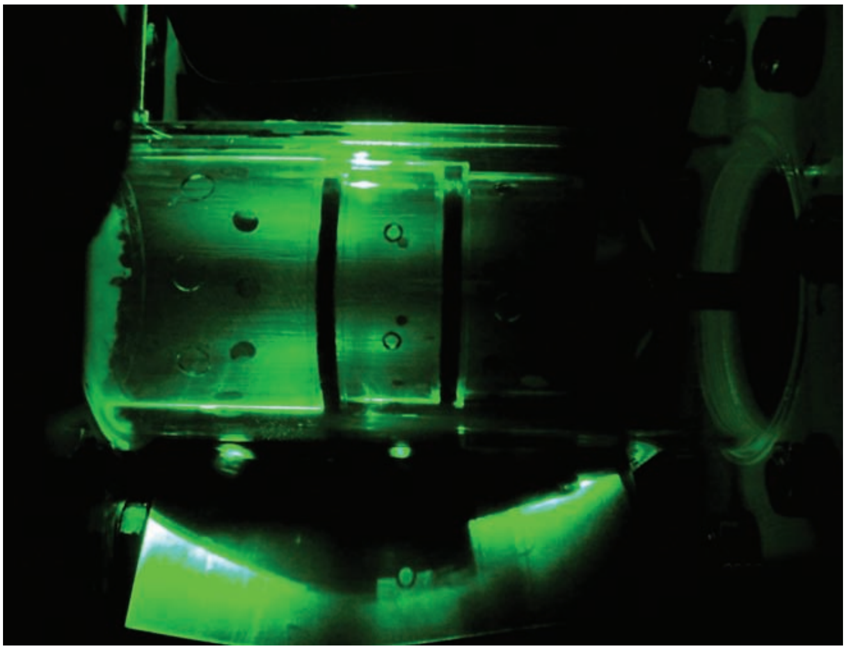
The experimental flow field and thermal measurements in an experimental can-type gas turbine combustor

by Bronwyn Meyers and Prof Josua Meyer

In a study conducted in the Department of Mechanical and Aeronautical Engineering, experimental data was collected in order to create a test case that can be used to validate computational fluid dynamics (CFD) simulations and the individual models used in them for gas turbine combustor applications.



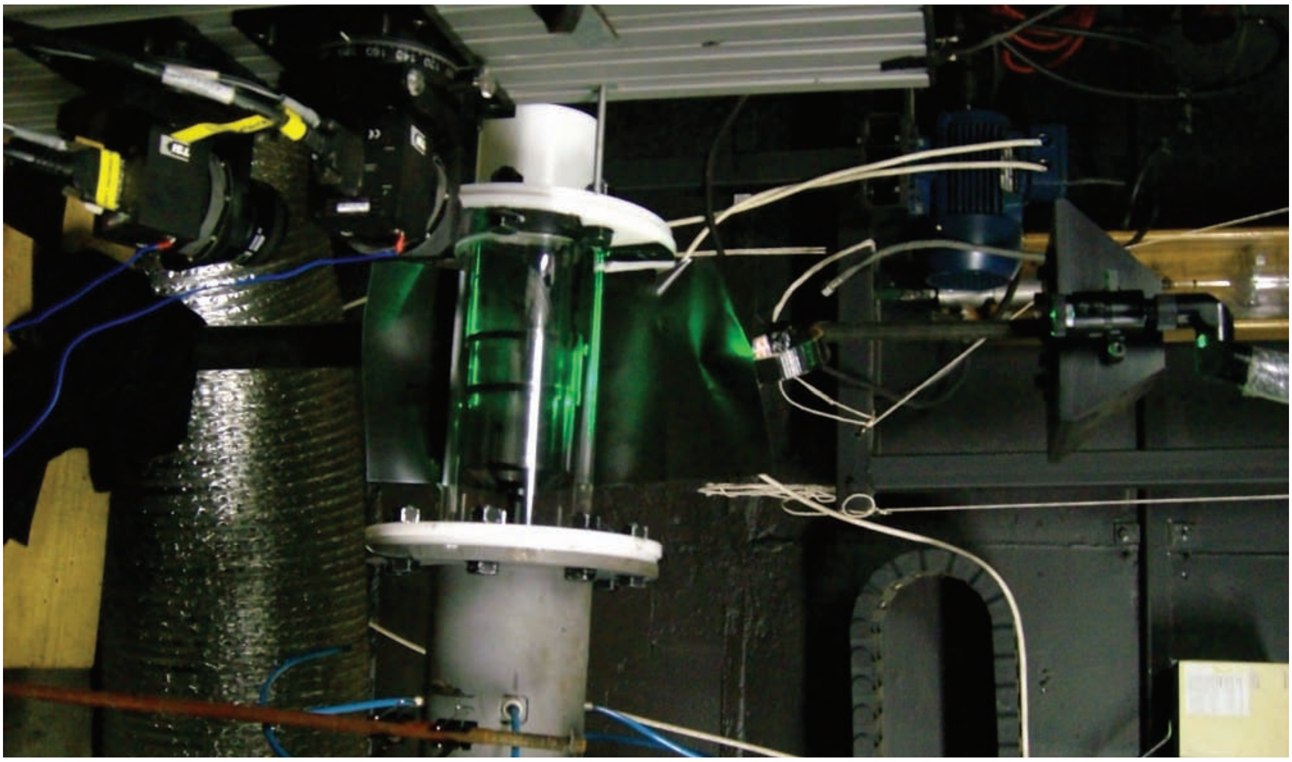
The transparent test section was illuminated using a laser for PIV measurements.

In many cases, the CFD results of gas turbine combustors do not correlate well with experimental results. For this reason, the simulation method used needs to be tested before CFD can successfully be used for combustor design. This test case encompasses all the features of a gas turbine combustor such as a swirler, primary, secondary and dilution holes, as well as cooling rings.

Experiments were performed on the same combustor geometry for both non-reacting and reacting flows. The non-reacting flow experiments consisted of stereoscopic particle image velocimetry (PIV) measurements performed at various planes in the three zones of the combustor. Data was collected on planes, both in line with the holes and in-between the holes of each zone.

For the reacting experiments, the temperatures on the outlet plane were measured using a thermocouple rake, thus a temperature contour plot on the outlet plane was produced. Furthermore, the combustor can was modified with passive inserts, which were tested to determine their influence on the outlet temperature distribution during reacting runs. In this set-up, the outlet velocity profiles were also measured using a pitot tube during both non-reacting and reacting flows. In addition to the outlet temperature distribution and velocity profiles, images of the flame patterns were captured, which showed the positions of flame tongues, fluctuating flames and steady flames. Carbon burn patterns on the walls of the combustor liner were also captured. From the data collected during the reacting runs, the pattern factor, profile factor, overall pressure loss and pressure loss factor were calculated.

The non-reacting experiments were performed using the PIV-produced three-dimensional velocity vector fields throughout the combustor. These experiments were performed at various flow rates, which gave an indication of which features of the combustor flow were affected by the flow rate. When comparing the

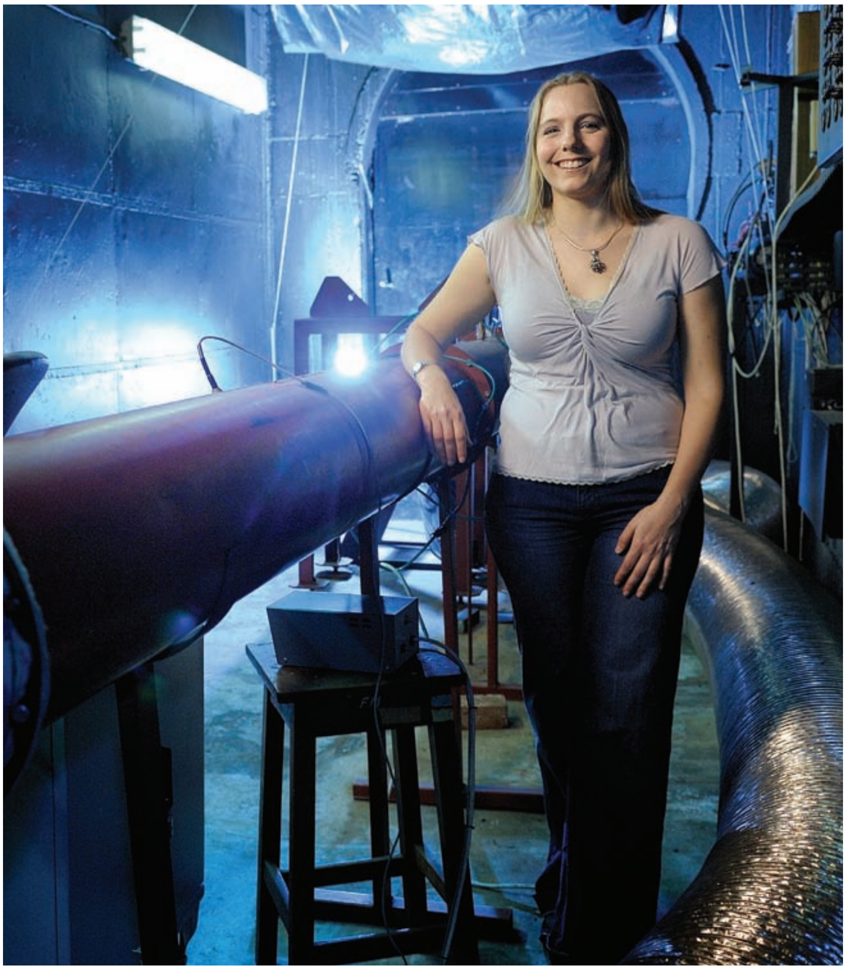


The transparent test section was illuminated using a laser for PIV measurements.

individual PIV images alongside one another, the temporal nature of the combustor flow was also evident.

The reacting experiments revealed a hot region of exhaust gas around the outer edge of the exhaust, while there was a cooler region in the centre of the outlet flow. The PIV flowfield results revealed that the reason for the hot outer ring-like region was due to the path the hot gases would take. The hot combustor gas from the primary zone diverges outwards in the secondary zone. It is then further forced to the outside by the dilution recirculation zone. The hot flow leaves the combustor along the wall, while the cooler air from the jets leaves the combustor in the centre.

The experiments performed produced a large variety of data that can be used to validate a number of aspects of combustor simulation using CFD. The non-reacting experimental data can be used to validate the turbulence models used and to evaluate how well the flow features were modelled or captured during the non-reacting stage of the combustor simulation process.



Bronwyn Meyers is a graduate of the Department of Mechanical and Aeronautical Engineering at the University of Pretoria.