Large-Eddy Simulation and experiments on non-premixed highly turbulent opposed jet flows

M.W.A. Pettit a, B. Coriton b, A. Gomez b, A.M. Kempf a,⇑

a Department of Mechanical Engineering, Imperial College London, South Kensington Campus, Exhibition Road, London SW7 2AZ, United Kingdom
b Department of Mechanical Engineering, Yale University, New Haven, CT 06520-8286, USA

Abstract

An experimental and computational study is presented on highly turbulent non-premixed counterflows under both isothermal and reactive conditions. Experimentally, Hot Wire Anemometry (HWA), two-dimensional Particle Image Velocimetry (PIV) and OH Planar Laser Induced Fluorescence (PLIF) were applied. Computationally, Large-Eddy Simulations (LES) with a steady flamelet model were used to simulate the flow inside the nozzles and in the opposed flow region, using three different grid resolutions between 1.0 and 0.2 mm (0.5–70 million cells). The combined experimental and computational approach enabled the cross-validation of the simulation, and provided additional insight into the flow field. Both isothermal and burning conditions were examined with turbulent Reynolds numbers reaching a value of 900, demonstrating the system capability of reaching conditions of relevance to practical systems. Importantly, the simplicity of a compact, bench-top experiment is retained. The extension of the computational domain to a region within the nozzles with no optical access reveals the mechanism by which a specially designed turbulence generating plate (TGP) and burner housing yield turbulence intensities well exceeding 20%. The simulated and measured data were found to be in good agreement for first and second velocity moments, for the axial velocity autocorrelation function and for the normalised mean OH fluorescence. Similarity of OH-based flame morphology between experiments and computations also confirms that the LES successfully captures key features of the flow.

© 2010 Published by Elsevier Inc. on behalf of The Combustion Institute.

Keywords: Turbulent flame; Counterflow; Non-premixed; Large-Eddy Simulation; LES

1. Introduction

Advances in combustion research require that theoretical, numerical and experimental work are combined to reach a deeper physical understanding of complex problems such as turbulence, mixing, and reaction. The joint approach requires that the test cases are tackled experimentally and computationally in a coordinated effort. An ideal test case would provide good optical access for laser diagnostics, offer high turbulent Reynolds numbers, and be sufficiently compact to permit efficient field measurements and simulations. The flow must be well-defined, reproducible, and should only depend on known boundary conditions that can easily be prescribed in the simulation.

1540-7489/$ - see front matter © 2010 Published by Elsevier Inc. on behalf of The Combustion Institute.
doi:10.1016/j.proci.2010.06.140

Please cite this article in press as: M.W.A. Pettit et al., Proc. Combust. Inst. (2010), doi:10.1016/j.proci.2010.06.140
For the investigation of laminar flames, the opposed jet configuration has become the standard test case for analytical, numerical and experimental studies of non-premixed, partially premixed, and fully premixed flames. Since laminar opposed jet flames are thin and flat, they represent an idealised steady flamelet that can be used not only for theoretical analysis but also to generate the (steady flamelet) tables that are often used in the simulation of turbulent non-premixed flames.

Turbulent opposed jet (TOJ) flames have been examined for many years, with early work from Cambridge by Kostiuk et al. [1,2]. More recently, TOJ flames were examined at Imperial College by Hardalupas, Lindstedt, Taylor, Whitelaw and co-workers [3–6]. At Darmstadt, Geyer designed a new TOJ for Raman Rayleigh line measurements where the laser was aligned with the burner centreline [7,8]. The burner was also examined at Purdue [9,10], at Sandia National Labs [11], and at Imperial College in Geipel’s [6] upgraded version. All of these burners create decaying wind-tunnel turbulence from perforated plates upstream of the nozzle. Near to the turbulence generating plate (TGP), velocity fluctuations are very strong but largely dominated by intermittency between the jets from the TGP and the wakes turbulence is anisotropic, inhomogeneous, and does not show a fully developed spectrum. Such ‘young’ turbulence is very difficult to describe through statistic models, and successful Large-Eddy Simulations (LES) should consider the jets from the plates to obtain accurate predictions. To obtain further insight into the flow near the TGP, Böhmer et al. [12] have analysed the flow within the nozzles by high-speed PIV measurements taken through a transparent (glass) nozzle. Further downstream of the TGP, the turbulence begins to revert to a more isotropic behaviour with a more classical spectrum, but much of the kinetic energy has dissipated at this point.

A TOJ burner was recently developed at the Yale Center for Combustion Studies [13] with the goal of addressing some of the shortcomings of the previous approaches, thereby creating strong, fully developed turbulence. By employing high-blockage plates and judiciously selecting the feed stream composition, turbulent Reynolds numbers one order of magnitude larger than in previous work were achieved.

The goal of the present contribution is to examine the isothermal and reactive flow field of the Yale TOJ burner by applying laser diagnostics and Large-Eddy Simulation. To shed light on the turbulence generation scheme, the simulation of the flow is extended to the TGP’s within the burner housing. The paper begins with a description of the experimental apparatus developed at Yale, followed by a description of the numerical simulations performed at Imperial College. Selected results are then presented, compared and discussed.

2. Experiments

The Yale turbulent opposed jet burner consists of two vertical opposed co-axial nozzles with a separation of 19.0 mm, corresponding to one-and-a-half nozzle diameters. Turbulence is generated in a plenum chamber upstream of the nozzle, in which a specially-designed TGP is housed. A fast jet emerges from the plate and passes through a contraction, leading to recirculation and strong turbulence. The nozzle is surrounded by an annular shroud of nitrogen to shield the flame from the environment, and to prevent the products of combustion from recirculating near the nozzle and mixing with the reactants. A single nozzle of the burner is shown in Fig. 1, and the simulated flow field inside a nozzle is visualised by adding a marker fluid (Fig. 2).

The present work compares computational and experimental results for three different set-ups: firstly, a single nozzle case was analysed to obtain detailed velocity statistics in the nozzle exit plane; a second test case involved the non-reactive investigation of the complete opposed jet configuration, and finally, a non-premixed flame was stabilised near the stagnation plane. In the latter case the fuel stream contained 35% vol. methane (CH₄) in nitrogen (N₂), and the oxidiser stream contained pure oxygen (O₂), yielding a stoichiometric mixture fraction Z_f = 0.515. Table 1 shows the test cases that were examined with the relevant volumetric flow rates Q, bulk velocities U_{bulk},

\[
\begin{align*}
Q_f &= \text{volumetric flow rate of fuel} \\
Q_o &= \text{volumetric flow rate of oxidiser} \\
U_{bulk} &= \text{bulk velocity} \\
\end{align*}
\]

Fig. 1. One of the two opposed nozzles. Fluid enters the plenum from the left through the ‘petal’-shaped hole in the turbulence generating plate (TGP). A contraction accelerates the fluid towards the nozzle exit. An inert nitrogen co-flow shields the jet from the environment.

Fig. 2. Instantaneous snapshot of mixing inside a nozzle from the fine grid (\(\Delta = 0.2 \text{ mm}\)) simulation of case SJ50, 30 ms after a marker fluid is injected to visualise the strong turbulence and mixing inside the nozzle. Pixels correspond to cells; no interpolation was applied.

Please cite this article in press as: M.W.A. Pettit et al., Proc. Combust. Inst. (2010), doi:10.1016/j.proci.2010.06.140
Table 1
Cases examined and presented. The finegrid simulations for SJ50 and OF85 were not run to convergence because of their high cost.

<table>
<thead>
<tr>
<th>Case description and case code</th>
<th>$Q$ (l/min)</th>
<th>$U_b$ (m/s)</th>
<th>$Re$ ($10^4$)</th>
<th>$A$ (mm)</th>
<th>$NCPU$ (1)</th>
<th>Cells ($10^6$)</th>
<th>CPU ($10^4$ h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single jet SJ50</td>
<td>50</td>
<td>6.58</td>
<td>4.76</td>
<td>0.5</td>
<td>8</td>
<td>2.6</td>
<td>1.6</td>
</tr>
<tr>
<td></td>
<td>50</td>
<td>6.58</td>
<td>4.76</td>
<td>0.2</td>
<td>64</td>
<td>40.0</td>
<td>–</td>
</tr>
<tr>
<td>Single jet SJ75</td>
<td>75</td>
<td>9.87</td>
<td>7.12</td>
<td>0.5</td>
<td>8</td>
<td>2.6</td>
<td>1.6</td>
</tr>
<tr>
<td>Opposed jet OJ50</td>
<td>50</td>
<td>6.58</td>
<td>4.76</td>
<td>1.0</td>
<td>1</td>
<td>0.5</td>
<td>0.1</td>
</tr>
<tr>
<td></td>
<td>50</td>
<td>6.58</td>
<td>4.76</td>
<td>0.5</td>
<td>16</td>
<td>3.7</td>
<td>1.7</td>
</tr>
<tr>
<td></td>
<td>50</td>
<td>6.58</td>
<td>4.76</td>
<td>0.2</td>
<td>44</td>
<td>58.3</td>
<td>17.0</td>
</tr>
<tr>
<td>Opposed flame OF85</td>
<td>85</td>
<td>11.2</td>
<td>8.06</td>
<td>0.5</td>
<td>64</td>
<td>4.6</td>
<td>1.5</td>
</tr>
<tr>
<td></td>
<td>85</td>
<td>11.2</td>
<td>8.06</td>
<td>0.2</td>
<td>64</td>
<td>70.0</td>
<td>–</td>
</tr>
</tbody>
</table>

The two-dimensional flow field between the nozzles was examined by PIV. The flow was seeded with 1–2 μm olive oil droplets and aluminium oxide particles for the non-reactive and reactive conditions, respectively. A 0.6 mm-thick laser light sheet covering the entire region between the nozzles was observed by a CCD camera in double exposure mode. The resulting image pairs were analysed by an adaptive multi pass FFT cross-correlation algorithm using the TSI Inc. ‘Insight 3G’ software. The size of the interrogation area was reduced from 962 to 322 pixels with 50% overlap, leading to probe volumes of $0.138 \times 0.138 \times 0.6 \text{ mm}^3$ and $0.193 \times 0.193 \times 0.6 \text{ mm}^3$ for the non-reactive and reactive cases, respectively, which are comparable to the cell sizes that were used for the LES on the fine ($0.2 \text{ mm}^3$) and medium ($0.5 \text{ mm}^3$) grids. The PIV camera pixel size was calibrated with a dot grid target resulting in less than 0.1% error. The accuracy of the PIV measurements is assessed by considering the ratio of interrogation window size to the integral length-scale [3]: a ratio of between 0.04 and 0.05 for this case indicates uncertainties in turbulent intensity measurements of less than 10%.

Statistics for the flame front topology were derived from OH PLIF excited by a 3.5 mJ Nd:YAG pumped dye laser, tuned to excite the Q1(8) transition of the (1,0) band of the $A^2\Sigma^+ \rightarrow X^2\Pi$ system at a wavelength of 283.5 nm. The resulting 310 nm fluorescence signal was recorded on an intensified CCD camera (Cooke Dicam-Pro) through a 105 mm UV-Nikkor lens at f/4.5.

3. Numerical simulations

The Yale TOJ was simulated by Large-Eddy Simulation using the ‘PsiPhi’ code, which uses similar numerics to the Darmstadt ‘flows’ code [7,8]. PsiPhi solves the generalised transport equations for filtered conserved scalars $\Psi$, in this case the momentum components $\rho \mathbf{u}$, the density $\rho$ and the mixture (fraction) $\rho f$, from which all dependent quantities $\Psi$ are calculated. A low-Mach assumption is applied so that the density depends on the chemical state and temperature $T$ but not on pressure $p$ ($\partial \rho / \partial p = 0; \partial \rho / \partial T < 0$). PsiPhi uses an equidistant Cartesian grid with a cell-centred arrangement of the variables. The simple cubic cells avoid any problems with inhomogeneous anisotropic LES filters, ensure good numerical accuracy, allow for very efficient parallelisation and vectorisation, and help avoid short local time-steps that reduce efficiency throughout the domain. Scalar transport is described by 2nd order central schemes for both diffusion and convection, although the scalars $\rho$ and $\rho f$ are advected by a Total Variation Diminishing (TVD) scheme (CHARM) to avoid oscillations in bounded quantities upon which the chemical state depends. Advancement in time is performed by a 3rd order explicit low storage Runge–Kutta
scheme. The time step width is determined from the Courant–Friedrichs–Lewy (CFL) condition [14] for convection (CFL = 1.0 for isothermal cases, CFL = 0.7 for the flame).

The burner geometry is described using immersed boundary conditions on a cuboid computational domain with a cross-section of \(48 \times 48 \text{ mm}^2\) and a length that spans the entire region between the TGP of both nozzles. Three different grid resolutions were used with a filter (cell) width of \(\Delta = [1, 0.5, 0.2] \text{ mm}\), using up to 70 million cells as shown in Table 1. It should be stressed that typical simulations of turbulent opposed jets would only consider the volume between the nozzle exits, reducing the computational effort by one or two orders of magnitude. Simulations were parallelized using MPI to run on up to 64 processors, the finest simulation requiring a total of 2 CPU-years. Typically, statistics were deemed to be converged after a simulated physical time of 0.7 s, although cases SJ50 and SJ75 were allowed to run up to 1.2 s to determine integral time- and length-scales. Table 1 indicates that the ‘PsiPhi’ code demonstrates good parallel scalability. Comparing our simulation (OJ50, \(3.7 \times 10^6\) cells) to that of the Darmstadt TOJ [7] (TOJ-NR, \(3.9 \times 10^6\) cells) shows that ‘PsiPhi’ requires approximately 1/3 of the total CPU time (on 16 Opteron cores) for the same number of timesteps as the efficient single-proces- sor ‘flowsi’ code (on a single Pentium 4). Olb- richt’s [15] cross-validation has also shown that Darmstadt’s ‘FASTEST-ECL’ and ‘PsiPhi’ compare well.

Transient Dirichlet boundary conditions were applied at the inflow planes using a generator for artificial turbulence [16] to trigger realistic jet break-up. Inflow turbulence was generated with an arbitrary intensity of 2% with a length-scale of 0.5 mm. Laterally, von Neumann conditions with enforced outflow were used to avoid fluid entrainment and solution instability.

The non-premixed flame was described with a mixture fraction approach relying on a steady flamelet table that was obtained as a solution for the GRI 3.0 mechanism [17]. A laminar opposed jet flame was solved using the ‘ChemID’ code [18,19] of Eindhoven University of Technology to obtain a steady flamelet table including density, temperature, and species concentrations as a function of the mixture fraction. This chemistry model cannot capture extinction, which would require more detailed models (e.g. CMC, MMC, trans- ported PFD, LEM, FGM), which would be beyond the scope of this paper. The dependence of molecular viscosity upon temperature was obtained from Sutherland’s Law assuming that the mixture has similar properties to air. The sub-grid distribution of the mixture fraction was assumed to follow a top-hat function [20] which was parameterised by the local filtered mixture fraction and its sub-grid variation as determined from a gradient model [20,21]. The assumed top-hat distribution is easier to use and computationally more efficient, while promising a more accurate description of the filtered density function (FDF) than a \(\beta\)-function could [20]. The filtered momentum equation, as well as the mixture fraction transport equation, are closed using an eddy viscosity/diffusivity approach with a turbulent Schmidt number of 0.7 [22]. The turbulent viscosity was obtained from the classic Smagorinsky model [23] with \(C_s = 0.12\) for coarse- and medium-resolution grids and a lower constant of \(C_s = 0.065\) for the fine grid (\(\Delta = 0.2\) mm).

The Taylor length-scale \(\lambda\) is determined to be around 0.8 mm from experiments [13] with a corresponding Taylor Reynolds number \(Re_t = 164\). Geyer et al. [8] use the relationship \(\lambda/\eta_s = 151/4 \cdot Re_t^{1/2}\) to estimate a Kolmogorov scale of \(\eta_s \approx 0.21 \text{ mm}\), compared to \(\eta_s = 0.032 \text{ mm}\) in the present case. Our simulations resolve down to a filter width of \(\Delta = 0.2\) mm.

With LES, classical convergence to a grid-inde- pendent solution is no longer possible as the model contribution changes with the grid. The quality of a simulation is measured by its success- ful prediction of an experiment, but further insight into its accuracy is helpful, in particular inside the nozzles where experimental data is not available. Several quality indicators [24–26] have been suggested, but it must be stressed that they are not mathematically sufficient to ensure an accurate simulation. (Some quality indicators may yet turn out to be a sufficient criterion for an inaccurate simulation.) Celik et al. [25] suggested a model for an index \(i \approx (1 + 0.05(\overline{\nu} + \overline{\nu}_f)/\overline{\nu})^{0.53}\) for which values smaller than 80% supposedly indicate insufficient resolution, while a (somewhat arbitrary) value of 95.2% is meant to correspond to a 100% resolved DNS. In practice, only the value of 80% is of any relevance, which simply corresponds to a viscosity ratio \(\eta_f/\overline{\nu}\) of 20. This viscosity ratio is directly dependent upon the Smagorinsky constant, which is itself a model parameter.

Figure 3 shows \(r_v\) for case OF85 based on Favre-filtered quantities for the coarse, medium, and fine grids. The mean has local maxima of \([27,18,9,25,2.55]\) for the coarse, medium, and fine grid, indicating insufficiently resolved regions near the inlet on the coarse grid. It must be stressed that \(r_v < 20\) cannot confirm an accurate simulation, whereas \(r_v > 20\) does point towards a problem. We therefore only present data obtained from the medium and fine grids. To check for grid independence, case OJ50 was calculated for a long sampling period on both the medium (\(\Delta = 0.5\) mm, \(3.7 \times 10^6\) cells) and fine (\(\Delta = 0.2\) mm, \(58 \times 10^6\) cells) grids. A good agreement was obtained (Fig. 7), supporting the supposition that the medium grid suffices.
4. Results and discussion

4.1. Single nozzle

Figure 2 shows a snapshot of the flow development inside one nozzle, 30 ms after start-up and the initial injection of a white marker fluid. The jet break-up observed is largely due to ‘flapping’ in the direction of the shortest axis of the inlet ‘petal’ structures, i.e., normal to the projection in Fig. 2. Inspection of the nozzle plenum reveals that a significant recirculation zone has formed within the plenum contraction, indicated by the counterflowing marker fluid surrounding the main jet, with counterflowing velocities of up to 35% of the inlet jet velocity.

Figure 4 shows a comparison of numerical and experimental data for the free jet (single nozzle) cases SJ50 and SJ75. Figure 4a and b show the normalised mean axial velocity and its standard deviation along the centreline from inside the nozzle contraction to a point downstream of the nozzle exit, and Fig. 4d and e present the same velocities along a diameter 0.5 mm downstream of the nozzle exit. The autocorrelation function of the axial velocity at a point on the centreline 0.5 mm downstream of the nozzle exit is shown in Fig. 4c and f. The resulting integral length-scales \( L \) and turbulent Reynolds numbers \( Re_t = u L / \nu \) are shown in Fig. 5.

These single nozzle simulations provide insight into the turbulence-generating mechanisms within the nozzle plenum, with a large-scale, high-energy recirculation zone forming around the central jet through the nozzle contraction. This flow leads
to an axial turbulence intensity of around 30% at the nozzle exit. The prediction of the correct turbulence level depends on the level of turbulence imposed at the inflow boundaries, which controls the jet break-up point within the nozzle plenum (more information on this inflow sensitivity is provided in Section 4.2.1). Figure 4e shows the good agreement between the measured and simulated turbulence levels at the nozzle exit. The axial turbulence intensity approaches very high values within the nozzle. The peak axial velocity at the nozzle exit reaches 8 m/s at 50 standard litres per minute (SLPM), and 12 m/s at 75 SLPM, and the slightly pointed shapes of the velocity profiles in Fig. 4d suggest a legacy of the inlet jet persisting beyond the nozzle exit.

Autocorrelation functions of axial velocity measured on the centreline at the nozzle exit were determined in experiments and simulations using Taylor's hypothesis, and integral time-scales of 0.35 and 0.28 ms corresponding to length-scales of 3.01 and 3.62 mm were found at 50 and 75 SLPM, respectively. Figures 4c, f and 5 show that integral time- and length-scales are predicted within 10% of the measured values, and that the length-scale seems to scale with the flow rate. The under prediction of the turbulent Reynolds numbers is a direct result of the underestimated length-scale values. Power Spectral Density (PSD) functions are also available for this configuration [13].

4.2. Counterflow nozzles

Figure 6 shows instantaneous snapshots of all three velocity components for the non-reactive case OJ50. The flow from the TGP breaks up rapidly, not only due to image-normal flapping of the flat jet emitting from the TGP, but also the very high shear and turbulence production on the corrugated surface of the jet. Recirculation zones (dark regions in Fig. 6a) are evident in the region surrounding the central jet, contributing to jet break-up, jet deflection, and the production of statistically reproducible turbulence.

4.2.1. Non-reactive

Figure 7 shows numerical and experimental data for the non-reactive opposed jet OJ50. Figure 7a–c show velocity data along the centreline between the nozzle exits, while Fig. 7d–f show velocity data along a diameter 0.5 mm from the upper nozzle exit. LES results are shown for both the medium ($\Delta = 0.5$ mm) and fine ($\Delta = 0.2$ mm) grids.

Figure 7a shows the characteristic incompressible stagnation flow profile. The axial velocity fluctuates significantly in the mean stagnation plane ($Fig. 7b, z = 0$), as a result of the steep velocity gradient and the axial oscillation of the stagnation plane, whereas the radial velocity component RMS is relatively constant along the centreline (Fig. 7c). The standard deviation of the axial velocity is under-predicted by approximately 8% and 16% on the medium and fine grid, respectively. However, it should be stressed that this does not indicate a poorer representation of the flow at the finer grid resolution: as mentioned before, the level of turbulence at the nozzle exit depends on the point of jet break-up inside the nozzle. Recent work [27] on opposed jets has shown that, somewhat counter-intuitively, higher levels of inflow turbulence can yield lower velocity fluctuations at the stagnation plane and vice versa.

The experimental data in Fig. 7d shows a mean velocity peak on the centreline, implying that the jet breaks through the recirculation zones within the nozzle plenum as in the single jet cases (SJ50, SJ75). This phenomenon is captured by the fine grid simulation. The mean radial velocity across the nozzle exit is also in good agreement with the experiments, in particular considering that this quantity is far more sensitive than the mean axial velocities. A radial turbulence intensity of 25% is observed on the centreline at the nozzle exit (Fig. 7f), which is higher than the turbulence intensities of 9–14% achieved in previous opposed jet experiments [6,28].

The most significant deviations between the simulations and experiments occur near the walls of the nozzles, far away from the main area of interest (the stagnation plane and the mixing
layer) where the non-premixed flame is established under reactive conditions. At these walls, two factors lead to increased (turbulent) viscosity and to numerical diffusion, thickening the boundary layer and reducing the fluctuation levels: Smagorinsky’s subgrid model is known to produce excessive viscosities near the walls, and the immersed boundary treatment makes it necessary to apply upwind biased convection schemes near the walls to ensure stability. However, the simulation on two different grids has shown that (a) the deviations near the walls can be reduced by grid refinement and that (b) the flow in the area of interest (the stagnation plane near the centreline) is not affected by the deviations near the walls. On the whole, the agreement between experiments and simulations is very encouraging.

4.2.2. Reactive

To analyse the flame under conditions without local extinction, the plenum was extended by shifting the TGP 19 mm upstream, leading to a somewhat lower turbulence (LT) level at the nozzle exit. The fuel and oxidiser streams have different densities and momentum flow rates, so that the mean stagnation plane is shifted approximately 1.5 mm towards the less dense fuel stream. To accommodate for this shift, the \( z \)-coordinate was defined based on the actual stagnation plane. The fully converged simulations were performed on the medium grid \( (\Delta = 0.5 \text{ mm}) \) that was deemed sufficient on the basis of the non-reactive grid refinement study. A finer simulation \( (\Delta = 0.2 \text{ mm}) \) was also started but was not run to fully converged statistics because of the excessive estimated cost (5 CPU-years).

Figure 8a–c show experimental and numerical results for mean axial velocity, axial velocity fluctuation, and radial velocity fluctuation along the axis between the nozzle exits, relative to the stagnation plane located at \( z = 0 \). (d) Mean axial velocity, (e) mean radial velocity, and (f) radial turbulence intensity across the nozzle exit diameter, 0.5 mm downstream of the upper nozzle exit. (For the expensive finegrid simulation \( (58.3 \times 10^6 \text{ cells, 2 CPU-years}) \), the number of samples was doubled by exploiting the symmetry about the stagnation plane.)
normalised to obtain a unit integral value over $z$. With OH being a marker for the flamefront, the mean OH concentrations along the centreline can also be interpreted as the distribution of the probability to find the flame at a position $z$.

It is interesting to compare the axial velocity fluctuations in the iso-thermal (OJ50) and reactive (OF85) cases shown in Figs. 7b and 8b. A wider profile with a lower maximum is measured for the reactive case, and the simulations confirm the different shapes.

Figure 9 shows the mass fraction of OH between the nozzle exits at three instances from LES and PLIF images. Demonstrate the range of flame morphologies that are typical of these flames: we observe flat, thin, curved, and strongly corrugated flames, as well as flames that have locally thick regions. The rapid deformation of the flame into such convoluted structures is an indication of the high turbulence levels between the nozzles, while the thinness of the flame is testament to the high strain rates. Both computational and experimental data show similar morphologies, which further contributes to the model validation.

5. Conclusions

An experimental and numerical study of a highly-turbulent ‘opposed jet’ flow field has been presented under both isothermal and reactive conditions. Hot Wire Anemometry (HWA), Particle Image Velocimetry (PIV) and Planar Laser Induced Fluorescence of OH (OHPLIF) were applied to examine the turbulent flow and flame between the nozzles. Large-Eddy Simulations (LES) were performed and ‘quality controlled’ by a grid refinement study and an error estimator. It must be stressed that existing quality indicators may be misleading and can never confirm that a grid resolution is sufficient or that a simulation is accurate. The LES provided deeper insights into the flow and flame, in particular in regions with no optical access and for quantities that were not available from diagnostics. As a result, flow field data inside the nozzles, temperature, and mixture fraction are available. The simulated and measured data are in good agreement for first and second moments, as well as for the velocity autocorrelation function. Our experimental and computational findings confirm that the new burner can achieve higher turbulence levels than previous designs, which bodes well for the examination of
turbulent combustion regimes of relevance to practical systems.

Acknowledgements

The Imperial College authors gratefully acknowledge the financial support of the Engineering and Physical Sciences Research Council through the High Performance Computing Doctoral Training Award scheme at Imperial College London and the UK High End Computing Tera-scale Resources (HECToR) service. The Yale authors gratefully acknowledge the financial support of ACS/PRF (Grant #FA9550-06-1-0018). We are indebted to Rob Barlow of Sandia National Labs, the main organiser of the ‘Turbulent Non-Premixed Flames (TNF) Workshop’, at which the present collaboration was established.

References