CHAPTER 1

INTRODUCTION

1.1 The problem considered

Gas turbine engineering faces many challenges in the constant strive to increase the efficiency of engines during the various stages of development and design. One area where a need exists to fundamentally understand the physical phenomena involves turbulent reactive flows. The flow field inside a gas turbine combustor is extremely complex as a result of its multi-phase, strong three-dimensional behaviour combined with chemical reactions. In addition to this the combustion-chamber geometry and approach flow are usually complex.

Until recently, development of combustors primarily consisted of empirical or semi-empirical modelling combined with experimental investigations. Due to the associated costs and development time, a need exists for an improved method of development. Although experimental investigations can never be substituted completely, mathematical models incorporating numerical methods have shown to be an attractive alternative to conventional combustor design methods. At present, increased effort is made to describe combustor flows numerically, but due to the limitations of current numerical codes in terms of modelling complex geometries, turbulence, multi-phase flows etc., further development of numerical models are necessary.

Accurate and detailed experimental data is needed to correctly validate numerical models. During the past decade very little research has been done that has focused primarily on obtaining experimental measurements for this purpose. Available data stems from model combustors that is not fully representative of practical combustors and/or does not

capture the flow field in detail. Detailed measured properties of a combustor that is representative of practical combustors in terms of swirling air, operating zones, film-cooling devices etc., are thus required. This dissertation investigates a single can combustor with the purpose of obtaining experimental data and using the data to validate a numerical model that was developed to simulate the combustion process.

1.2 Review of related literature

Advanced gas turbine combustion is one of the most complex and difficult processes associated with fluid dynamics and heat transfer. Lengthy and costly experimentation on full-scale hardware is required in the design and development of combustion equipment [1]. Atmospheric and high-pressure test facilities are mostly used for the testing of different designs [2]. The test facility and test rig are typically instrumented to monitor:

- combustor airflow;
- fuel flow;
- inlet pressure and temperature;
- combustor pressure drop;
- liner temperature (using both surface-mounted thermocouples and thermal paint);
- combustor exit temperature; and
- exhaust gas composition.

The efficiency of a combustor is then determined by measuring the pattern factor and radial temperature profile at the combustor exit plane using a series of thermocouple rakes. Each rake is comprised of thermocouples mounted radially across the combustor annulus height. By using small increments to index the rake, very detailed temperature maps of the combustor exhaust flow can be obtained.

Although these test facilities and experimental investigations have been applied successfully to analyze combustor performance, the disadvantages surpass the advantages and are summarized as:

- lengthy development time;
- costs associated with experimental development;
- challenging operating conditions and the complexity of real combustors limit the extent and detail of measurements that can be made.

Self-evident is the need for an improved method of development. One method that has shown to be an attractive alternative is the use of mathematical models combined with numerical procedures. At present, increased effort is made to obtain computational fluid dynamic (CFD) models that can accurately predict combustion properties. Nevertheless, the numerical codes have to be validated by means of experimental data [3].

Charles et al. [4] provide a collection of detailed data covering a range of conditions and configurations for the purpose of model development, verification, and application. The data presented include mean and root-mean-square values of the axial and azimuthal velocities, as well as Reynolds stress data obtained using two-component laser velocimetry, and mean temperature from thermocouple measurements. The aerodynamic field was established using an R-type thermocouple probe mounted on a three-axis positioning traverse. Axial and azimuthal velocity and temperature measurements were acquired at eight axial locations downstream of the nozzle and at ten uniformly spaced radial locations from the centre-line to the outer wall. The data sets presented reflect the significant changes in the aerodynamic and thermal structure of a swirl-stabilized combustor due to relatively modest changes in the inlet boundary conditions. Unfortunately, the chamber design does not incorporate wall jets and is not representative of gas turbine combustors.

Bauer et al. [3] point out that the high complexity of real combustors impedes experimental as well as numerical investigations. Consequently, model combustors with

simple geometries have been used for scientific studies. Local measurements of the temperature, species concentration and the velocity in the three-dimensional flow are presented using liquid and gaseous fuel. Mean velocity profiles and fluctuating velocities are measured using laser anemometry equipment. With respect to the high temperatures expected, PtRh/Pt thermocouples are used. However, several preliminary studies using ordinary thermocouple probe designs demonstrated that the experimental accuracy is strongly affected by the conduction losses of the probe, which depend on the design. Therefore, a special thermocouple sensor was used. According to the local measurements, the sphere has a diameter of 2.5 millimetre offering a large surface for convective heat transfer in comparison to the heat conduction losses through the wire supports with a diameter of 65 μm. However, due to the integrating nature of the sensor design only time averaged temperature measurements could be performed. The problem of the radiative heat losses of the probe was not yet solved. Actual models, which account for radiative losses suffer from a poor knowledge of emissivities of the probe, the gas and the combustor walls. However, the new probe design rendered identical radial temperature profiles in the combustor, independent of the traversing direction. Therefore, no conventional correction method has been applied. In contrast to the velocity measurements, the investigation of the temperature field was restricted to the mixing zone of the combustor due to the presence of unburned liquid droplets in the primary zone. Impingement of fuel droplets on the probe would have caused unacceptable errors and led to destruction of the probe sensor. It was concluded that by comparing the experimental results, it was illustrated that the selected fuel type determines the combustion process decisively.

Bicen et al. [5] describe in more detail experimental methods previously used to measure velocity and temperature characteristics in a model of the low-emission can-type combustor. The combustor consists of a hemispherical head followed by a circular barrel, which contains rows of primary and secondary dilution holes. The barrel is terminated by a circular-to-rectangular nozzle. The main sources of imprecision associated with the experimental techniques were identified, the estimates of the related errors were evaluated and some sample results were presented to demonstrate the effect of air-fuel

ratio on the velocity and temperature field of the combustor. The combustor was operated near atmospheric pressure and with an inlet air temperature of 312 K. Various air-fuel ratios (AFR) were used, covering a range from forty to seventy which, typically, encompass full power to ground-idle conditions in real combustion chambers. Temperature measurements were obtained with an uncoated thermocouple fabricated from 40 μm diameter platinum, 13% rhodium-platinum wire and which was digitally compensated. The thermocouple wire was supported on 500 μm diameter wires of the same material cemented in alumina, which in turn was placed in a stainless steel tube of four millimetre outer diameter. The axial velocity profiles indicate that the primary zone recirculation increased in strength and decreased in width compared to that with isothermal flow. The combustion also results in uniform velocity profiles at the exit. The axial velocity profiles with different AFR's and in the primary zone are of similar shape and the maximum positive velocity observed close to the wall is essentially independent of AFR. The velocities near the centre-line, however, are influenced. For example, the centre-line value increases, as the AFR is increased. The primary zone flow appears to be only weakly dependent on mixture strength. In contrast, the velocities at the exit plane are strongly affected by AFR. The higher temperature and thus lower densities associated with the lower value of AFR result in higher velocities. An increase in AFR reduces, as expected, the temperature, particularly in regions downstream of the primary holes. In the primary zone, however, the temperatures are very similar for the two AFR cases and confirm the observed trends in the primary zone velocity field.

Operating with preheated inlet air and near atmospheric pressure, an investigation of the scalar characteristics of the combusting flow in a model annular combustor was also done [6]. The experiments were carried out with an inlet air temperature of 200 K above ambient for air-fuel ratios of nineteen and fifty corresponding to take-off and ground-idle conditions. Temperature was obtained by fine wire thermocouples with digital sampling. Concentrations of unburned hydrocarbon (UHC), CO_2, CO, O_2 and H_2 were obtained by a sampling probe with appropriate gas analysers. The thermocouples were made from 80μm platinum/13%-rhodium and platinum wires. The temperature signal was amplified, digitized and processed in a microcomputer to determine the mean and root-mean-square

values of temperature fluctuations. Concentration measurements were made by extracting samples of gas through a water-cooled probe inside and at the exit of the combustor. The inside and outside diameters of the water-cooled probe were one and eight millimetre respectively and consequent flow disturbances can be important, particularly in the primary zone, but cannot readily be quantified. In the primary zone, measurements are in less detail and were restricted to species concentrations only, as it was not possible to make thermocouple measurements due to high temperatures prevailing in this zone. The pattern factor with preheat is around 0.3, which is very close to that of engine practice, compared to 0.55 achieved without preheat. Increasing the air-fuel ratio has a similar effect to that without preheat, as in both cases the overall temperature decreases, and results in a higher pattern factor of around 0.48. This clearly indicates the inferior combustion efficiency achieved in the absence of preheat. As expected, the higher the average temperature, the lower the emission indices and pattern factor and the higher the efficiency.

Although experimental research has been conducted on both laboratory bench scale and full-scale hardware, the required database is still not available. This is due to several factors. Firstly, the relatively simple model combustors amenable to modelling and optical access for laser diagnostics typically do not exhibit some of the geometrical and operational features characteristic of practical combustors. Secondly, full-scale combustor beds preclude the use of optical diagnostics, have poorly defined boundary conditions, and have prohibitive operating costs.

A model gas turbine can combustor has been developed which provides optical access for laser diagnostics and clean boundary conditions amenable to modelling, and also incorporates critical features of practical combustors such as discrete wall jets, swirl induced flow field, liquid spray atomization, and elevated pressure operation [7]. The primary objective was to characterize the performance of this model combustor. A two-component, dual beam laser anemometer system was used to characterize the axial and azimuthal velocity components of the flow field. Temperature measurements were accomplished with the use of a thermocouple probe using an R-type exposed junction

thermocouple. The thermocouple was supported by a 1.6 millimetre outside diameter Inconel tube, 38 millimetre long, which was in turn mounted on a 6.4 millimetre outside diameter Inconel tube for structural rigidity. Water-cooling was provided through the larger Inconel tube to assure the structural integrity of the probe. The common perception of the flow field in a gas turbine combustor is the presence of an on-axis recirculation zone produced as a result of the interaction between the swirl air and the primary jet flow. By increasing the swirl air, the recirculation zone was transformed to an on-axis configuration. A parametric study was conducted to establish the extent to which the proportion of air directed to the swirler needed to be increased to establish on-axis recirculation for the case of reacting flow. It was found that, for a fixed flow rate through the swirler, increasing the primary jet air always led to a weakening of the recirculation zone. Except to provide closure to the dome region, a condition was not found for which the primary jet flow played a role in enhancing the recirculation zone strength or size. It was also found that near the nozzle, the spray pattern is characterized from the dips in the temperature profile and the correspondingly cooler temperatures on either side of the centre-line. Evidence of an asymmetry in the spray distribution can be deducted from the data and in consequence, a hot streak exists in the combustor. The temperature remains high downstream of the primary jets; this is evident of further reaction in the secondary zone. It was concluded that the presence of liquid injection and reaction significantly alters the flow field characteristics.

Experimental work done for the validation of numerical models is vital and necessary to develop physical insight and to refine numerical codes. In general, measurements that capture the velocity flow field of model combustors have been obtained successfully using laser anemometry systems [8]. However, from the scalar properties, i.e. species concentration, velocities and temperatures, the temperature field is the most important physical property and influences the design of the turbine section. Apparent is the lack of detailed temperature measurements. Black et al. [9] point out that due to the enormous heat load inside combustors and the turbulent character of the flow field, very few internal measurements within burning combustion rigs whose geometry and operating conditions are relevant to aero-engine combustors have been made. A need therefore

exists to obtain detail temperature measurements that capture the thermal field of gas turbine combustors.

CFD modelling of combustors and fuel nozzles has progressed to the extent that CFD simulations are frequently included in the design process [10]. It is important to note that the CFD applications for gas turbine combustors are driven not only by the potential cost and time savings due to reduced development time. It is also driven by the capability of the CFD results to provide additional insight into complex problems that do not lend themselves to analytical solutions and may be too costly and time consuming to pursue experimentally [11].

In an effort to reduce the time associated with development of a new combustion system for an advanced turbofan engine, analytical models have been applied to both the internal and external flow field of a combustor [12]. Combustor design goals were established in terms of temperature rise, pattern factor, pressure drop, lean blowout fuel/air ratio and maximum metal temperature. For the internal flow analysis the measured spray characteristics of the fuel nozzles, along with the local pressure drops and external velocities obtained from the external flow analysis were used as the initial and boundary conditions. During the flow field optimization study, airflow splits, orifice circumferential and axial locations and the number of orifices were adjusted to arrive at the best configuration. The combustor discharge conditions, including total pressure and temperature and gaseous emissions, were measured with multiple-element radial rakes mounted on a rotating drum. The drum was rotated in ten-degree increments over the full three hundred and sixty-degrees to completely characterize the discharge conditions of the combustor. The liner wall temperatures were measured initially with thermal-sensitive paint and later with skin mounted thermocouples. The combustor internal flow analysis provided good guidance in the design of the combustor orifice pattern. With the exception of system pressure drop, all of the design goals were achieved with only local cooling modification required during the rig test evaluation to meet the goals.

Due to the limited computing capabilities, many numerical investigations have focused on one specific area of gas turbine combustion, i.e. multi-phase flow, combustion reactions, turbulence modelling, etc. Research has also been conducted in the field of spray combustor modelling [13]. Problems of current models are an inability to model chemical reaction accurately and the shortcomings of simple turbulence models. Additionally, in spray combustion systems, an accurate representation of the fuel droplets is often considered important. In the present study an algebraic stress model was employed for turbulence. It was shown that this model gives a significant improvement over k-ε models when predicting swirling flows. Models of the Magnussen type were employed for chemical reaction and a detailed treatment of the fuel spray by means of a lagrangian, droplet-tracking technique. The latter takes into account the effect of slip between gas and droplets and the effect of turbulence on the droplets. The velocity profile, at the combustor exit plane, was measured using an uncooled stainless steel pitot tube. Temperature profiles within the combustor and at the exit were measured by means of a platinum-platinum 13% rhodium thermocouple. This was operated unshielded and a correction for radiation was applied to the data. It was concluded that turbulence has a significant effect on the fuel spray distribution and it appears that an algebraic stress model can be used satisfactorily, without adjustment of the modelling coefficients normally employed for isothermal flow. Combustion models, which require knowledge of only mean scalar variables, have given promising results.

A numerical study of radial and axial swirlers in terms of their effect on primary zone operation, as well as their impact on downstream radial temperature profile and emissions have also been investigated and discussed [14]. A three hundred and sixty-degree rotating drum assembly containing six thermocouples was installed in the plane of the first turbine nozzle inlet to measure exhaust gas temperature distribution. CFD modelling was employed to study the effect of two different swirlers on combustion and emission performance. The model provided detailed insights, not possible to achieve experimentally, into the aerodynamic behavior and the creation and consumption of intermediate species.

Current combustion research has the objective of establishing combustion technologies that ensure low emissions by future civilian aircraft fleets. One promising concept under intensive study is the staged combustor. The combustor consists of three stages or zones. The first zone consists of a fuel nozzle and rich-burn combustion chamber. This is followed by a quick-quench zone where the rich mixture is cooled by an array of air dilution slots. The third or lean-burn zone is a combustion chamber that is operated at lean conditions. By operating at overall rich and lean conditions, little NO_x is produced. Cline et al. [15] focus on the modifying of the numerical code, primarily intended for in-cylinder combustion calculations, to analyze the flow in a complete staged gas turbine combustor, including the air-blast fuel nozzle, for three different nozzle airflow distributions. For computational efficiency, the staged combustor was divided into two sections: the rich-burn section, which includes the rich-burn zone and the air-blast fuel nozzle, and the mixer section, which includes the quick-quench and lean-burn zones. The inflow conditions for the mixer section were obtained from the results of the rich-burn section calculation. For the rich-burn section the numerical code was modified to allow three separate air passages for the air-blast fuel nozzle inner, outer, and dome airflow. The boundary conditions were modified to allow separate inlet conditions for each air passage downstream of the swirl vanes. The inflow boundary conditions were the specification of the density and axial velocity. The outflow boundary condition was the specification of a uniform static pressure. For the mixer section, the numerical code was modified to accept the results of the rich-burn section as inflow conditions for the mixer calculation. Again, the inflow boundary conditions are the specification of the three velocity components, density, turbulent kinetic energy, and turbulent length scale. Because the flow is subsonic, the mixer inflow pressure cannot be specified, but instead must be calculated as part of the solution. If the calculated inflow pressure does not differ significantly from the rich-burn section value at the same location, then the solution is assumed to be complete. However, if the calculated inflow pressure differs significantly, then the rich-burn section calculation must be repeated. The new outflow boundary condition for the rich-burn section would be the pressure obtained from the mixer solution. For the three cases investigated, the mixer section inlet pressure was considered to be close enough to the rich-burn section pressure and, therefore, the calculations were

not repeated. The results obtained illustrated the complicated flow field present in staged combustors. In particular, the thermal protection requirements of the quick-quench zone used in the study may be similar to those of the rich-burn zone despite the presence of the cool dilution jet. From the results obtained it appears that the modified numerical code can be used to study the effects of various staged combustor designs and operating conditions, such as the air mass flow split, equivalence ratio, amount of swirl, combustor wall shape, dilution jet mass flow and momentum, as well as the number, size, shape, and inclination angle of the dilution slot.

The potential of the combustor concept with Rich-burn / Quick-quench / Lean-burn was also investigated in a separate study [16]. The flow field and the temperature and mixture distribution were measured under atmospheric conditions with conventional and laser techniques. Local gas composition of CO_2, CO, H_2, O_2, NO and UHC was measured using suction probes. Temperatures were measured with a Al_2O_3 layered PtRh-Pt 30/6 thermocouple and the measured temperatures were corrected for radiation. In the combustion model used the species reaction time scale is thought to be very small compared to the turbulent time scale, such that mixture can be considered to be in chemical equilibrium. The discretisation process, basic to many numerical codes, applies the assumption that in a small volume properties are constant for a small time step. So the chemical state of the mixture can be described by the equilibrated mixture and its energy level. However, in the small time-step considered, turbulence influences the mixture and the energy, such that a number of equilibrium states in each volume exits. This time dependent behavior can be described by probability density functions, which give the distribution in time of all possible states. The calculations started in the entrance plane of the mixing zone with measured velocity, temperature and composition distributions. The number of measured points was small compared to the number of grid entrance points. Moreover, the measured distributions did not completely show the expected periodicity, which is necessary for the calculation. In the interpolation process used, periodicity was enforced. Due to the limited optical access only the axial and vertical velocity components were measured. It was concluded that the number of measurements was not large enough, only in two axial-velocity planes, to allow a detailed prescription of

velocity distributions. The higher momentum ratio between jet flow and primary zone flow in the experiment relative to the calculation, leads to a higher penetration. Moreover, differences in the momentum of neighbouring jets generate turbulence and consequently promote turbulent exchange, which improve mixing. Incomplete penetration and less azimuthal turbulent exchange cause the calculated temperatures of the central region of both planes to decrease slower axially than the comparative measured temperatures.

Combustion option of a general three-dimensional numerical code, using the k-ε model coupled with chemical equilibrium hypothesis was applied to a sector of an annular gas turbine combustion chamber [17]. The aim of the work is to show the ability of standard turbulent combustion modelling to predict some characteristics of a real combustion chamber. A Laser-Doppler velocimeter was used for velocity measurements. The temperature was measured by fine-wire thermocouples with digital sampling, and concentrations of CO_2, CO and O_2 were obtained on dry basis by sampling through a microprobe and a water-cooled probe. The numerical code is based on finite difference and finite volume discretisation for scalar transport equation with the standard k-ε model used for turbulence calculation. The β-function is used as probability-density function of the distribution of a mixing rate scalar f, which is equal to one in pure fuel regions and zero in pure air regions. Eight equations are resolved: mass conservation, the three components of momentum conservation, k and ε equations, and mean value and variance of mixing rate scalar f. Fast kinetics and chemical equilibrium are assumed in turbulent combustion. The results showed that by using the standard k-ε model coupled with chemical equilibrium hypothesis, applied to a sector of an annular combustor gave good quantitative results.

Biswas et al. [18] also note that a widely used approach to predict the turbulence characteristics is to use the k-ε turbulence model for closure, and that an accurate prediction of combusting flow requires proper physics-based modelling of turbulence and combustion. However, the standard k-ε model needs to be improved to adequately predict the complex swirling flow in the gas turbine combustor. In the present work, a standard

k-ε turbulence model is modified to take into account the secondary swirling effect of swirl by formally operating on the algebraic Reynolds stress model. A combustion model based on the eddy breakup concept is used. For this model, three transport equations are solved for: a passive mixing rate scalar which is equal to one in pure fuel regions, and to zero in pure air regions and mass fraction of fuel and oxidant in flow. Using the above-mentioned combustion and turbulence model, a three-dimensional numerical code to solve the turbulent combustion in gas turbine combustor is developed on the basis of semi-implicit method for pressure linked equations (SIMPLE) scheme. The numerical code was validated for several cases, and the k-ε model improved for swirling flows predicted the profiles of various quantities satisfactorily.

Recent progress in two aspects of simulation - the modelling of turbulence and the modelling of combustion chemistry, have been described [19]. The PDF (probability density function) method is used to simulate turbulent combustion. At a given time, the flow is represented by a large number of notional particles. Each particle represents a fixed mass of fluid with a certain position, velocity, temperature, and chemical composition. The simulation consists of computing the time evolution of the particle properties. To describe the chemistry for turbulent-flow simulations, a simplified description is needed. ILDM (intrinsic low-dimensional manifolds) were developed to simplify the representation of the chemical kinetics. The composition of a fluid particle in the flow can be represented as a point in a multi-dimensional composition space; then, as time goes on, the point moves in the space under the actions of chemical reaction and molecular diffusion. The ILDM method simplifies the situation by considering that instead of filling the multi-dimensional composition space, the particles are attracted to low-dimensional manifolds within the space. The ILDM method provides a mathematical prescription for identifying the intrinsic manifolds of specified dimensionality for any fuel/air mixture. The PDF turbulence model was used in combination with the ILDM chemistry model to make calculations of velocity, density and composition in turbulent flames. The model calculations agree well with the experimental data.

A phenomenological modelling approach to combustor design supplemented by key experiments which depict the fundamental physical phenomena present within combustors is feasible and a necessity in minimizing combustor development time and cost requirements [1]. A two-dimensional axisymmetric modelling approach with emphasis placed on combustion models and their application to reacting flows were investigated. A primitive, pressure-velocity-variable, finite-difference code is applied to predict two-dimensional axisymmetric turbulent reacting flows. The method and program involve a staggered grid system for axial and radial velocities and a line relaxation technique for the efficient solution of the equations. Turbulence simulation is by way of a two-equation k-ε model and combustion via three chemical reaction schemes based on Arrhenius and eddy-breakup concepts for diffusion and premixed situations. The present contribution focuses on combustion simulation schemes for turbulent flames. In combustion simulation, the overall one-step reaction scheme fails to predict the formation of intermediates, such as carbon monoxide. The two-step scheme can provide the prediction of carbon monoxide, which will improve the combustion simulation dramatically, but the formation of intermediate hydrocarbons is ignored. The four-step scheme describes a transformation mechanism of the hydrogen fuel into intermediate hydrocarbons and hydrogen, and the oxidation of intermediates to carbon monoxide, carbon dioxide, and water vapor. In the computer simulation of a chemically reacting flow field, the problem is simulated by simultaneous nonlinear partial differential equations. The governing partial differential equations are reduced to a set of finite-difference equations for values at points of the grid system covering the solution domain. These, together with appropriate boundary conditions, constitute a system of strongly coupled simultaneous algebraic equations. The task is to obtain predictions for velocities, pressure, temperature, species concentrations, and turbulence quantities throughout the region of interest. The three chemical reaction schemes investigated are:

- One-step reaction scheme

$$C_xH_y + (x+\frac{y}{4})(O_2 + nN_2) \rightarrow xCO_2 + \frac{y}{2}H_2O + n(x+\frac{y}{4})N_2$$

- Two-step reaction scheme

$$C_xH_y + (\frac{x}{2} + \frac{y}{4})(O_2 + nN_2) \rightarrow xCO + \frac{y}{2}H_2O + n(\frac{x}{2} + \frac{y}{4})N_2$$

$$CO + \frac{1}{2}O_2 \rightarrow CO_2$$

- Four-step reaction scheme

$$C_xH_{2x+2} \rightarrow \frac{x}{2}C_2H_4 + H_2$$

$$C_2H_4 + O_2 \rightarrow 2CO + 2H_2$$

$$CO + \frac{1}{2}O_2 \rightarrow CO_2$$

$$H_2 + \frac{1}{2}O_2 \rightarrow H_2O$$

The results showed that near the inlet of the combustor, the predicted temperature profiles are in good agreement with the data. However, in the developing and middle regions, all three models over-predict the temperature level because of the over-estimated reaction rates. Generally, the higher-order scheme shows better agreement with the data than the lower-order results due to the relatively slow reaction rate.

Similarly, Dong et al. [20] applied a primitive, pressure-velocity-variable, finite-difference code to predict two-dimensional axisymmetric turbulent reacting flows. The method and program involve a staggered grid system for axial and radial velocities and a line relaxation technique for efficient solution of the equations. Turbulence simulation is by way of a two equation k-ε model and combustion via three chemical reaction schemes based on Arrhenius and eddy-breakup concepts for diffusion and premixed situations. Combustion flow field predictions were given to illustrate the effects of various reaction schemes on the flow field (velocities, species concentrations, temperature and turbulence

levels). The combustion schemes investigated are the one-step, two-step and four-step scheme. Improved simulation of turbulent chemically reacting flow is possible by the appropriate choice of the combustion model. It is deduced that higher-order chemical reaction schemes perform better than lower-order schemes. The two-step and four-step schemes both show promise for application in gas turbine combustors, because more kinetic information can be qualitatively calculated with these two schemes. The mechanism of the four-step chemical reaction scheme is appealing, and with further parameter evaluation and testing, it is likely to permit more accurate prediction of major species, such as unburned fuel, carbon monoxide and hydrogen.

In general, there are two distinct approaches to the numerical simulation of multi-phase fluid dynamics [21]. One is based on the lagrangian treatment, where the denser of the phases is followed throughout the flow field on lagrangian trajectories. The second is based on the eulerian approach, predicting disperse two-phase flows, on the assumption that there are two space-sharing interspersed continua present. The following objectives were established:

- to demonstrate the feasibility of solving the equations;
- to demonstrate the accuracy of the results for a one-dimensional, two-phase problem, and the plausibility of the results for some practical industrial problems; and
- to provide information about computer costs.

One of the practical problems considered was the application to a combustor/burner design. The k-ε model of turbulence was used to calculate the effective viscosity in all regions of the flow, except where strong swirl exists, where a mixing-length model is used instead. Use is made of a single-step combustion model either diffusion or kinetically controlled. The disappearance rate of fuel was calculated in the kinetically controlled model as the minimum of an arhenius expression and the eddy-break-up expressions. It was found that flame length decreases as combustion air temperature increases. As air temperature is increased both internal and external recirculation zones are enhanced, suggesting a better flame stability. It is also highlighted that turbulence

models for two-phase flows are still at the speculative stage. The great disadvantage of the current turbulence models is that they ignore "intermittency". All real turbulent flows exhibit fragmentariness, i.e. large-scale inhomogeneities, ignored by conventional turbulence models.

Kurreck et al. [22] investigated a three-dimensional combustor flow in detail. A test case was selected to validate a two-phase flow simulation code that was developed. The solution strategy for the prediction of coupled two-phase flows with an eulerian / lagrangian approach is:

- calculate the gaseous phase without droplet source terms;
- calculate the liquid phase and source terms;
- repeat step one including the droplet sources; and
- repetition of steps three and two until convergence is achieved.

The gaseous phase boundary conditions are obtained from experimental results. For the liquid phase it is important, especially for the atmospheric case, to select suitable droplet starting conditions, since the solution of the calculation is strongly influenced by these values. Adequate modelling of the atomization process is very time-consuming and with reasonably accuracy for a technical application currently not available. Therefore, it is of prime importance to obtain all information about droplet diameter spectrum, starting position, staring velocities and temperatures from experimental data. From the results it was found that the core of the spray is relatively cold. The temperature decreases because of the evaporation of droplets. The predicted temperatures are higher compared to the measurements. One reason for this may be the radiative losses of the temperature probe, especially in the main reaction zone. Another reason is the assumption of complete reaction in the underlying model for the heat release.

A CFD home code based on the finite-volume method and body-conforming non-orthogonal but structured grids for calculating 3D turbulent reactive flows was employed to simulate a single annular reverse-flow combustor both at ground idle and full load

power conditions [23, 24]. Turbulence is simulated by way of the standard k-ε model along with the wall function treatment for the near-wall regions. The conservation equations solved for the gas phase are those for momentum, mass, kinetic energy of turbulence and its dissipation. Two-phase effects also require to be modelled, because in combustion systems fuel is often injected in the form of a spray of small liquid droplets. Among the numerous spray models available in literature, the deterministic separated flow model was chosen. This model assumes that the fuel is injected into the combustion chamber as a fully atomized spray, which consists of spherical droplets. The liquid-phase equations are based on the Lagrangian formulation of the droplet trajectory, transient heating and vaporization. The entire fuel spray is constructed using a finite number of size ranges obeying a two parameter droplet size distribution, which is assumed to be of Rosin-Rammler type. The description of droplet heating and evaporation is added to the computational model using two equations, which are solved separately. The former is solved only during the preheating period until the droplet temperature reaches boiling point, at constant diameter. Subsequently, the droplet temperature remains constant and the decrease in diameter occurs. The correct implementation of the appropriate boundary conditions is crucial in obtaining meaningful and accurate results. Unless there is mass injection involved in treating solid surfaces, the no-slip and no-through-flow conditions are imposed. The wall is adiabatic and no surface boundary conditions on either pressure or density are required. At the outflow boundary, a zero gradient of the contravariant velocity components is specified from which the Cartesian components are derived. A one-dimensional code was used to determine air flow distribution through the rows of holes across the liner as well as pressure drops and injection angles. The errors in numerical results are due to the uncertainty of the velocity and mass flows specified at various inlets, the assumption of fast chemistry, the assumption of a standard k-ε turbulence model and the neglection of radiation effects. The experimental errors are due to sector to sector non-uniformity, radiation losses from the thermocouple and those caused by the intrusion of the probes into the flow. The results of the analysis suggest that the modelling techniques employed have practical application for the prediction of main combustion characteristics. The agreement with experimental data used to validate the code ranged from fair to good.

In the work done by Ebbinghaus et al. [25] CFD is used to analyse the mixing of three rows of jets arranged in different patterns in a can combustor and to predict the resulting temperature profiles. The aim was to achieve a more uniform temperature profile, both radially and circumferentially. A one-step global reaction mechanism for propane is assumed. The chemical reaction is modelled using the Magnussen mixing controlled reaction model and an Arrhenius rate mechanism, the smaller of the two rates defining the reaction rate for each cell. The physical properties of the combustion reactants and products for each cell are assumed to be dependent on species concentration. The combustion walls are assumed to be adiabatic. The standard k-ε model of turbulence, the Power Law differencing scheme and the SIMPLE solution algorithm were used. From the numerical results it was found that an improvement in the uniformity of the circumferential temperature distribution was achieved if the number of dilution holes was equal to or larger than the number of secondary holes. The primary jets have no discernible influence on the temperature distribution at the exit plane, as they are masked by the secondary and dilution jet flows. Temperatures were measured using bare wire K-type and R-type thermocouples. The thermocouple was mounted on a support which provided movement on three translational axes and it was aligned parallel to the axis of the can to minimize the perturbations. From the measurements it was found that all results exhibit a non-cyclic temperature peak. It was evident from the experiments that temperature hot spots are due to non-cyclic behaviour, which have previously been shown to be the result of uneven fuel or air distribution. If those variations in the boundary conditions were known, this could be incorporated in the numerical model, but as flow symmetry will no longer be present, the numerical effort would increase accordingly. Using a basic numerical model the influence the of air jet pattern on the temperature profile of a can combustor was investigated and the comparison of three different designs with experiments in general validated the predictions. As would be expected, only qualitative agreement is achieved if the fuel and airflow boundary conditions are not equal to the experimental conditions. To quantitatively compare temperatures, the inlet conditions of the model must coincide with the experiment. A weakness of the numerical model is the underestimation of mixing behind jets which

results in an over-prediction of the cyclic temperature variations immediately downstream of the jets.

Numerical modelling has generally been successful in predicting trends in mixing performance, but mixing rates are usually underpredicted when compared with experimentally measured mixing rates [26]. In a numerical study, ways to reduce pattern factor in reverse flow combustors by increasing large-scale circumferential mixing were investigated. The numerical model consisted of:

- a primary zone model to determine dilution zone inlet conditions; and
- three dilution zone/transition duct models that extended to the turbine inlet guide vanes.

The basic capabilities of the numerical code include:

- co-located, fully implicit, and strongly conservative finite volume formulation;
- solution of two- and three-dimensional Navier-Stokes equations for incompressible and compressible flows;
- cartesian and non-orthogonal curvilinear coordinates;
- single and multi-domain grid topology;
- upwind, central, second-order upwind, and Osher-Chakravarthy differencing schemes;
- standard and extended k-ε turbulence models, the two-scale turbulence model of Kim and Chen and the low-Reynolds-number k-ε model of Chien;
- instantaneous, one-step, and two-step combustion models;
- modified form of Stone's strongly implicit solver and conjugate gradient solver; and
- pressure-based solution algorithms including SIMPLE and a variant of SIMPLEC.

The numerical details of the calculations include:

- whole field solution of u momentum, v momentum, w momentum, pressure correction, turbulent kinetic energy k, dissipation rate ε, total enthalpy, and mixture fraction;
- upwind differencing for all variables;
- variable fluid properties;
- adiabatic walls;
- standard k-ε model with wall functions;
- turbulent Prandtl number of 0.9;
- one-step reaction kinetics model; and
- premixed fuel and air (primary zone model only).

A solution for the primary zone was obtained for the purpose of providing mainstream inlet boundary conditions for the dilution zone models. It was concluded that numerical tests using three-dimensional CFD models have been performed to analyse an advanced dilution hole concept for small reverse flow combustors. The advanced dilution hole concept consisted of injecting the dilution air jets with a high circumferential component. The results of the numerical tests showed that this has the potential of reducing pattern factor as much as 60 percent compared to the base-line dilution hole configuration.

Hornsby et al. [27] describe the use of CFD in advancing the development of an injection system to optimize the low NO_x performance of a generic combustor design. Extensive use has been made of CFD to improve the prediction of fuel-air mixing in addition to the internal flow pattern of the combustor. Potential problems of the CFD models are:

- the availability of adequate boundary conditions;
- a further problem is that of geometry, in particular how accurately the model represents the geometry and if the grid is sufficiently refined to capture the nature of the flow; and

- the physical modelling in the CFD code itself. If the parameter in question depends strongly on a phenomenon which is not included in the mathematical models then the analysis could prove futile.

CFD modelling was used as a tool to aid engineering development, rather than a detailed analysis to provide quantitative performance data. The alternative approach of detailed quantitative analysis not only carries with it a large time penalty, but also a requirement for correspondingly detailed data measurement both upstream of the combustor for boundary condition definition, and inside the combustor for validation purposes. The physical model for this study made use of:

- the k-ε turbulence model
- analysis is steady state
- density calculated as an ideal gas
- wall functions
- the combined eddy break-up/chemical kinetics reaction rate model
- fuel is CH_4 using 2-step reaction and using the reaction rate data of Dupont
- combustion is adiabatic and radiation is not significant
- properties at model inlets assumed to be flat, based upon bulk flow properties from test data.

The methodology used to apply CFD to DLN combustion systems was proved to be fast and effective in optimizing the emissions characteristics of the combustor. Ways to increase the accuracy are:

- further grid refinement
- enhanced turbulence models
- enhanced combustion models
- transient analysis

Most models include only the reacting flow inside the combustor liner with assumed profiles and flow splits at the various liner inlets [10]. A logical next step is to model the entire flow field from the compressor diffuser to the turbine inlet with the flow inside and outside the combustor liner fully coupled. Two reasons for doing this are:

- flow splits and boundary conditions for the combustor liner inlets are modelled explicitly and no longer need to be approximated; and
- liner wall temperatures can be predicted when the flow field on both sides of the liner walls are modelled in a coupled fashion.

In general, it is necessary to couple the combustor solution and at least part of the swirler passages because of the strong recirculating flow in the region of the swirler. Although conjugate heat transfer approaches that couple heat transfer in solid material with the fluid flow solution have been available for some time, the authors are not aware of any attempt to directly model gas turbine combustor liner wall temperatures using CFD. This research describes CFD analysis of a model combustor from the deswirl vane exit of a centrifugal compressor to the turbine nozzle inlet. The total airflow through the combustor was prescribed by the single boundary condition at the compressor diffuser. All of the remaining flow splits, except for the flow through the dome, were a product of the CFD model. Kinetics were modelled using a one-step finite rate reaction with equilibrium reaction products. This efficient model accounts for the significant effects of finite reaction time and provides a much better prediction of flame temperature compared to simple one-step models with only H_2O and CO_2 as products. The PDF model was used to account for turbulent/chemistry interaction. The two-variable PDF was applied to the fuel mixture fraction and to the progress variable. The liquid fuel spray motion and evaporation was modelled as lagrangian droplet packets that influence the eulerian gas phase equations through mass, energy, and momentum source terms. The comprehensive model made it possible to predict flow splits for the various openings into the combustor liner and to remove the guesswork required for prescribing accurate boundary conditions for those openings. The coupled modelling of the flow inside and outside the liner,

combined with conjugate heat transfer analysis and participating gas radiation, provided a direct prediction of liner wall temperatures.

McGuirk et al. [28] also point out that computational modelling of combustor systems is generally performed in two stages: Firstly, the external aerodynamics are considered, allowing the calculation of the mass flow splits into individual ports of the combustor. Secondly, the internal flow field is calculated, using the bulk jet inlet conditions found from the external model. The models employed for the external calculations are often empirically based, or use simple, one- or two-dimensional computational models. Although these methods give an indication of the flow split between the primary, intermediate and dilution ports, they provide little detailed information about the jet properties as they enter the combustion chamber. Jet velocity profiles, jet entry and angle profiles and jet turbulence levels will, however, all influence the internal flow field to some extent. It is often suspected that poor predictions of combustor exit temperature profiles are due to the unresolved jet properties used as inlet conditions. Indeed, the best predictions of such an important parameter as exit temperature pattern factor, are usually gained only when test data are used to specify the velocity field in the various combustor inlet streams. In order to avoid this dependency on test data, it will be necessary to extend the area of CFD modelling to include the external flow within the annuli surrounding the flame tube and carry out coupled external/internal calculations. It seems therefore timely to revisit the question of describing air-entry port details within CFD models of combustors and the accurate modelling of combustor geometries.

One of the most recent improvements on the development of a general-purpose combustion code and its applications to flow analysis of a production gas turbine combustor are described by Anand et al. [29]. The highly modular structure provides for easy implementation and enhancements of physicochemical models for turbulence, combustion, spray, evaporation, etc. Two of the main modules are: the chemical reaction module that incorporates a two-step general hydrocarbon combustion chemistry with the eddy-breakup combustion model and a presumed Probability Density Function (PDF) laminar flamelet combustion model; and the spray module in which a stochastic

Lagrangian transport and evaporation models are used with an efficient droplet tracking algorithm. The standard k-ε model is used to account for the turbulence effects. A user-friendly and efficient grid-generator that generates a complex single-block grid incorporating all the typical features of gas turbine combustors, namely multiple swirlers, shrouds, primary and dilution holes, transition, etc. has been developed. The numerical model includes the five most commonly used types of boundary conditions: inflow, outflow, symmetry, periodicity, and solid wall. Wall boundary conditions are formulated by the standard wall function approach. The inflow boundary conditions include axial swirler, nodal injection, effusion cooling, and fuel nozzle. An axial swirler is an annular inlet on an i-plane, over which the flow velocity is defined by its axial, radial and tangential components. A nodal injection is an inlet over which the flow has constant properties. An effusion cooled surface is modelled by inflow fluxes of cooling air evenly distributed over the cooled area and fed into the flow field as source terms. These inflow boundary conditions are most relevant to combustors, but also general enough to cover a wide range of flow simulations. A detailed computational analysis was carried out in conjunction with experimental testing and design changes of a production gas turbine combustor to help understand the flow field and to guide the design modifications. The analysis involved a detailed modelling of all major physical aspects of the combustor such as dome swirlers, fuel nozzles, inner and outer heat shields, splash plates, primary and dilution holes, effusion cooling, etc. A baseline configuration and several modifications were considered. The flow distribution to the various combustor liner inlets is calculated using a one-dimensional flow distribution analysis taking into account the compressibility effects, and the different diffuser pressure drops to the dome, and the inner and outer combustor annuli. The mass flows, effective areas, jet velocities and jet angles obtained from this analysis were used as inlet boundary conditions. The mass flow distributions were also verified through cold-flow measurements of flow distribution for the liners. The inlet conditions for the fuel spray, including the Sauter mean diameter, drop-size distribution and the droplet velocities are derived, based on the nozzle type, geometry and the nozzle operating conditions. The only modelling parameter or constant optimized in this entire study was the Schmidt number that controls the diffusion of the scalars only. Standard values were used for all other modelling constants, such as the k-ε

model constant. The same value of the Schmidt number was used for all the scalar variables, namely the fuel mixture fraction, unburned fuel and CO mass fractions. The value of the Prandtl number for enthalpy was also set to be the same as the Schmidt number used. The value of the Schmidt number (Sc) was varied over the range 0.5 to 1.0. Temperature measurements on the rig are made by a two-arm rotating thermocouple probe, with five shielded thermocouples located at different span heights on each of the arms. It was found that for a Schmidt number equal to 0.6, the computed results match well with the rig data both in magnitude and the shape of the profile. As Sc is increased from 0.6 the diffusion of the dilution jets into the primary zone combustion products, or vice versa decreases, and hence the dilution jets penetrate further towards the centre of the combustor annulus. Thus the mid-span of the annulus gets cooler while the inner and outer regions get warmer. On the other hand, if Sc is reduced, the diffusion of the dilution jets into the combustion products increases, and the jet penetration reduces. This causes an increase in the temperature around mid-span and a reduction on either side. The value of Sc controls the mixing of fuel and air in the primary zone and has a strong influence on the temperature field in the primary zone. This, in conjunction with dilution jet penetration, has a strong influence on the exit temperature profile. Several other combustor modifications were also analyzed. In all the cases the computational results captured the trends of the shifts in the temperature profile very well. While in some of the cases even the magnitudes were predicted accurately, there were some discrepancies in the magnitude for some cases. However, the trends were accurate enough to assess the effect of design changes - even minor changes such as changing the shape of the dilution orifices from holes to slots of the same area - that the model proved to be a valuable tool in evaluating design changes and in guiding future modifications. The success of the numerical model, in spite of the well-known limitations of the turbulence and combustion models used, is partly due to the robustness and improved accuracy of the numerical techniques used, compared to staggered-grid and grid-aligned velocity components used in alternate methods. Another important success factor is the proper specification and modelling of the combustor inflow features. The results presented show, in general, a significant sensitivity to flow-rates and boundary conditions/modelling of the various combustor features.

1.3 Need for this study

It can be concluded from the literature survey that numerical modelling is the key to future developments in the field of gas turbine combustion. Due to the advantages associated with CFD, focus has shifted from analyzing combustion on a chemistry level to a physics based model. However, further development is still necessary to optimize the usable resources.

The lack of sufficient experimental data - specifically the thermal field - that can be used for validation purposes is apparent. This is as a result of the connected costs of experimental investigations, the operating conditions and the complexity of combustors. Temperature measurements mainly consist of a detailed distribution at the exit plane of the combustor with a very limited number of temperature measurements inside the combustion chamber. Also apparent is the fact that very few measured liner wall temperatures are used for direct validation purposes. One of the reasons being that a very small change in the location of a probe can make all the difference between recording and failing to record a "hot spot" [30]. Most of the experimental work that has been investigated, focused on combustors with simple geometries that is not fully representative of practical combustors. A need therefore exits to obtain more detailed temperature measurements for a conventional gas turbine combustor during operating conditions. Such measurements have to be able to capture the three-dimensional thermal field inside the combustor, together with exit gas and liner wall temperatures.

Despite the increased effort and progress that have been made in computational methods, the literature survey revealed that CFD is not yet capable of accurately predicting the properties to a standard where CFD can be implemented as primary design tool supported by minimal experimental work. CFD is rather employed as an aid in the design process and experimental testing is consequently still needed. Numerical models developed "in-house" also prove to be better equipped than models developed using commercial codes.

A trend that was captured from the various numerical investigations is the overprediction of temperatures, associated with incorrect mixing and reaction rates.

In most instances, the standard k-ε model is employed to model turbulence in combustors. Despite various modifications to the k-ε model to account for swirl induced flow, none has yielded increased accuracy. A need exists for an improved turbulence model, especially in combustion applications.

Due to the limited computing capabilities available in the past, only the internal flow field was modelled, with the boundary conditions specified at the various inlets of the combustor obtained either experimentally or by using a one-dimensional computer code. A certain amount of uncertainty is therefore present in the specifications of the jet angles and discharge coefficients. At present the computing capabilities are at the point where the combustor has to be modelled from compressor outlet to turbine inlet - thereby combining the external and internal flow field. This is a necessity to reduce the assumptions and errors made during the specification of the inlet boundary conditions. It is evident that numerical models have the capabilities, but still need to be developed to their full potential.

The purpose of this study is twofold. Firstly, to experimentally investigate the physical properties associated with a research combustor that is geometrically representative of practical combustors. Detailed experimental data that constitutes temperature measurements will be procured. Secondly, to use the experimental data for the validation of a computational fluids dynamic model that was developed to simulate the research combustor using a commercial code.

1.4 Outline of this study

Chapter two provides a basic review of the combustion principles that is necessary to understand the subsequent sections. The basic design features of a conventional gas

turbine combustor are described, as well as the purpose of each feature. The chemical reactions that occur during combustion are very complex and a simplified reaction that occurs during combustion is presented. One of many ways to evaluate the efficiency of different combustor designs is discussed.

Chapter three examines the experimental measurements that were taken to establish the physical properties during operating conditions, and the geometrical positions of these measurements are proffered. The various features of the experimental setup are described and a detailed description of the research combustor is given.

Chapter four describes the computational fluids dynamic model incorporated to simulate the combustion process. The commercial code is capable of implementing different combustion schemes and a description of the Presumed-PDF model of unpremixed turbulent reaction that was used for the combustion model investigated is given. The computational grid and geometry modelling are very important in obtaining realistic predictions. These are investigated and discussed. Equally important is the boundary conditions prescribed at the inlets and outlets. The operating and boundary conditions chosen for the numerical model are presented.

Chapter five discusses the experimental and numerical results arrived at. Based on the experimental measurements, an evaluation is made of the overall efficiency of the research combustor. The numerical results are discussed and an evaluation is also made of the capabilities of the numerical model to capture the measurements and trends obtained experimentally.

Chapter six summarizes the work done in this study. Conclusions reached during the investigation are drawn and contributions to the field of gas turbine combustion are highlighted. Recommendations for further work relevant to the present study are made.

CHAPTER 2

COMBUSTION PRINCIPLES

2.1 Preamble

The previous chapter provided a review on some of the research that has been undertaken in the field of gas turbine combustion. The conventional combustion-chamber design has been developed based on principles obtained from experimental investigations and empirical formulations. This chapter provides a short review on these principles of gas turbine combustors.

The basic design features of unpremixed combustors are discussed as well as the respective function of each feature or element. The chemical reactions that occur during combustion are very complex and involve a number of various reactions. A simplified equation for combustion of hydrocarbon fuels are given and discussed. The different chamber designs must be evaluated in terms of efficiency and this is done by obtaining an efficiency parameter. One of several ways to assess the efficiency of a combustor is also presented. The information is obtained from the well-known text of Lefebvre [30] and Harman [31].

2.2 Basic design features

The basic design features of a conventional gas turbine combustor are shown in Figure 2.1.

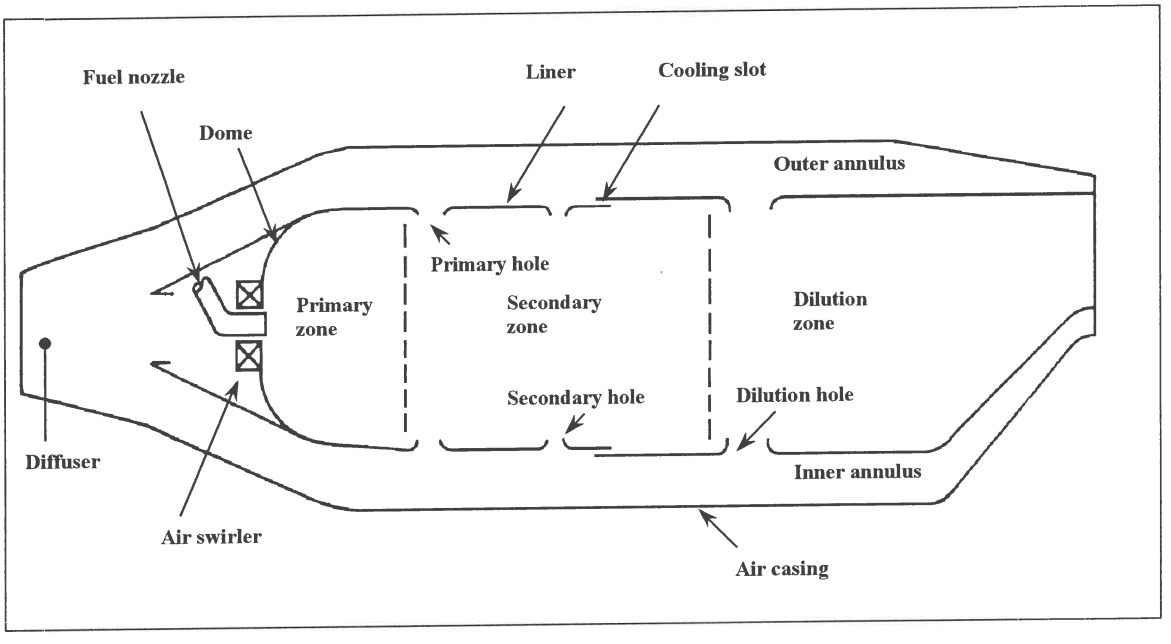

Figure 2.1 Main components of a gas turbine combustor

In general all chambers incorporate, as key components, an air casing, diffuser, liner and fuel injector. The function of the liner is to provide a region of low velocity in which combustion is sustained by a recirculatory flow of burned products that provide a continuous source of ignition for the incoming fuel-air mixture. The liner is usually divided into three zones, i.e. primary, secondary and dilution. The functions of the primary zone are to anchor the flame and to provide sufficient time, temperature, and turbulence to achieve essentially complete combustion of the fuel. Large-scale recirculation is obtained by the use of a small number of large jets. With this system, mixing of fresh mixture and combustion products occurs slowly, and, in consequence, the maximum volumetric heat-release rate tends to be low. However, burning can be maintained over a wide range of fuel flows and down to low pressures. In general, a satisfactory recirculation flow path can be achieved by the use of an air swirler combined

with opposed jets flowing radially inward from the liner wall. This flow path is in the form of a toroidal vortex and causes the partly burned mixture to recirculate several times back to the point of fuel entry: this allows ample residence time for the reactions to proceed, as the spiral flow path lengths are quite long, and the process is continually regenerated by re-entry into the hottest region. However, apart from the manner in which air is distributed throughout the primary zone, another major design variable is the actual quantity supplied. As the fuel flow rate is determined by the overall combustor fuel/air ratio, the only independent control that can be exercised over the primary-zone fuel/air ratio is via the amount of air employed in primary combustion. It is the primary-zone fuel/air ratio that governs the nature of the combustion process and the overall chamber characteristics.

If the combustion gases issuing from the primary zone were admitted directly into the dilution zone, then the dissociated products they contain, such as CO and H_2, would be quenched so rapidly that recombination could not occur in the short time available before the gases are discharged from the chamber. In consequence, the level of combustion efficiency would be less than 100 percent. To alleviate this problem, it is customary, whenever practicable, to insert a secondary zone between the primary and dilution zones. The main function of the secondary zone is to provide conditions that are conductive to recombination and thus to the elimination of dissociated products from the gases entering the dilution zone. Temperature plays a crucial role in this process: If it is too low, then recombination will proceed very slowly and will not be completed in the time available. If the temperature is too high, then the gases entering the dilution zone will contain dissociated species in equilibrium concentrations. Although the role of the secondary zone is somewhat nebulous and difficult to define, its importance to overall combustion performance should not be underestimated. In addition to recovering the dissociation loss, it also serves as an extension of the combustion zone under conditions for which combustion performance is limited by inadequate reaction and/or evaporation rates. Thus a sizable secondary zone can appreciably augment the overall combustion efficiency at low power conditions, and enhance the multifuel capability of the engine. The secondary zone also contributes to temperature traverse quality in two ways. It lowers the general

temperature level of the predilution gases, and it provides time and distance for the amelioration of any hot spots in the primary zone efflux, which might otherwise pass through the dilution zone and cause overheating of the nozzle guide vanes.

The role of the dilution zone is to admit the air remaining after the combustion and wall-cooling requirements have been met, and to provide an outlet stream with a mean temperature and a temperature distribution that are acceptable to the turbine. It is generally accepted that a satisfactory temperature profile is dependent upon both adequate penetration of the dilution jets and use of the correct number of jets to form sufficient localized mixing regions. The most important temperature parameters are those that affect the power output of the engine and the life and durability of the hot sections downstream. As far as overall engine performance is concerned, the most important temperature is the turbine inlet temperature T_4, which is the mass-flow-weighted mean of all the exit temperatures recorded for one standard of liner. Since the nozzle guide vanes are fixed relative to the combustor, they must be designed to withstand the maximum temperature found in the traverse. Thus, the parameter of most relevance to the design of nozzle guide vanes is the overall temperature distribution factor, which highlights this maximum temperature. It is normally defined as

$$\text{Pattern factor} = \frac{T_{\max} - T_4}{T_4 - T_3} \tag{2.1}$$

where T_{max} = maximum recorded temperature

T_3 = mean inlet air temperature

T_4 = mean exit temperature

The liner must be structurally strong to withstand the buckling load created by the pressure differential across the liner wall. It must also have sufficient thermal resistance to withstand continuous and cyclic high-temperature operation. This is accomplished through the use of high-temperature, oxidant-resistant materials combined with the effective use of cooling air. Film cooling makes use of a number of annular slots through

which air is injected axially along the inner wall of the liner, to provide a protective film of cooling air between the wall and hot combustion gases. The cool film is gradually destroyed by turbulent mixing with the hot gas stream, so that normal practice is to provide a succession of slots along the length of the liner. At the downstream end of the liner the flow acceleration in the nozzle tends to suppress the hot-stream turbulence, and the cooling film can persist for a much greater distance. The main advantage of the method is that the injection slots, or cooling strips, can be designed to withstand severe pressure and thermal stresses at high temperatures for periods up to several thousand hours. Moreover, the stiffness provided by the cooling strips result in a liner construction that is both light in weight and mechanically robust. A basic limitation of the method is that it does not allow a uniform wall temperature. The wall is coolest near each slot and increases in temperature in a downstream direction to the next slot.

The processes of liquid atomization and evaporation are of fundamental importance to the behavior of a gas turbine combustion system. Normal fuels are not sufficiently volatile to produce vapor in the amounts required for ignition and combustion unless they are atomized into a large number of drops with corresponding vastly increased surface area. The smaller the drop size, the faster the rate of evaporation. The influence of the drop size on ignition performance is of special importance, since large increases in ignition energy are needed to overcome even a slight deterioration in atomization quality. Spray quality also affects stability limits, idle combustion efficiency, and the emission levels of smoke, carbon monoxide, and unburned hydrocarbons. A common method of achieving atomization is by forcing the fuel under pressure through a specially designed orifice. Because of the need to minimize combustor length, a spray angle of around 90-degrees is customary. This is impossible to attain with a simple drilled orifice, and it is generally achieved by imparting swirl to the emerging fuel jet. The principal advantages of pressure-swirl atomizers are good mechanical reliability and an ability to sustain combustion at very weak mixture strengths. Their drawbacks include potential plugging of the small passages and orifice by contaminants in the fuel and an innate tendency toward high soot formation at high combustion pressures.

2.3 Chemical reactions

The combustion of hydrocarbon fuels can in general be expressed as:

$$Fuel + Air \rightarrow Combustion\ \ products \tag{2.2}$$

Using kerosene as an example, with an average molecule of $C_{12}H_{24}$, its combustion in air to form carbon dioxide and water is expressed by the simple equation

$$C_{12}H_{24} + 18O_2 + 67.7N_2 \rightarrow 12CO_2 + 12H_2O + 67.7N_2 + heat \tag{2.3}$$

The ratio by weight of this stoichiometric mixture, in which exactly the right amount of air is available to complete the combustion of fuel, is determined from the molecular weights

$$\text{Air: } (18\times32) + (67.7\times28) = 576 + 1895.6 = 2471.6 \tag{2.4}$$

$$\text{Fuel: } (12\times12) + (24\times1) = 144 + 24 = 168 \tag{2.5}$$

The air/fuel ratio is thus 14.712 while fuel/air ratio (f/a) is given by the inverse and is equal to 0.06795. The equivalence ratio ϕ is used to relate a particular fuel/air ratio to the stoichiometric value

$$\phi = \frac{(f/a)_{actual}}{(f/a)_{stoichiometric}} \tag{2.6}$$

In practice, the chemical reaction is by no means as simple as suggested by equation 2.3. There are far more reactants than the basic fuel molecule and oxygen, and the over-all reaction is made up by the combination of various more simpler reactions. Hydrocarbon fuels contain numerous different sizes and forms of molecules, from the light gases to the heavy oils and waxes, of which $C_{12}H_{24}$ describes only the mean composition of the

molecules present in kerosene. The different molecules have a wide range of boiling points, specific gravities and other properties, with slight variations in lower calorific value. The heavier fractions take longer to break down than the lighter fractions, while branched molecules take longer than straight-chain molecules. As the molecules are broken down by the heat of the combustion a number of smaller molecules and radicals form which interact with others as they combine and divide a continuing sequence. Some of the reactants which are present during the combustion process may be seen in the colour of the flame: blue indicates ideal combustion of a fully mixed supply of fuel and air, white or yellow regions show the formation of carbon and green shows a locally rich mixture with considerable formation of radicals.

The many reactants take part in a wide variety of reactions, some giving out heat (exhothermic) and some absorbing heat (endothermic). The reactions may go forward or backwards, depending on the equilibrium concentration of the different reactants. High values of temperature and pressure increase the energy and frequency of collisions between reactant molecules, but some collisions are glancing blows yielding no reaction. As combustion continues, the reactant concentrations reduce exponentially while the temperature rises: the rate of reaction is proportional to $e^{-K/RT}$ where K is the activation constant of the substance. When this sequence of reactions ends, the effect is that the reaction in equation 2.3 is complete and the constituents of the outlet gas are the low energy, inert final products. Sometimes, however, the reaction does not run to completion. The chemical processes all require a period of time, the duration of which varies for the different reaction. This time may be sometimes too short, if the reactants are quenched by contact with a draught of cooler air or gas or by contact with the cooled container: this leaves carbon monoxide (CO) and unburned hydrocarbon molecules in the outlet gas. Alternatively, the time spent at high temperature may be excessive, permitting the formation of oxides of nitrogen (NO_x) and the dissociation of water and carbon dioxide. The effects of incomplete combustion are seen in the emission of various pollutants and in the loss of combustion efficiency and performance.

2.4 Combustion efficiency

In the process of combustion in a gas turbine, liquid fuel is usually injected into a region of flow reversal, and combustion is initiated after a time interval during which partial or complete evaporation is achieved. The primary purpose is to raise the temperature of the airflow by efficient burning of the fuel; thus, an important design requirement is a means of relating combustion efficiency to the operating variables of air pressure, temperature, and mass flow rate, and to the combustion-chamber dimensions. One of the most widely used approaches for describing combustion efficiency when the overall rate of heat release is limited by chemical kinetics is the burning-velocity model. Here the combustion zone is envisaged as being similar in structure to the flame brush produced on a bunsen burner under turbulent flow conditions. Combustion performance is then described as a function of the ratio of turbulent burning velocity to the velocity of the fresh mixture entering the combustion zone. It is assumed that evaporation rates and mixing rates are both infinitely fast, and that all the fuel that burns does so completely. Combustion inefficiency arises when some of the mixture succeeds in passing through the combustion zone without being entrained by a turbulent flame front. Combustion efficiency is defined as

$$\eta_c = \frac{heat\ released\ in\ combustion}{heat\ available\ in\ fuel} = \frac{\rho_g A_f S_T c_{pg} \Delta T}{q\dot{m}_A H} \tag{2.7}$$

Now $c_{pg}\Delta T = qH$ by definition; also the flame area A_f may be assumed to be proportional to the combustor reference area A_f. Thus equation 2.7 simplifies to

$$\eta_c \alpha \frac{S_T}{U_{ref}} \tag{2.8}$$

If one expresses U_{ref} in terms of $\dot{m}_A$, P_3, and A_{ref} and describes S_T in terms of laminar burning velocity and turbulence intensity, equation 2.8 becomes

$$\eta_c = f\left(\frac{P_3 A_{ref}(P_3 D_{ref})^m \exp(T_3/b)}{\dot{m}_A}\right)\left(\frac{\Delta P_L}{q_{ref}}\right)^{0.5m} \tag{2.9}$$

It was demonstrated that combustion-efficiency data obtained during low-pressure tests on several types of combustion chambers could be satisfactorily correlated by assuming m=0.75. Owing to the relatively small variations in inlet temperature experienced by most combustion chambers, equation 2.9 is not very sensitive to the value of b. Satisfactory correlation of data is obtained using a constant value for b of 300. Experimental evidence supporting the inclusion of a liner pressure-drop term in equation 2.9 is meager. For this reason, and because the liner pressure-loss factor does not vary much from one combustor to another, the pressure-loss term is usually omitted, and equation 2.9 reduces to the well-known loading parameter θ.

$$\eta_{c,\theta} = f(\theta) = f\left(\frac{P_3^{1.75} A_{ref} D_{ref}^{0.75} \exp(T_3/300)}{\dot{m}_A}\right) \tag{2.10}$$

Equation 2.10 has been applied with considerable success to the correlation of experimental data on combustion efficiency, and it has proved very useful in reducing the amount of rig testing required to evaluate new combustor designs. As shown in Figure 2.2 only a few test points are needed to establish the complete performance curve for a chamber. Furthermore, it is possible to predict, with reasonable accuracy, combustion efficiencies at flow conditions that lie outside the capacity of the test facility, provided that at these extrapolated conditions the combustion performance is not limited by fuel evaporation or by any factor other than chemical reaction rates. The main advantage is that it provides a method of scaling combustor dimensions and operating conditions to common values so that any differences in performance that remain can be attributed directly to differences in design.

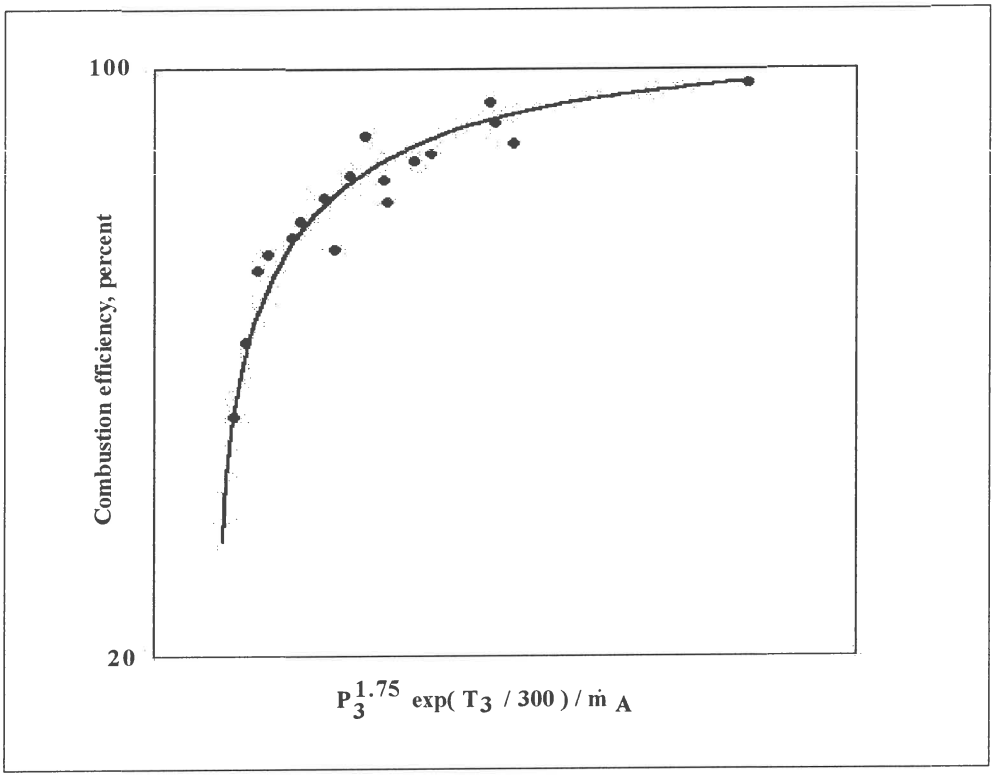

Figure 2.2 Correlation of combustion-efficiency data for an aircraft engine combustor

2.5 Summary

The topic of combustion is so vast that this chapter provided a very basic review of the operation of combustion chambers that is relevant to understanding the subsequent sections. It has been shown that for combustion to occur a low velocity flowfield must be established where stable burning over a range of different air-to-fuel ratios can exist. This recirculating flowfield is obtained by swirling air entering the chamber combined with opposed jets flowing radially inward from the liner wall. It is this process, more than anything else that controls the combustion inside combustors.

CHAPTER 3

EXPERIMENTAL MEASUREMENTS

3.1 Preamble

Numerical models have to be validated against experimental measurements in the form of temperatures, velocities and species concentration. The temperature field is the most important property and is a key component in the development of the turbine section. The need for detailed temperature measurements that capture the thermal field of a gas turbine combustor has been identified from the literature survey. In this chapter the experimental investigation of a single-can combustor is examined. The experimental setup and instrumentation used to record and monitor various parameters are discussed. The research combustor is representative of practical combustors in terms of a primary, secondary and dilution section, film cooling and swirling air. A description of the geometrical features and operating zones are given. The mass flow splits at the inlets as obtained during isothermal conditions are also presented. Experimental measurements were made using thermocouples and constitute liner wall, inner and exit gas temperatures. Each of these is discussed and the geometrical locations of the probe positions are given.

3.2 Experimental setup

Combustion primarily consists of the mixing process between fuel and air. In practical combustors the rate of airflow is usually fixed and the rate of fuel flow is varied to control the amount of heat release. The experimental setup for the actual research combustor is shown in Figure 3.1 and a schematic drawing in Figure 3.2. Air is provided by a centrifugal high-pressure low-volume blower with straight radial blades. The blower is capable of providing a static pressure above 6 kPa between mass flow rates of 0.1 and 1.0 kg/s. The rate of airflow is controlled by a frequency inverter connected to the motor and measured using an orifice flowmeter. The orifice was designed and manufactured according to BS1042 with a D and D/2 pressure tapping.

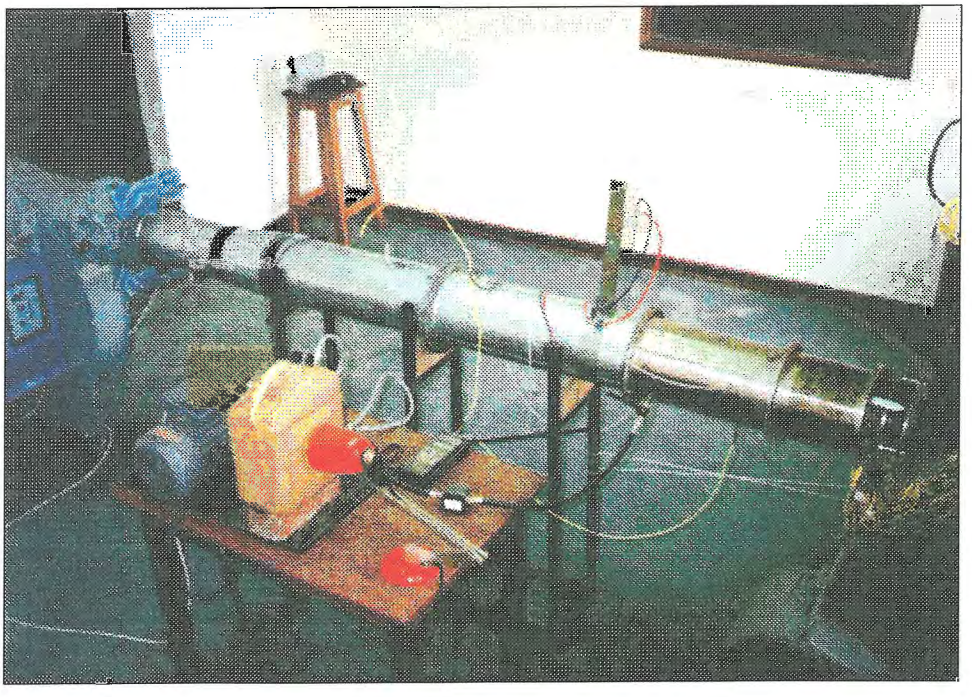

Figure 3.1 Test rig used for the experimental investigation

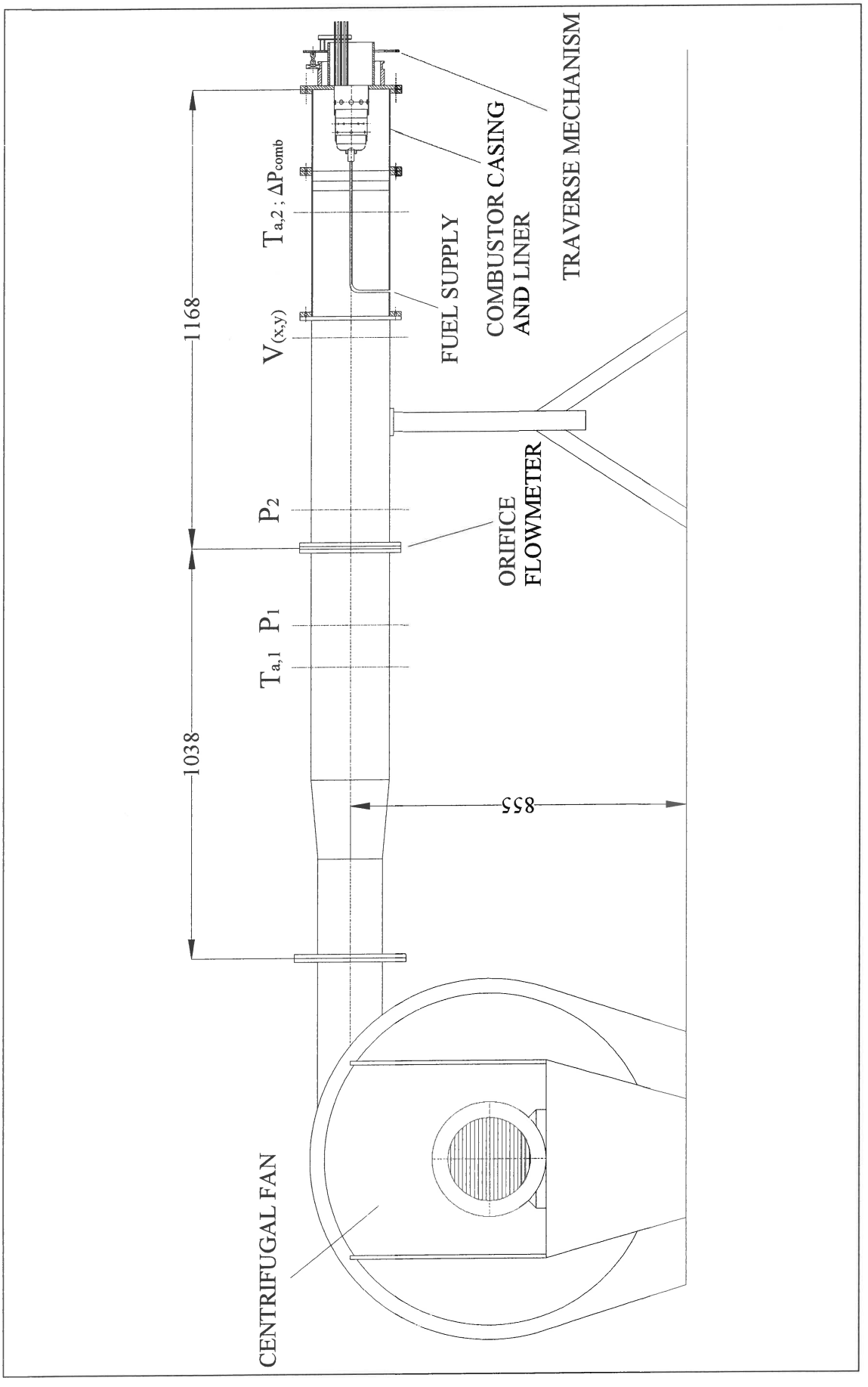

Figure 3.2 Schematic drawing of the experimental setup

The calibration factor or flow coefficient α is obtained from equation 3.1.

$$Q = \alpha A_t \left[\frac{2(\Delta P)}{\rho} \right]^{1/2} \tag{3.1}$$

Q = Volume rate of flow ($m^3.s^{-1}$)

A_t = Throat area of the obstruction (m^2)

ΔP = Pressure change (Pa)

ρ = Density ($kg.m^{-3}$)

The density was calculated from the ideal-gas equation of state with the air temperature measured upstream of the orifice flowmeter by a thermocouple. The pressure change was obtained using a calibrated digital micromanometer. It is important to obtain the correct relationship between throat area and pressure change. If the throat area of the orifice is too small, flow separation will occur resulting in a non-uniform flow profile downstream. The volume flow rate, Q, through a pipe or duct is just the integral of the velocity distribution, V(x,y), over the cross-sectional area A [32]:

$$Q = \int_A V(x,y)dA \tag{3.2}$$

The velocity distribution V(x,y) was measured at a distance of 0.53 m downstream of the orifice flowmeter. The velocities were measured vertically and horizontally every 5 mm across the diameter of the duct using a pitot tube. Small fluctuations occurred and to compensate for this the average of 180 samples were calculated at each measurement position. Equation 3.2 assumes that the velocity profile is symmetrical and from the velocities measured vertically across the duct - illustrated in Figure 3.3 - a small amount of asymmetry is evident. The volume flow rate was therefore calculated as the average obtained from the velocities in the upper half of the duct and the lower half of the duct. A flow coefficient of 0.62 was calculated from the experimental measurements and compares favourably with the theoretical value of 0.61.

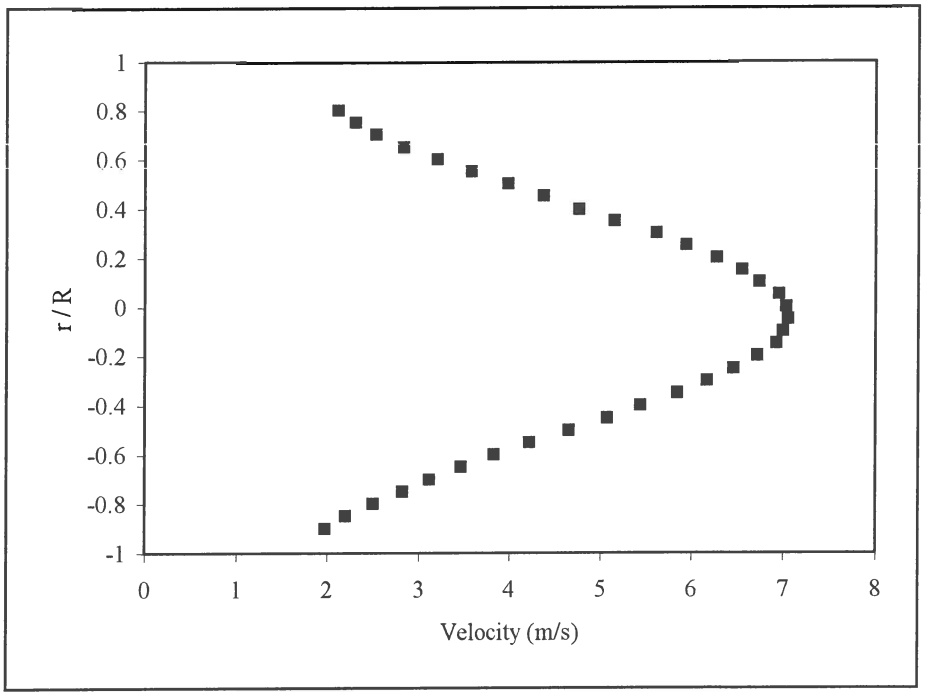

Figure 3.3 Velocity distribution measured vertically across the duct

Fuel is injected into the combustion chamber using an internal gear pump that is capable of providing a delivery pressure of 1500 kPa. The fuel flow rate is controlled by changing the delivery pressure. This is done in two ways – by adjusting the pressure on the pump and through a frequency inverter that is connected to the motor. The fuel flow rate was measured using a positive displacement flowmeter with a pulse sensor. The meter was connected to a multi-function controller with a period measurement mode. Calibration was done by measuring the mass rate of flow as a function of time. This was repeated several times to obtain the relationship between mass rate of flow and amount of pulses per period.

$$\dot{m}_f = cp \tag{3.3}$$

$\dot{m}_f$ = Mass rate of flow ($g.s^{-1}$)

c = 1.36×10^{-3}

p = Pulses per period

The fuel is transported in an uncooled fuel supply line connected to a Monarch nozzle (1.0 GPH, 90° PLP). The nozzle provides atomization of the liquid fuel at a conical angle of 90-degrees. Calibration of the nozzle was done by measuring the pressure change at various mass flow rates, shown in Figure 3.4. The pump inlet and fuel nozzle was provided with filters to prevent blockage due to contaminants present in the fuel.

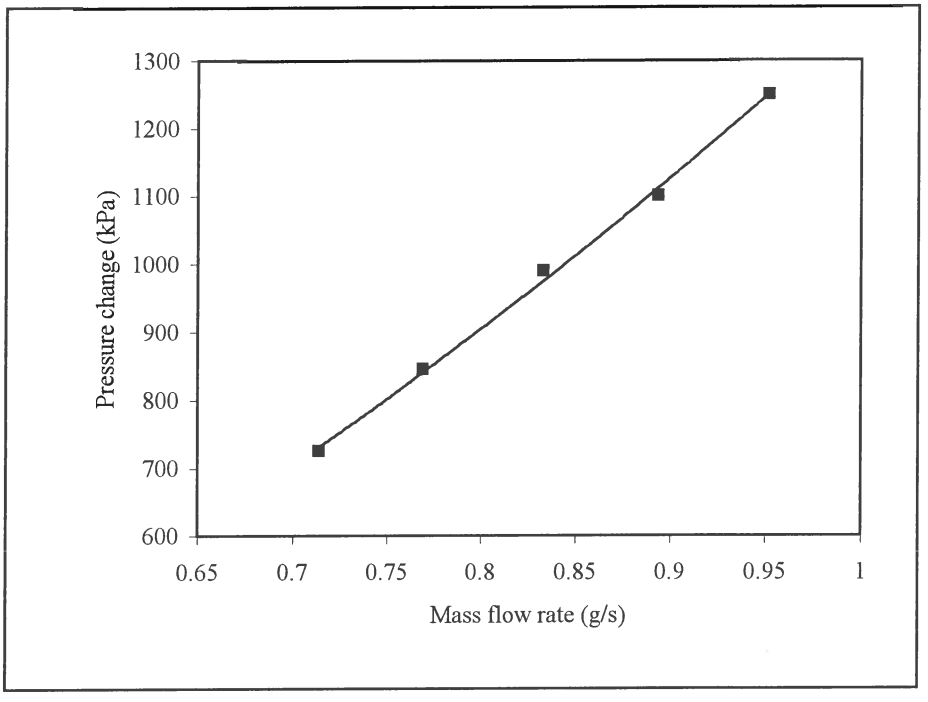

Figure 3.4 Fuel nozzle calibration curve

The inlet air temperature is a critical parameter in terms of combustion modelling and was measured 0.3 m upstream of the orifice flowmeter and 0.85 m downstream using two K-type thermocouples. The pressure change across the combustor was measured at a distance of 0.85 m downstream of the orifice flowmeter. The pressure change was measured relative to atmospheric pressure. The various temperatures experimentally measured are discussed in the subsequent sections. Three analog-to-digital cards with a total of 24 differential input channels recorded the measured temperatures. Electrical and electromagnetic noise can result in substantial measurement errors when measuring low-level thermocouple signals. To prevent errors as a result of noise the thermocouples were

insulated against ground loops and the analog-to-digital cards included filters and complete separation of analogue and digital circuitry. All data were recorded at a sampling rate of 1 Hz.

3.3 Research combustor

The research combustor shown in Figure 3.5 is made from stainless steel and coated with a thermal barrier coating (TBC) on the inside of the liner. The reference length (L) and radius (R) of the combustor are 174.8 and 41.2 mm, respectively.

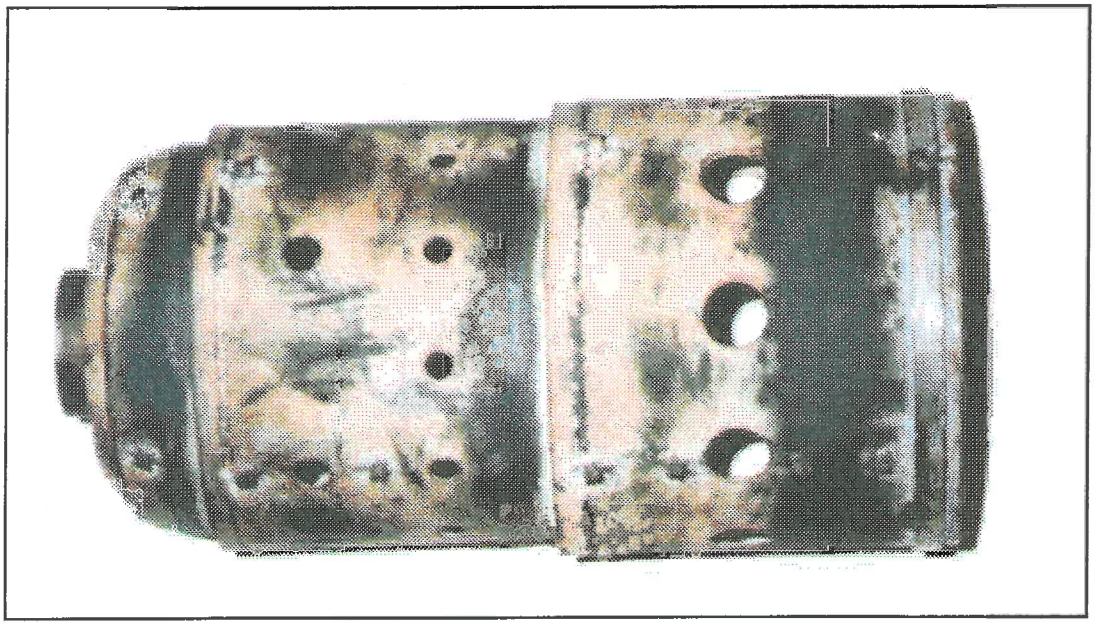

Figure 3.5 (a) Plan view of the research combustor

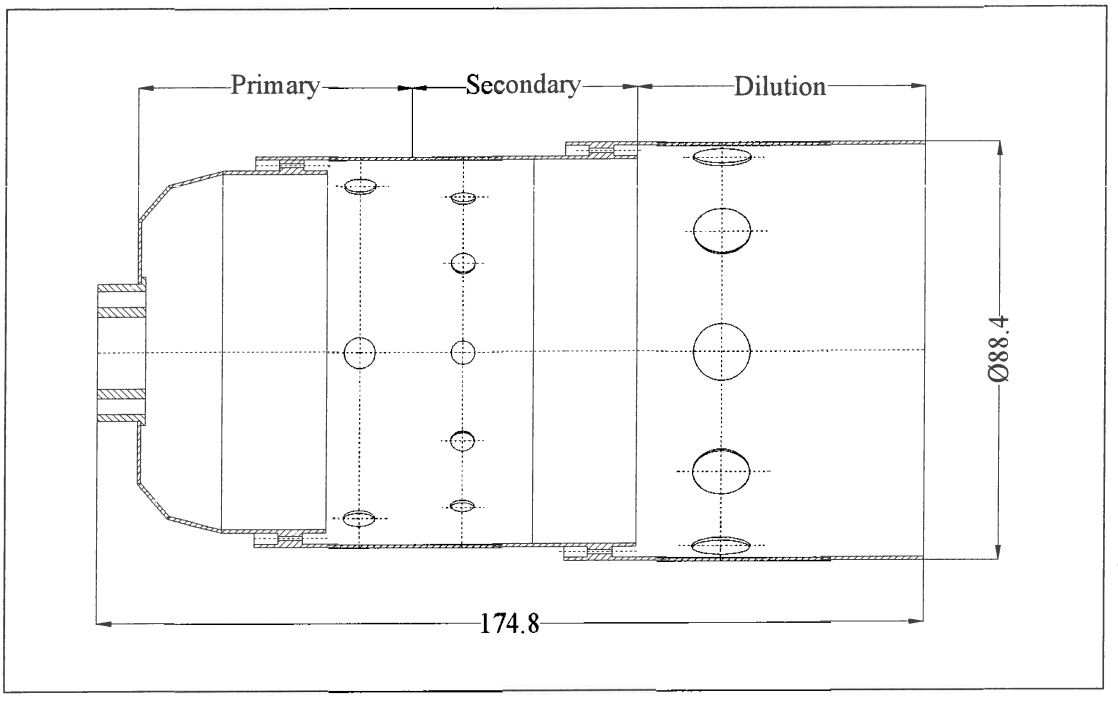

Figure 3.5 (b) Sectored view of the research combustor illustrating the various operating zones

The purpose of the TBC is to lower the emissivity on the inside of the combustion chamber, and thereby reducing the radiant heat transferred to the inner liner wall. The single can combustor is divided into the primary, secondary and dilution zone. Six jets are located in the primary zone, twelve jets in the secondary zone and ten jets in the dilution zone, ensuring good mixing between fuel an air. Film-cooling air is injected into the secondary and dilution zone using stacked rings with 40 entry holes. An axial swirler with a 45-degree swirl angle and curved blades enhances the mixing process and the development of low velocity recirculation regions. Although the combustor was tested at atmospheric conditions, the geometry is very representative of practical combustors that are characterized by a turbulent, three-dimensional swirling flow field. The combustor was designed to operate with an air mass flow rate of 0.1 kg/s. At this point an experimental pressure change of 2.32 kPa was measured across the combustor. This represents a 2.45 percent pressure change and is well within the accepted 5 percent of practical combustors. The various flow splits between the swirler, primary, secondary and dilution jets obtained during isothermal conditions together with the geometrical areas are given in Table 3.1.

Hole type	Ag (mm^2)	Mass flow %
Swirler	184.19	8.35
Cooling I	25.45	1.73
Primary	205.27	12.50
Secondary	235.62	15.26
Cooling II	25.45	1.74
Dilution	1084.34	60.52

Table 3.1 Isothermal mass flow splits and geometrical areas

The primary zone operates at stoichiometric conditions. The advantages include a high volumetric heat-release rate and a relatively low rate of carbon formation; the primary zone can consequently be made small, and the combustion process is characterized by freedom from coke deposition and exhaust smoke. Its main disadvantages are a high rate of heat transfer to the liner walls and high emission on nitric oxides [30]. The combustor is designed to operate on both liquid and gaseous fuel and all the results recorded were obtained using kerosene. The operating conditions are determined from the stability loop. Using a fixed air mass flow rate, and varying the fuel flow until extinction occurs, the stability loop shown in Figure 3.6 was obtained. It is evident that very lean air-to-fuel ratios can be obtained with the type of liner material determining the limit for rich air-to-fuel ratios. The diameter ratio of the combustor casing relative to the combustor liner was equal to 2.42. This is large compared to normal practice and was done in an effort to reduce the influence that the measuring equipment has on the airflow in the annulus.

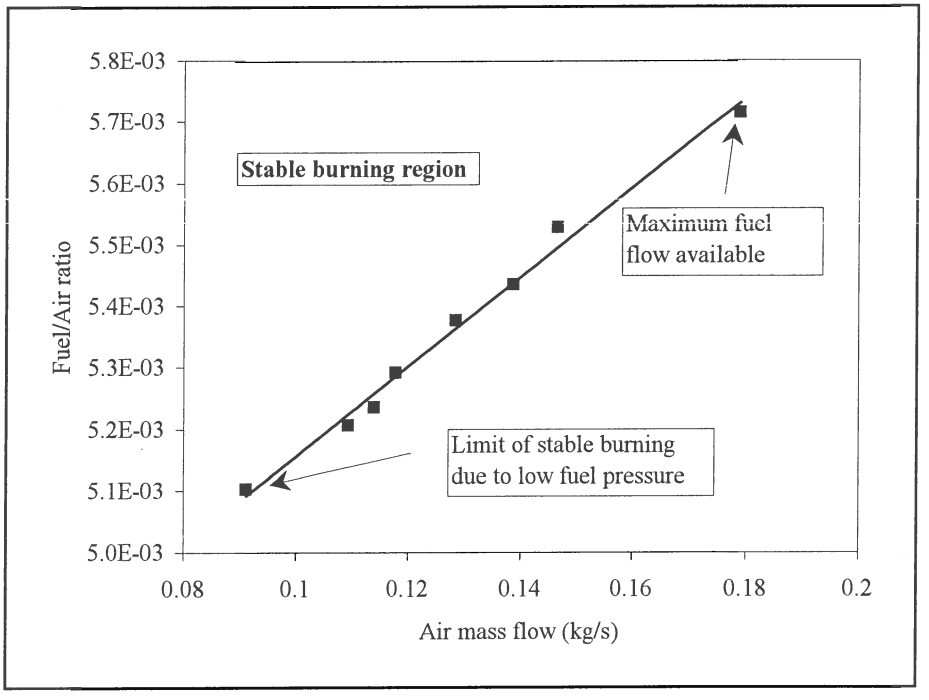

Figure 3.6 Combustion chamber stability loop

3.4 Liner wall temperatures

Twenty temperatures on the outer liner wall were measured using K-type thermocouples. Each thermocouple was protected by a 1.5 mm diameter 300 mm long stainless steel sheath. The thermocouples were attached to the liner wall by means of a small bracket. The respective geometrical measurement positions are shown in Figure 3.7. The combustor was divided into four positions, with each position at an angle of 40-degrees from the horizontal centerline. At position one, eight thermocouples were placed axially along the combustor from the primary to the dilution zone. At positions two, three and four, four thermocouples were placed that coincided with measurement points at position one. The placement of the measurement points was done so that the distribution of wall temperatures from the primary to the dilution zone could be captured, as well as the distribution of temperatures along the perimeter of the combustor.

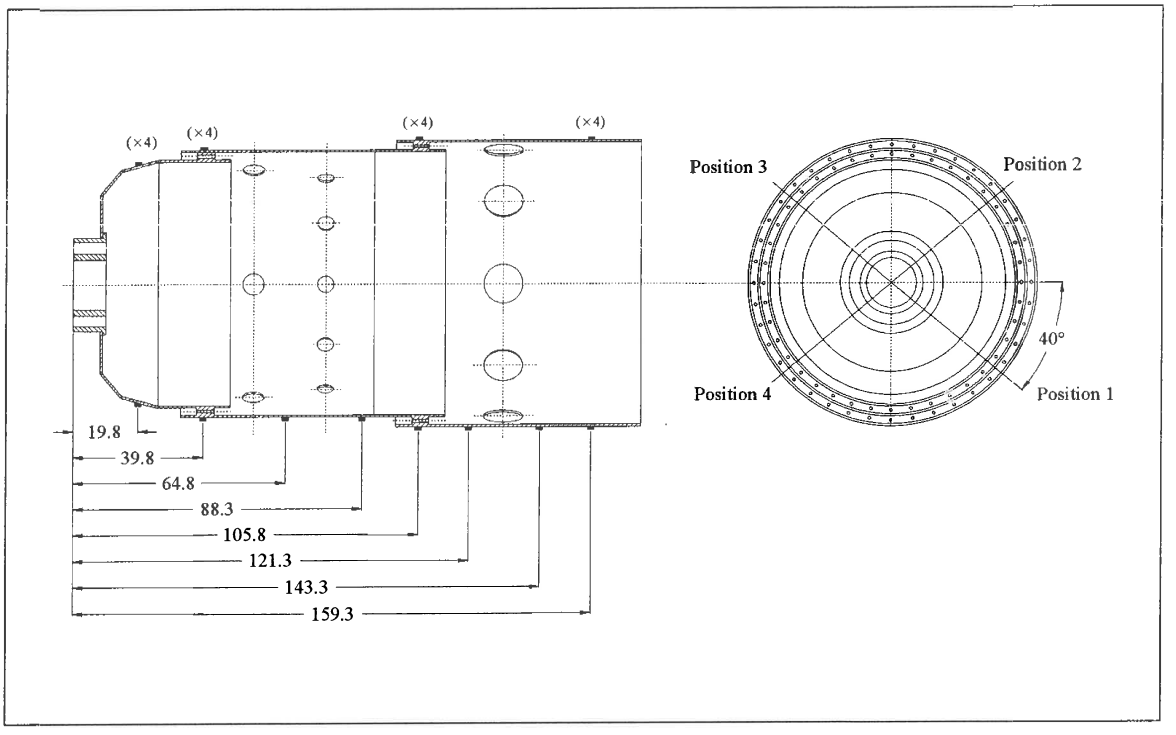

Figure 3.7 Combustor liner wall temperature measurement positions

3.5 Exit gas temperatures

The exit gas temperatures were measured using five K-type thermocouples protected in a 1.5 mm diameter, 300 mm long stainless steel sheath. The thermocouples were fitted with radiation shields and positively ventilated, in an effort to render true convective gas temperatures. The thermocouples were equi-spaced along the radius of the combustor at the outlet plane and mounted on a traverse mechanism that enabled measurements to be taken at 10-degree intervals, Figure 3.8. A total of 180 temperatures were recorded to capture the temperature distribution across the outlet plane of the combustor.

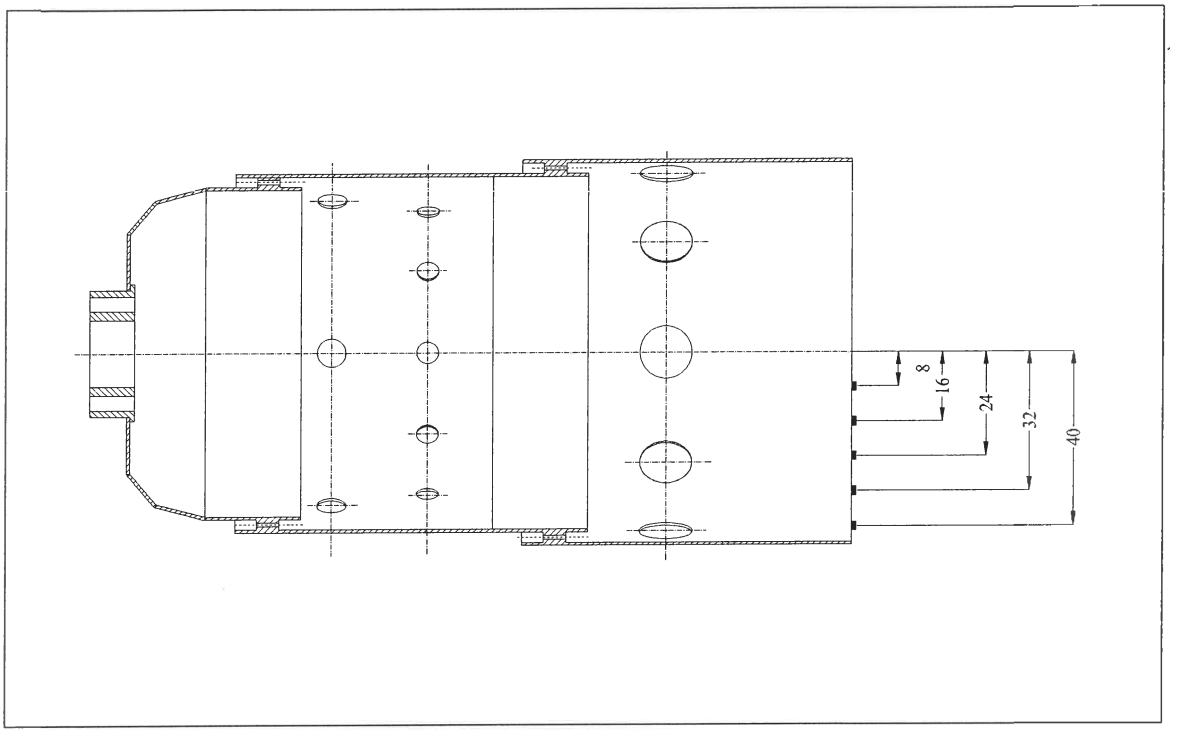

Figure 3.8 Exit gas temperature measurement positions

3.6 Inside gas temperatures

Enclosed casings and very high gas temperatures inside combustors have resulted in very few detail measurements of the inside temperatures. Operating at lean overall conditions and with the exit plane of the combustor open to atmosphere, the research combustor provided the ideal opportunity to obtain detail measurements of gas temperatures. Temperatures were measured using an R-type thermocouple with a maximum continuous working temperature range of 0 to 1600-degrees Celsius. The thermocouple is housed in a 4.6 mm diameter 300 mm long ceramic sheath and the hot junction is therefore not directly exposed to radiant heat transfer. The geometrical measurement positions are shown in Figure 3.9.

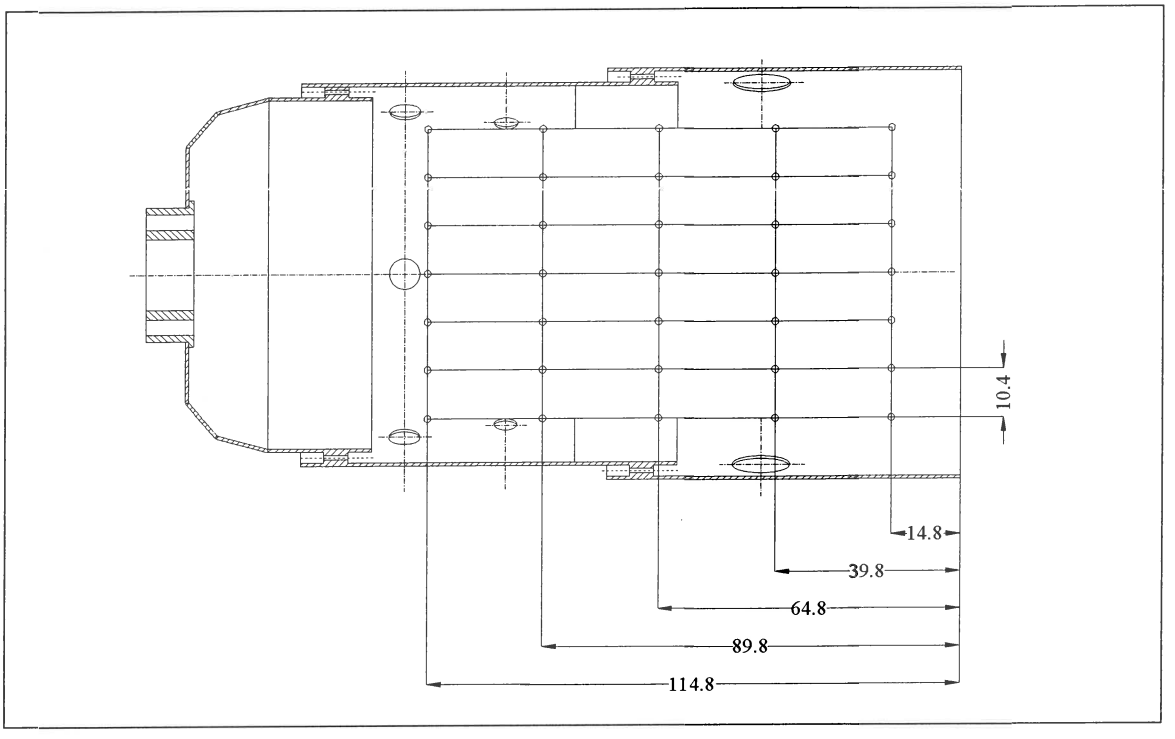

Figure 3.9 Inside gas temperature measurement positions

Measurements were taken at five axial planes from the secondary to the dilution zone. The thermocouple was mounted on the traverse mechanism and measurements were taken at 30-degree intervals. A total of 48 temperatures were recorded at each plane to capture the temperature distribution of inside gas temperatures. One thermocouple was used to obtain the measurements. This means that it is a very time-consuming process, and although measurements could have been taken at smaller angles the 30-degree intervals were used so that all the measurements could be obtained in one test session.

3.7 Summary

This chapter described the test rig used to obtain physical measurements for a can type combustor during operating conditions. The main features of the setup were discussed and a description of the research combustor was given, illustrating the different zones and film cooling devices employed. The various measurements that were recorded as well as

the geometrical positions were examined. Although atmospheric test conditions were used, the geometry of the chamber is representative of practical combustors.