Isothermal flow in a gas turbine combustor
– a benchmark experimental study

P. Koutmos and J. J. McGuirk
Imperial College of Science and Technology, Mechanical Engineering Dep., Fluids Section, Exhibition Road, London SW7 2BX, UK

Abstract. An experimental investigation of the three-dimensional flow field within a water model of a can-type gas turbine combustion chamber is presented. Flow visualisation demonstrated that internal flow patterns simulated closely those expected in real combustors. The combustor comprised a swirl driven primary zone, annulus fed primary and dilution jets and an exit contraction nozzle. LDA measurements of the three mean velocity components and corresponding turbulence intensities were obtained to map out the flow development throughout the combustor. Besides providing information to aid understanding of the complex flow events inside combustors, the data are believed to be of sufficient quantity and quality to act as a benchmark test case for the assessment of the predictive accuracy of computational models for gas-turbine combustors.

1 Introduction

New designs of gas turbine combustion systems are often aimed at simultaneous improvements in several, if not all, combustor performance parameters. High priority is usually given to expansion of the range of stable operating conditions, improvement of exit temperature traverses, and (more recently) reduced pollutant emissions. All of these features are crucially dependent on internal flow patterns and the associated rates of mixing. This fact alone has motivated the need for a better and more fundamental understanding of the processes taking place inside a combustion chamber. Further, no design techniques currently in general use can produce a combustor design that does not require subsequent development and the development process for a system which meets at least some of the above goals has also to date been largely experimental and empirical. Whilst this approach has been reasonably successful, increased use of computational procedures in parallel to experimentation would be expected to accelerate the development of improved combustor designs and reduce development costs. This is particularly evident in view of the number of interacting flow parameters within the combustor such as swirler flow, jet and cooling slot flows, influence of downstream exit contraction nozzle etc. Needless to say, the availability of suitable experimental data is also critical to the development and validation of mathematical models for combustor flows (Bruce et al. 1979; Coupland and Priddin 1986; Sturgess and Syed 1980). The motivation behind the investigation described herein was therefore twofold: (i) to provide comprehensive measurements which can be used with confidence to assess and further develop existing numerical models, and (ii) at the same time to provide data which allow better understanding of the parameters controlling the complex aerodynamics of combustor flows.

Because of the complexity of such flows the approach usually adopted for the validation of numerical models (and thus the acquisition of suitable data) is to divide the real flow into a number of well-controlled component flows that emphasise only one or two of the important physical processes at a time. Many of the experimental investigations which have provided detailed information on internal combustor flow fields have consequently employed axisymmetric geometries. This minimises the number of circumferential planes on which measurements must be made and allows attention to be focussed on, for example, the influence of swirl on simulated primary zone flow structure (Altgeld et al. 1983; Vu et al. 1983; Rhode et al. 1983; La Rue et al. 1984). Although undoubtedly important for the immediate swirler vicinity, the fact that in real combustors the axisymmetry is destroyed by the presence of radially inflowing jets weakens the general relevance of these measurements. Other studies have chosen to reverse the above roles, namely to neglect swirl and place all emphasis on the behaviour of rows of jets in a confined cross-flow (Khan et al. 1981); relevance here is clearly restricted to combustor dilution zones. Recently attempts have been made to provide measurements in flows which contain both swirl and normally discharged jets. So et al. (1985) and Ahmed and So (1987) have presented some useful data, but have unfortunately chosen to restrict attention to a single jet, which severely limits the practical relevance of their data. Their investigations did show however that jet mixing and penetration are significantly affected by the swirling motion. This implies that a strong interaction exists between swirler fluid and jet fluid and test data for model validation should contain both features. Only then
can the ability of the model to represent the feedback mechanisms between swirler, primary jet and dilution jet flows in a real combustor be properly tested. Detailed measurements which go some way to achieving this have been obtained by Jones and Toral (1983). The combustor was a realistic can-type geometry comprising a swirler, primary and dilution jets and an exit nozzle but only the temperature and gas composition fields were obtained. Some velocity information is available (Bicen and Jones 1986) but optical access problems restricted this to the immediate vicinity of the jet holes; important information on the internal flow structure is therefore missing.

The present investigation aims to provide a complete set of velocity information for a realistic combustor geometry containing both swirl and jet flow features. The model used is geometrically identical to that used in the experiments of Jones and Toral (1983) and Bicen and Jones (1986) with the only omission being the wall cooling arrangements. It is an unfortunate consequence of the desire to provide a comprehensive mapping of the internal volume that this can only be easily achieved under isothermal flow conditions using a perspex model. Water flow models are still used by industry during the empirical design process, and computational models also imply that the flow patterns can be similar in reacting and isothermal flow (Coupland and Priddin 1986). Nevertheless, the use of isothermal flow presents a need to ensure that the flow structure is similar to that expected in a burning combustor before detailed measurements are undertaken. Neither does the restriction to isothermal flow preclude the use of the data for computer model validation, since both numerical and turbulence modelling aspects will be severely tested by the complex flow under study here. The present data provide a suitable benchmark test case which any mathematical model should be capable of predicting adequately if it is to be considered for use as a design tool. In addition to providing the detailed information under one set of flow conditions, the current work also quantifies the changes brought about by modifying the swirler geometry, and demonstrates how the detailed measurements may be used to evaluate global design parameters of use to combustor designers such as the primary jet recirculation ratio.

2 Flow configuration

The experimental facility and a sketch of the model combustor geometry used are presented in Figs. 1 and 2 respectively. The main geometrical characteristics of the model combustor were intended to be similar to those found in can-type combustors [e.g. as in the Rolls-Royce Spey engine (Gradon and Miller 1968) and its later derivatives (Bhangu et al. 1983)]. The model was manufactured from perspex and consisted of a hemispherical “head” section attached to a cylindrical central barrel of 74 mm internal diameter which terminates in a circular to rectangular contraction nozzle.

The combustor was located concentrically in a larger diameter tube so that an evenly spaced surrounding annular passage was formed. The curved vane aerodynamic swirlers typical of production chambers (Bhangu et al. 1983) were attached to the head with their central hub blocked off (Fig. 3). No attempt was made to simulate any fuel injector flow, but this possesses such small momentum compared to other inflows that it has a negligible influence on setting the flow pattern. The swirl number (based on the usual defini-