm (left) and ω = 10000 rpm (right), for a slice through the center point of the vessel. For all cases, the highest flow velocities are given by the moving speed of the fan blades at their maximum radii. Therefore, the maximum value of u almost increases linearly with ω. It is largest near the fans and smallest at the axes of symmetry and in the core region of the vessel, because the flow the ventilators generate the flow in opposite directions.

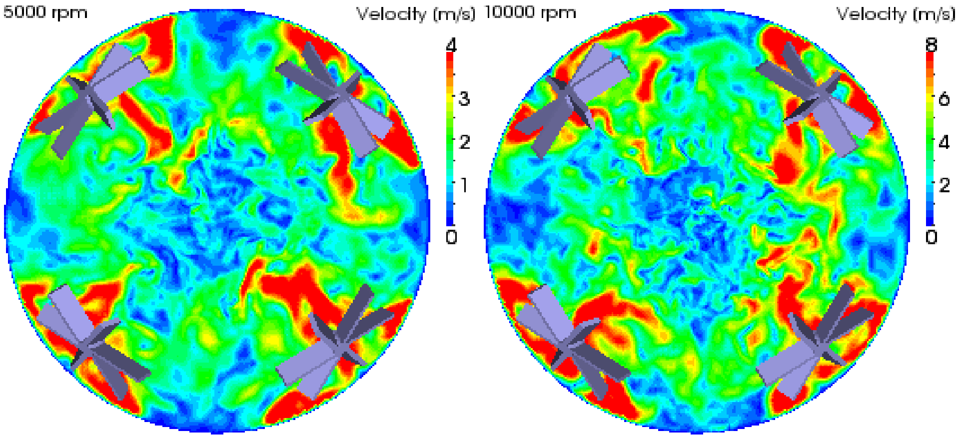

Figure 2. Instantaneous contours of the magnitude of flow velocity u for different fan rotating speed: ω = 5000 rpm (left) and 10000 rpm (right).

3.2. Mean and RMS Velocities

Figure 3 depicts contours of the time mean (left) and rms (right) values of the magnitude of flow velocity $\bar{u}$ and u' for the case of ω = 10,000 rpm, which increase from the center of the vessel towards the fans. $\bar{u}$ is zero in the center point, whereas u' is non-zero there due to the turbulent fluctuations. In addition, $\bar{u}$ and u' are not evenly distributed along the circumferential direction, they are larger in the diagonal planes across the fans (white lines) compared to the vertical and horizontal symmetric planes (black lines). Contours of $\bar{u}$ and u' are not mirrored perfectly regarding the symmetry axes, because the simulated physical time of 1.2 s may not be sufficient to smooth out long-term turbulent fluctuations. Results for other rotation speeds are qualitatively similar, $\bar{u}$ and u' however vary linearly with ω.

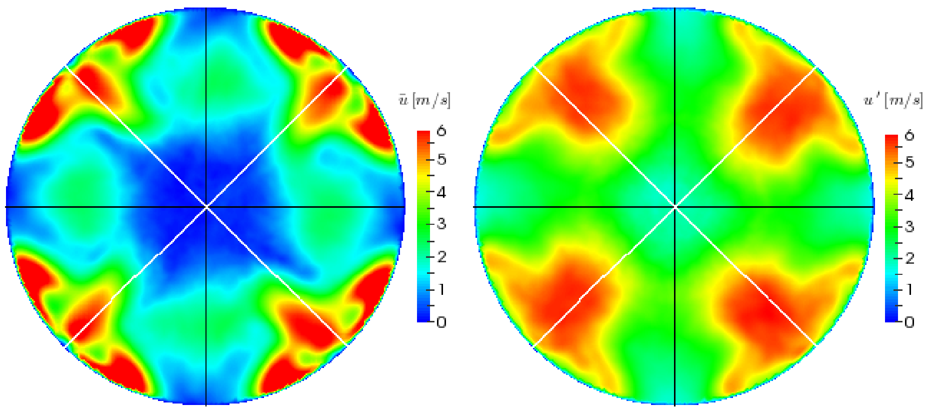

Figure 3. Contours of time-mean (left) and rms (right) velocities calculated by LES for the rotation speed of fan at ω = 10000 rpm.

In Fig. 4 on the left, the calculated and measured profiles of u' along the horizontal axis are compared for different ω, which increase with the radius from the center to a maximum value and decreases while approaching the vessel wall. A larger ω leads to an increase in u', so that the turbulent kinetic energy increases with ω too. On the right hand side of Fig.4, u' shows a quasi-linear increase with ω at different radial positions. The calculated data (solid line) and the slopes of the curves are slightly lower than the data from LDA measurement (circles). The coarse grid resolution used for LES may lead to a higher numerical diffusion and hence, an attenuated turbulent fluctuation. In addition, the k-Eqn SGS model used in this work is strictly dissipative and suffers from the eddy viscosity assumption. The deviations of geometrical setups as well as the experimental uncertainties may also contribute to this difference. For instance, the shape of the vessel is assumed to be spherical, whereas other installations such as holders for the fans and windows (for the optical measurement) are present in the experiment. Despite these differences in the simulation and the measurement, the achieved results may be regarded as satisfactory, due to the complexity of the configuration by considering the full scale of the explosion vessel and including 8 rotating fans.

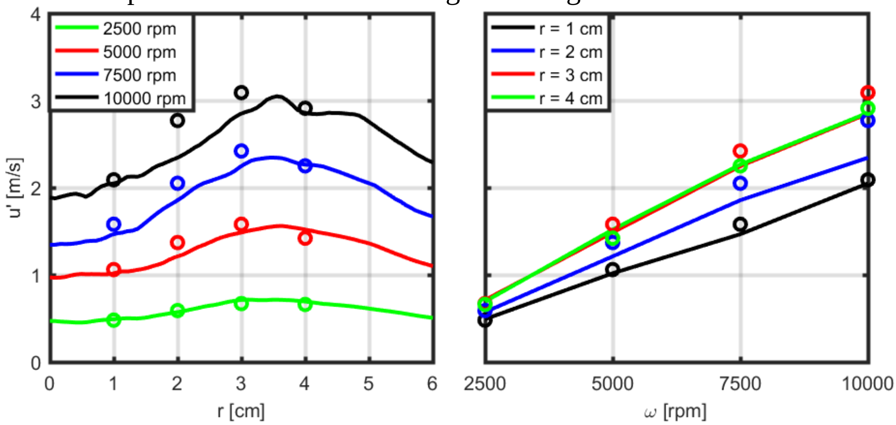

Figure 4. Comparison of calculated and measured profiles of rms velocity against the radial direction (left) and the rotation speed of the fan (right).

Based on the simulation results, the turbulent kinetic energy and eddy dissipation rate k and ε are evaluated by

$$k = \frac{1}{2}\left(\overline{u_i' u_i'}\right),\ \varepsilon = 2\nu\left(\overline{S_{ij}' S_{ij}'}\right)^2,\ S_{ij}' = \frac{1}{2}\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i}\right) \qquad (1)$$

with the strain rate tensor S_{ij}' [15]. $\bar{\cdot}$ and $\cdot'$ indicate time mean and instantaneus fluctuations. By using these parameters the integral length scale L_t is calculated with $L_t = C_\mu^{3/4}\, k^{3/2}/\varepsilon$, which shows a homogenous distribution in the core region and compares well with the measured data of $L_t \approx 4\, mm$ [2]. L_t from LES has been found to be slightly increasing with ω, whereas it was found to be independent of ω in previous experimental studies [2, 8, 9]. The turbulent Reynolds number $Re_t = u' L_t/\nu$ consequently increases with ω or u', leading to a smaller Kolmogorov length due to $\eta \propto Re_t^{-3/4}$. The decrease of η with increasing Re_t has been verified by evaluating η from LES by $\eta = (\nu^3/\varepsilon)^{1/4}$. Results for L_t and η, as well as the spectra of turbulent kinetic energy will be shown in the final presentation.

4. Conclusion

This work presents large eddy simulations (LES) of the turbulent flow in a fan-stirred explosion vessel, including several rotating fans using a moving mesh approach. The predicted turbulence field is nearly homogeneous and isotropic in the core region. The calculated rms velocities agree well with the measured data, which increase linearly with the rotation speed of the fans. The calculated integral length scale compares well with the measured value, which increases slightly with the rotation speed of the fans. Therefore, the proposed method is able to reproduce the turbulence characteristics generated in the fan-stirred bomb. The influence of turbulent flows on the flame propagation will be studied in future work.

Acknowledgements

The authors thank for the financial support by the Helmholtz Association of German Research Centres (HGF) for funding through the Research Unit EMR. This work utilizes computational resources from the High Performance Computing Center Stuttgart (HLRS) and the Steinbuch Centre for Computing (SCC) at KIT.

References

[1] Poinsot, T., Veynante, D., *Theoretical and numerical combustion*, Edwards Inc., Philadelphia, U.S.A., 2005.

[2] Weiß, M., Zarzalis, N., Suntz, R., “Experimental study of Markstein number effects on laminar flamelet velocity in turbulent premixed flames”, *Combust. Flame* 154, 671-691 (2008).

[3] Abdel-Gayed, R.G., Al-Khishali, K.J., Bradley, D., “Turbulent burning velocities and flame straining in explosions”, *Proc R Soc Lond A* 391: 393-414 (1984).

[4] Bradley, D., Haq, M.Z., Hicks, R.A., Kitagawa, T., Lawes, M., Sheppard, C.G.W., Woolley, R., “Turbulent burning velocity, burned gas distribution, and associated flame surface definition”, *Combust Flame* 133: 415-430, (2003).

[5] Lipatnikov, A., Chomiak, J., “Global stretch effects in premixed turbulent combustion”, *Proc Combust Inst* 31: 1361-1368 (2007).

[6] Bradley, D., Lawes, M., Mansour, M.S., “Explosion bomb measurements of ethanol-air laminar gaseous flame characteristics at pressures up to 1.4 MPa”, *Combust. Flame* 156: 1462-1470 (2009).

[7] Liu, C., Shy, S., Chen, H., Peng, M., “On interaction of centrally-ignited, outwardly propagating premixed flames with fully-developed isotropic turbulence at elevated pressure”, *Proc Combust Inst* 33: 1293-1299, (2011).

[8] Galmiche, B., Mazellier, N., Halter, F., Foucher, F., “Turbulence characterization of a high-pressure high-temperature fan-stirred combustion

vessel using LDV, PIV and TR-PIV measurements", *Exp. Fluids* 55: 1636 (2014).

[9] Fansler, T.D., Groff, E.G., "Turbulence characteristics of a fan-stirred combustion vessel", *Combust Flame* 80: 350-354 (1990).

[10] Sick, V., Hartman, M.R., Arpaci, V.S., Anderson, R., "Turbulent Scales in a Fan-stirred Combustion Bomb", *Combust Flame* 127: 2119-2123 (2001).

[11] OpenFOAM, The Open Source CFD Toolbox, User Guide (2014).

[12] Ferziger, J.H., Perić, M., *Computational methods for fluid dynamics*, Springer-Verlag Berlin Heidelberg, Germany, 2002.

[13] Moving Mesh Solver: https://www.slideshare.net/fumiyanozaki96/openfoam, http://web.student.chalmers.se/groups/ofw5/Advanced_Training/DynamicMesh.pdf.

[14] Fröhlich, J., *Large Eddy Simulation turbulenter Strömungen*, Teubner Verlag, Germany, 2006.

[15] Pope, S.B., *Turbulent Flows*, Cambridge University Press, Cambridge, U.K., 2000.