FLOXCOM FINAL REPORT

FLOXCOM

LOW-NOx FLAMELESS OXIDATION COMBUSTOR FOR HIGH EFFICIENCY GAS TURBINES

PROJECT N°: NNE5-1999-20246

CONTRACT N°: ENK5-CT-2000-00114

Final Technical Report (50 pp)

PROJECT COORDINATOR: TECHNION

ISRAEL INSTITUTE OF TECHNOLOGY

CONTRACTORS:

Israel Institute of Technology (Technion)	**IL**
Imperial College of Science Technology & Medicine (ICSTM)	**UK**
CINAR Ltd. (CINAR)	**UK**
Instituto Superior Tecnico (IST)	**P**
Institute of Fundamental Technological Research (IPPT-PAN)	**PL**
ANSALDO Ricerche Srl. (Ansaldo)	**I**
B&B AGEMA Gmbh (B&B_AGEMA)	**G**
Rheinish-Westfaelische Technische Hochschule Aachen (RWTH)	**DEAnsaldo**
Caldaie (CCA) I	

Reporting Period: From 1/11/2000 to 1/12/2003

Project Start Date: 1/11/2000 Duration: 36 MONTHS

Date of issue of this report: December 21, 2004	Project funded by the European Community under the “FP5 -EESD” Program (1998-2002)

FLOXCOM Participant Contact List

Partner #	Institute / Organization	Short Name	Street Name & Number	Post Code	Town/ City	Country Code	Title	Family Name	First Name	Telephone No.	Email
1	Technion - Israel Institute of Technology	Technion	Technion City	3200	Haifa	IL	Prof.	Levy	Yeshayahou	972-4-8293807	levyy@aerodyne.technion.ac.il
2	The Imperial College of Science, Technology and Medicine	ICSTM	Exhibition Road South Kensington	SW7 2AZ	London	UK	Prof.	Lockwood	Fred C	44 207 594 7032	f.lockwood@ic.ac.uk
3	CINAR Ltd	CINAR	11 Elvaston Place	SW7 5QG	London	UK	Dr.	Abbas	Tahir	44 20 7581 2245	tahir@cinar.co.uk
4	Instituto Superior Tecnico	IST	AV. ROVISCO PAIS	1049-001	Lisbon	P Portugal	Prof.	Coelo	Pedro	351-21-8418194	coelho@navier.ist.utl.pt
5	Institute of Fundamental Technological Research, Polish Academy of Sciences	IPPT-Pan	Swietokrzyska 21	00049	Warsaw	PL	Docent	Kowalewski	Tomasz	(48-22) 8269803	tkowale@ippt.gov.pl.
6	Ansaldo Ricerche s.r.l.	ARI	Corso Perrone 25	16152	Genova	I	Dr.	Mori	Giulio	39-010-6558383	mori@ari.ansaldo.it
7	B&B-AGEMA, Gesellschaft für Energietechnische Maschinen und Anlagen, Aachen GmbH	AGEMA	Jülicher Strasse 338	D-52070	Aachen	DE	Dr.-Ing.	Kusterer	Karsten	49 (241) 56878-20	kusterer@bub-agema.de
8	Rheinisch-Westfaelische Technische Hochschule Aachen / Aachen University of Technology	RWTH	Templergraben 55	52056	Aachen	DE	Univ.-Prof. Dr.-Ing	Bohn	Dieter	49-241-805451	post-bohn@idg.rwth-aachen.de
9	Ansaldo Caldaie	CCA	Via Milano Km 1600 70023 Gioia del Colle (BA)	70023	Bari	I	Dr.	Saponaro	Alessandro	39-080-3480246	Alessandro.Saponaro@ansaldoboiler.it

Table of Contents

Part 1: Publishable Final Report (NON CONFIDENTIAL)

1.1 Executive publishable summary

1.2 Publishable synthesis report

Part 2: Detailed Final Report (CONFIDENTIAL)

2.1 Objectives and strategic aspects

2.2 Scientific and technical description of the results

2.3 Assessment of Results and conclusions
2.4 Acknowledgements

2.5 References

Part 3: Management Final Report (CONFIDENTIAL)

3.1 List of deliverables

3.2 Comparison of initially planned activities and work actually accomplished

3.3 Management and co-ordination aspects

Part 1: Publishable Final Report **(NON CONFIDENTIAL)**

1.1 Executive publishable summary

Higher turbine entry temperatures, a prime factor in increased efficiency, tend to increase the NO_x emission. The project offers a technologically innovative gas turbine combustion solution, the flameless oxidation, for elevated operating temperatures while maintaining low-NOx emissions. This enables to produce clean energy power plants while burning regular or low-grade synthetic fuels from renewable sources and running at lower costs thus increasing EU market competitiveness and employment. The main objective was to pursue studies required to validate the engineering feasibility of flameless oxidation technology and to produce different combustors to demonstrate its improved performance. The theoretical results are validated through detailed experimental analysis of different sector combustors that were operated at different conditions and by different partners. The tests performed with the combustor sectors showed very stable combustion over a wide range of equivalence ratio and pressures with low NO_x emission. Further optimisation study should be dedicated in order to use the technology in practical gas turbines and aero engines.

1.2 Publishable synthesis report

The project offers a technologically innovative gas turbine combustion solution, the flameless oxidation, for elevated operating temperatures while maintaining low-NOx emissions. This enables to produce clean energy power plants while burning low-grade synthetic fuels from renewable sources and running at lower costs, thus increasing EU market competitiveness and employment. The main objective was to complete studies required to validate the engineering feasibility of flameless oxidation technology for gas turbines and to produce operating model combustors to demonstrate its improved performance. The theoretical results are validated through detailed experimental analysis of different sector combustors. Extensive operational performance demonstration of the advanced design combustor was carried out.

The results of the FLOXCOM project include a novel computational model for accurate combustor design; innovative technologies for wall cooling and fuel injection; two units of sector combustors, operating at atmospheric and elevated pressure and a full-scale pilot combustor with high temperature capability.

These results are itemized as follows:

- Improved combustion engineering methods and advanced design tools;
- Improved availability and reliability of the combustor obtained through lower pattern factor and thermal stresses;
- Proven technology for high efficiency small gas turbines (>35% in open cycle) and low-NOx (<20 ppm).

Continuation of the project for the design of a new GT based on the FLOXCOM technology is suggested. Preliminary contacts are already being made with different industries within the EC to establish a new consortium for the exploitation of this new technology. The target market is the small GT industry (200-1000 KW range), most probably for decentralized power generation.

1. The method in the FLOXCOM project for clean and efficient gas turbines, operating at high temperatures, is based on a technologically innovative combustion solution, the Flameless Oxidation method. The combustion method was recently approved as a patent, "Novel Design of Adiabatic Combustor", Levy, Y. and Arfi P., US Patent (Pending) No. 2002/0069648 A1, June 13 2002..

This promising technology allows operation of the combustor at high temperatures with ultra-low NOx levels. In addition, this combustion mode has further advantages over other advanced NOx reduction technology such as safety, reliability and the possibility for its incorporation in a heat exchanger cycle using high air temperatures at the combustor inlet.

The investigation was directed toward the completion of the studies required to validate the engineering feasibility of the Flameless Oxidation technology and to produce operating pilot combustors that will show its improved performance. The project makes use of innovative combustion predictions and advanced technologies of fuel injection and cooling, developed to fit the Flameless Oxidation combustor's specific needs. The theoretical results are validated through detailed experimental analysis of different sector combustors. Full-scale tests of pilot combustors are also scheduled. The work was equally divided between theoretical studies and experimental verifications. It commenced with basic studies required to improve understanding on the interactions between turbulence and combustion. This study is coupled to well defined and controlled combustion laboratory experiments. Basic studies also have been performed for an innovative fuel atomization method, which also serves as a momentum accelerator for the main vortex which is playing a major role in the operation of the newly developed combustor, while maintaining circumferential uniformity. An additional, detailed investigation was conducted to develop wall-cooling methods where the small-wall jets were optimized for maximizing the combined effects of the vortex momentum augmentation and wall temperature reduction and unification. The vortex characteristics were quantitatively visualized within transparent flow models using the PIV technique.

Milestones and Results:

The major milestone achieved throughout the project includes the following:

- Completion of a combustion prediction procedure for the internal aerodynamics and emission under flameless oxidation operating conditions
- Completion of an improved CFD procedure for a gas turbine combustor operating under extreme flameless oxidation conditions.
- Evaluation of the performance of the wall-cooling model.
- A validated and tested design for an efficient and low-emission gas turbine combustor.
- Testing of the combustor sector prototypes at atmospheric and elevated pressures.
- Experimental demonstrations of the main recirculation pattern, fuel injection and wall cooling characteristics.
- Manufacturing the pilot combustor.
- Final recommendations for future combustor of the following project, including the combustor’s geometry, construction and aerodynamics, will be determined.

Tests results are used to optimize the combustor geometry for minimum emission and maximum combustion stability, uniformity of wall temperature and circumferential distribution of exhaust gas temperature profiles. Different combustor models were produced, for detailed point measurements of velocities, temperatures and species concentration. The tests were performed under a variety of operating parameters including chemical reactive flows and pressurized conditions. The test results are used for further adjustments of the different models and for comparison with the CFD predictions. A complete pilot combustor prototype was produced The different combustor models were and will still be used for detailed and global performance measurements such as the internal flow field (single and two phase flows), combustion efficiency and stability, emission, circumferential variation of the exhaust gases temperature, internal and wall temperature distribution and more. Significant progress is also expected in combustion engineering related technologies through the gained knowledge.

This project tested and verified the new combustor technology. However, optimizations of various design parameters and endurance testing are still needed to complete the combustor design.

FLOXCOM maintains an updated web page (http://floxcom.ippt.gov.pl/) for dissemination of results.

Part 2: Detailed Final Report **(CONFIDENTIAL)**

2.1 Objectives and strategic aspects

The project offers a technologically innovative gas turbine combustion solution, the flameless oxidation, for elevated operating temperatures while maintaining low-NOx emissions. This enables to produce clean energy power plants while burning regular or low-grade synthetic fuels from renewable sources and running at lower costs thus increasing EU market competitiveness and employment. The main objective was to complete studies required to validate the engineering feasibility of flameless oxidation technology for gas turbines and to produce operating model combustors to demonstrate its improved performance. The theoretical results are validated through detailed experimental analysis of different sector combustors. Extensive operational performance demonstration of the advanced design combustor was carried out.

The results of the FLOXCOM project include a novel computational model for accurate combustor design; innovative technologies for wall cooling and fuel injection; two units of sector combustors, operating at atmospheric and elevated pressure and a full-scale pilot combustor with high temperature capability.

These results are itemized as follows:

- Improved combustion engineering methods and advanced design tools;
- Improved availability and reliability of the combustor obtained through lower pattern factor and thermal stresses;
- Proven technology for high efficiency small gas turbines (>35% in open cycle) and low-NOx (<20 ppm).

A continuation of the project for the design of a new GT based on the FLOXCOM technology is suggested. Preliminary contacts are already being made with different industries within the EC to establish a new consortium for the exploitation of this new technology. The target market is the small GT industry (200-1000 KW range), most probably for decentralized power generation.

2.2 Scientific and technical description of the results

The description of the work performed is organized according to the Work Packages (WP), as described in the document: “Description of Work”. In the following is a table summarizing the different WPs, the corresponding WP leaders and, were relevant, the additional partners participating in that WP.:

LIST OF WORKPACKAGES

WP No[1]	Work package title	WP LEADER	Additional Participant:
Coor.	Coordination	TECHNION	
1	Combustor Sectors Design and Manufacture	TECHNION	
2	Investigation of Flameless Oxidation Fundamentals & Development of Engineering Prediction Tools	ICSTM	TECHNION
3	Integration of Developed Mathematical Models and Combustor Design	CINAR	
4	Fuel Injection System Development	IST	
5	Laser Diagnostics of Vortical Flow	IPPT-PAN	
6	Hot Pressurised Tests of the Combustor Sector	Ansaldo Ricerche S.r.l.	Ansaldo Caldaie
7	Modelling and optimisation of convective wall cooling, wall temperature and stress analysis, turbine interface	B&B AGEMA	RWTH
8	Pilot Combustor Testing	RWTH	B&B AGEMA
9	Pilot Combustor Manufacture.	TECHNION	

2.2.1 WP1 Combustor Sector Design and Manufacture

WP leader -Technion

2.2.1.1 Specific Project Objectives

1. To analyze the numerical and experimental results achieved and integrate the different WP's periodically for the design of the combustor sectors.
2. To design and build the combustor sectors for testing at Ansaldo Caldaie and for IST and negotiate with the sub-contractors for the provision of accessories for a high-pressure combustor sector to be tested within WP6.

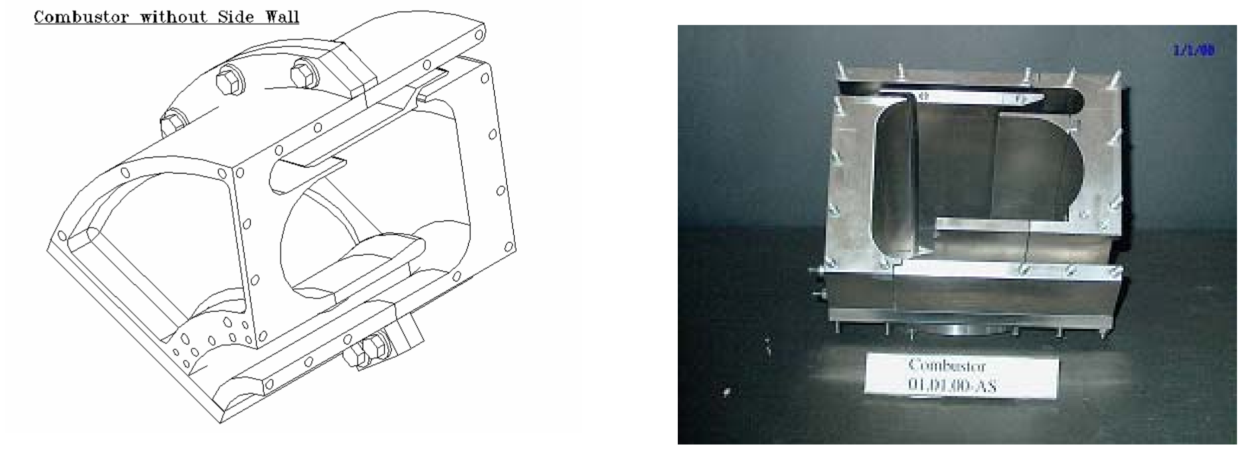
The work package is concerned with the design of the combustor sectors, based on the analysis of the numerical and experimental results achieved. It also concerns the design and construction of combustor sectors for ANSALDO and for IST including the provision of accessories for the high-pressure combustor sector.

2.2.1.2 WP1 Overview of Scientific and Technical Progress and Results

Preliminary experimental CFD calculations were performed by the Technion. Based on this, a preliminary combustor configuration was determined.

Once the configuration was set, a “Phase A” - Engineering model for the combustor sector has been planned and then sent to B&B AGEMA for observation and comment regarding the thermal stress issues. The design was modified according to AGEMA’s comments, even though, the shape stayed the same. Detailed design of the combustor sector was carried out and construction is underway.

Two units of the combustor sector were assembled and are to be operated by IST - Portugal and by Ansaldo - Italy.



a) Design

b) Combustor Sector Model

Fig 1.1 The combustor sector.

A low pressure testing facility was designed for IST for the purpose of checking the spraying and combustion processes in atmospheric pressure.

Both the low pressures as well as the elevated pressure test sectors are designed to operate within a testing facility - a pressure vessel with transparent walls. An elevated pressure testing facility was designed and is to be operated by Ansaldo Italy. The same design principal was applied for atmospheric pressure testing facility and is intended for IST.

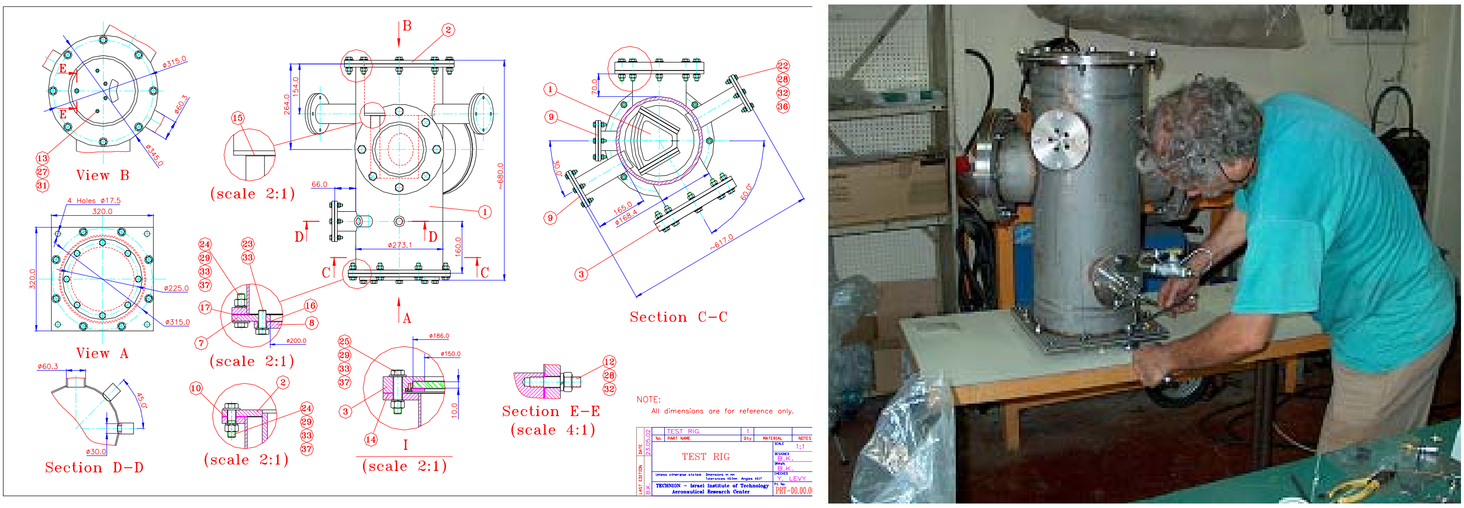


Fig 1.2 IST Test rig for hot atmospheric testing, design and actual

Both low and high-pressure testing facilities designs have been sent to CTC Ltd, UK, the subcontractor for evaluation of operational safety. The results and recommendations of the CTC critiques (Phase I and Phase II) were implemented. These changes mainly referred to the exhaust gases cooling section as well as to the thickness of the flanges. In addition, due to late requirement of the insurance company, a more detailed review of the engineering design was imposed, to comply with the ASME standard. This required an additional and detailed review that was

In the following details of the operating conditions and images of the combustor and the test rigs at different stages of assembly are given. The Operating Conditions of the rig are summarized below:

Table: the operating conditions of the combustor and the test rigs

	Nominal	**Units**
Sector dimension	60^0	-
Total air flow rate	0.1296	Kg/s
Temperature air inlet	330	K
Maximum air Pressure at inlet	120	KPa
Inlets		
Air 1		
Flow rate	0.128	Kg/s
Temperature	330	K
Pressure	120	KPa
Air2		
Flow rate	0.0016	Kg/s
Temperature	330	K
Pressure	120	KPa
Air3		
Flow rate	-	Kg/s
Temperature	-	K
Pressure	-	KPa
Fuel		
Methane	CH4	
Flow rate	0.00256	Kg/s
Temperature	300	K
Outlet		
Flow rate	0.13216	Kg/s
Temperature	1200	K
Pressure	100	KPa

Fig 1.3 The combustor sector test rig, assembled and incorporating the combustor sector.

In figure 1.3, several components can be seen including: the optical windows, the main air inlets, the ignition power supply, thermocouples connectors for monitoring the air temperature within the test rig and the temperature of the interface section as well as the supply ports for the secondary air, fuel, window cleaning air and high voltage supply for the igniters. Completion of design of the combustor test rig section at elevated pressure to be utilized at Ansaldo Italy.

Fig 1.4 The assembled combustor sector, mounted on the interface section to the Combustor test rig.

Fig 1.4 also shows the supply line for fuel and secondary air along side the thermocouple wire for monitoring the temperature of the interface unit.

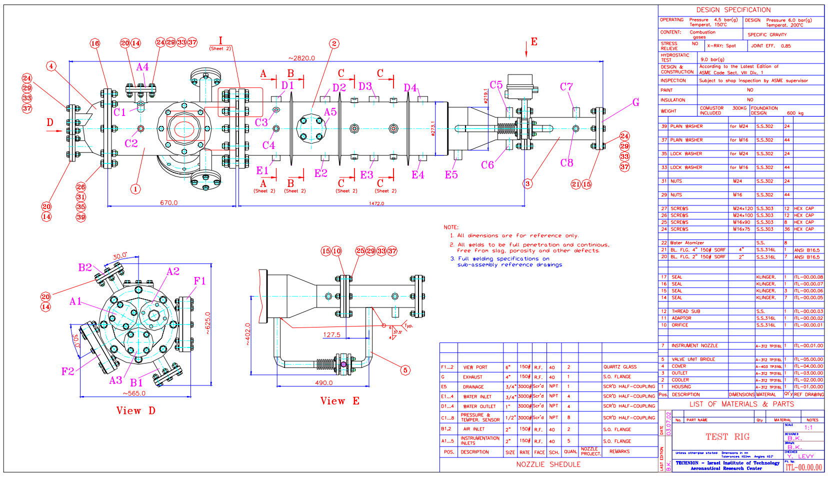


Fig 1.5 Completion of design of the combustor test rig section at elevated pressure to be utilized at Ansaldo Italy**.**

Construction of the combustor test rig section at elevated pressure has been finalized.

Fig 1.6 Main structure of the combustor test rig, complete and hydrostatically tested for 12 bars

Fig 1.7 Combustor sector assembled within test rig.

Fig 1.8 Combustor sector assembled within test rig, piping connections.

In the figure, supply lines for the gaseous fuel, secondary air and air for window cleaning, as well as the input ports can be seen. The optical window is replaced by metal flanges for pressure tests and shipping.

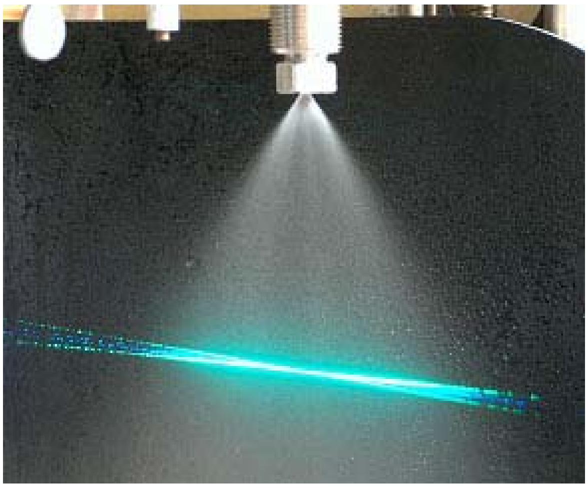


Fig 1.9 A. the atomizer for spraying water designed and built by the Technion for cooling the exhaust gases from the combustor test rig.

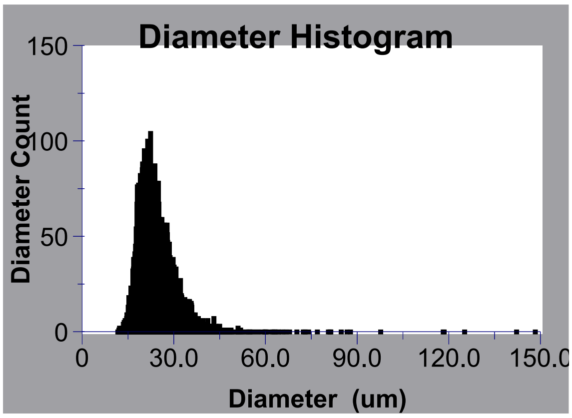


Fig 1.9 B. Size distribution of the droplets as measured by the PDPA system

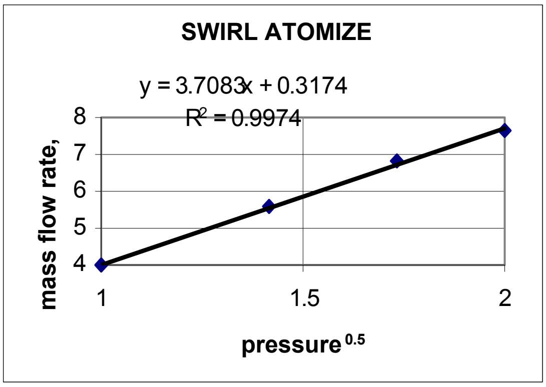


Fig 1.9 C. Flow rate characteristic of the atomizer

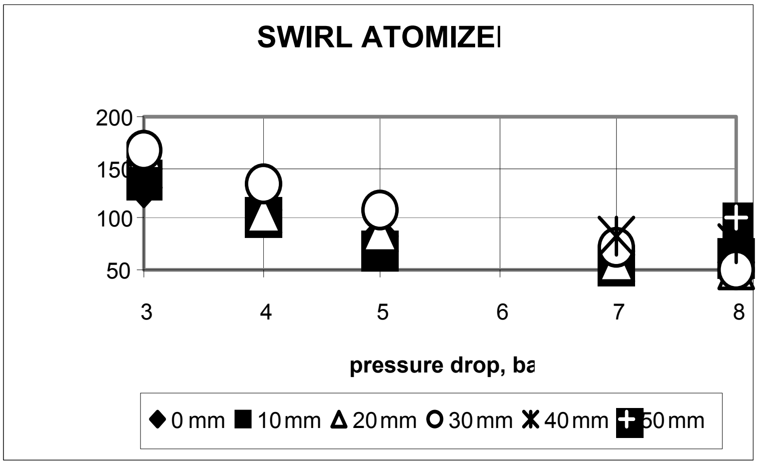


Fig 1.9 D. Variation of the mean droplet diameter (SMD) with fluid pressure

WP2: Investigation of Flameless Oxidation Fundamentals &Development of Engineering Prediction Tools

WP Leader: ICSTM,

Additional. Participant: TECHNION

2.2.2 WORK PACKAGE 2

2.2.2.1 Objectives and strategic aspects

The main objective for the Work Package 2 (WP2) in the FLOXCOM project is to investigate the fundamentals of flameless oxidation and to develop a mathematical model for its prediction. The main objective is broken down in to five sub-objectives below:

1. To devise improved modeling for the turbulence/chemistry interaction applicable to flameless oxidation.
2 To develop and validate a NOx prediction model using adequate chemistry for the FLOX mode with vitiated air.
3 To incorporate this modeling into an existing three-dimensional computational fluid mechanics (CFD) prediction code.
4 To test the resulting prediction procedure against laboratory data for turbulent combustion.
5 To deduce from this application generally applicable recommendations for enhancing the mixing, combustion and emissions performance of gas turbine combustors.

2.2.2.2 Scientific and technical description

Work description and methodology (ICSTM)

The WP2 work carried out by Imperial College London (ICL) involved the improvement and the validation of turbulence combustion models and pollutant emission models that is specific to the flameless oxidation mode of combustion. The improved mathematical models were incorporated in a three- dimensional CFD code and validated against experimental data available in scientific literature and measured data obtained from a test combustor device designed by Technion to operate in the flameless oxidation mode. The work package (WP2) was divided into the following sub-tasks:

Task 2.1. Development of an improved and rapid solution algorithm for the pdf transport equation.

Task 2.2. Incorporation of a reduced chemistry reaction scheme and extinction model to predict FLOX combustion mode.
Task 2.3. Development of an advanced pollutant prediction model with adequate chemistry for FLOX conditions.
Task 2.4. Combined experimental and numerical study of FLOX fundamentals.
Task 2.5. Incorporation of the developed combustion modeling within an existing three-dimensional CFD predictive code and validation of the combustion model.

Methodology

A mathematical procedure developed to predict the combustion characteristics of flameless oxidation has been validated against three separate experiments [1], [2], [3]. The detailed chemical reactions involving chemical species and their elementary reactions were incorporated in a flamelet modeling approach in order to account for finite rate chemistry. Detailed analysis of the results was carried out to verify the applicability of the model to predict flameless combustion. The global nitric oxide scheme is also tested against the NOx emissions data obtained from these experiments.

Mathematical models

The flow solver developed in Imperial College by [4] is employed to calculate the flow field, combustion and heat transfer processes. The code solves the balance equations for mass continuity, momentum, enthalpy and scalar transport. Scalar transport equations include the governing equations for turbulence kinetic energy, dissipation, radiation and a conserved scalar, i.e. mixture fraction for combustion. Turbulence is modeled using the standard k-ε model [5]. Thermal radiation is determined by the non-equilibrium diffusion radiation model [6].

In the simple global fast chemistry model, thermochemical variables are obtained directly from mixture fraction with the averaged thermochemical properties obtained by adopting the beta-shape probability density function (pdf) approach.

In the finite rate chemistry model, the flamelet modeling approach to turbulent combustion is employed. The model is based on two scalar fields: the mixture fraction and reaction progress variable, originally proposed by [7]. In the present study, computation of detailed chemistry based on the laminar flamelet concept is considered, which enables incorporation of rate controlling chemical effects. The present combustion model may be briefly summarised as follows. Modelled transport equations are solved for the mean mixture fraction, $\widetilde{f}$, its variance, $\widetilde{f''^2}$, and the mean reaction progress variable, $\widetilde{r}$. These equations may be expressed as:

$$\frac{\partial}{\partial t}(\overline{\rho}\widetilde{f}) + \frac{1}{\sqrt{|G|}}\frac{\partial}{\partial \overline{x}^j}\left\{\sqrt{|G|}\left[\rho v_m \widetilde{f} - \Gamma_{eff} J_\alpha^m \frac{\partial \widetilde{f}}{\partial \overline{x}^\alpha}\right] J_m^j\right\} = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\overline{\rho}\widetilde{f''^2}) + \frac{1}{\sqrt{|G|}}\frac{\partial}{\partial \overline{x}^j}\left\{\sqrt{|G|}\left[\rho v_m \widetilde{f''^2} - \Gamma_{eff} J_\alpha^m \frac{\partial \widetilde{f''^2}}{\partial \overline{x}^\alpha}\right] J_m^j\right\} = C_{g_1}\frac{\left\|\Gamma_{eff} J_\alpha^m \frac{\partial \widetilde{f}}{\partial \overline{x}^\alpha}\right\|^2}{\Gamma_{eff}{}^2} - C_{g_2}\rho\frac{\varepsilon}{k}\widetilde{f''^2} \qquad (2)$$

$$\frac{\partial}{\partial t}(\overline{\rho}\widetilde{r}) + \frac{1}{\sqrt{|G|}}\frac{\partial}{\partial \overline{x}^j}\left\{\sqrt{|G|}\left[\rho v_m \widetilde{r} - \Gamma_{eff} J_\alpha^m \frac{\partial \widetilde{r}}{\partial \overline{x}^\alpha}\right] J_m^j\right\} = C_r \rho_u \frac{S_L^0}{V_K}\frac{\varepsilon}{k}\widetilde{r}(1-\widetilde{r}) \qquad (3)$$

In the Eq. 3 for the reaction progress variable, the source term is modeled according to Lindstedt and Vaos [8], which incorporates both the classic eddy-break up concept and the influence of strain on the flamelets by the small scale eddies.

Assuming that detailed instantaneous species concentrations $Y(r, f, \chi)$ of the kinetic mechanism are known and that the state of the mixture at any spatial location is composed of chemically reacted and inert-mixed flow states, the average species concentration $\widetilde{Y} = \int_0^\infty \int_0^1 \int_0^1 Y(r,f,\chi)\,\mathrm{P(r,f,}\chi)\,\mathrm{dr\ df\ d}\chi$ may be computed as,

$$\widetilde{Y} = \widetilde{r}F(\widetilde{\chi})\int_0^1 Y_R(f)\frac{f^{a-1}(1-f)^{b-1}}{\int_0^1 f^{a-1}(1-f)^{b-1}\,df}\,df + (1-\widetilde{r})(1-F(\widetilde{\chi}))Y_M(\widetilde{f}) \qquad (4)$$

where, $F(\widetilde{\chi}) = \frac{1}{2}\left\{erf\left[\frac{1}{2\sqrt{2}}\left[\log_e\frac{\chi_q}{\widetilde{\chi}} + 2\right]\right] + 1\right\}$ (5)

and where, $Y_R(f)$ and $Y_M(f)$ represent the reacted and quenched flow states. In the above expression statistical independence is assumed for $P(r, f, \chi)$, a β pdf is presumed for the statistics of the passive scalar f, with a log normal distribution serving for the pdf of the scalar dissipation rate χ [9]. The exponents a and b are computed from the mean and variance of f, and $\widetilde{\chi}$ is the local mean scalar dissipation rate. Extinction is a sharply demarcated event, acting like a released guillotine when the limiting strain rate is exceeded. The mixture at a spatial location may therefore be represented as a combination of its chemically reacted and inert mixed flow states. In consequence, the pdf of *r* is a double delta pdf at the *r* bounds in *r*-*f* space, so that the kinetic computations may be performed independent of *r,* as given by Eq. 4.

The flamelet functions $Y_R(f)$ are obtained from apriori computed lookup tables, constructed using the one-dimensional counter flow detailed chemistry computational method of [10], conveniently implemented in Eq. 4 as Chebyshev polynomial approximations. In the present computations, the reduced chemical reaction mechanism for methane, involving 72 reactions and 28 species, proposed by [11] has been used for validations I and III, while validation II has been performed for a butane/propane mixture assuming the 200 reactions, 32 species scheme of [12]. The quenching scalar dissipation rates for the above two fuels at combustor operating conditions are computed using [10].

The global reaction scheme for nitric oxide as studied by De Soete [13] is considered in the computational procedure to predict nitrogen oxides emissions.

Assessment of results and conclusions

Validation I

The test facility in Wunning's experiment shown in Fig. 2.1 has been simulated. The predicted and measured radial temperature distribution at four axial locations (z=50,200,400 and 950mm) are displayed for the global fast chemistry model and the flamelet model. The global fast chemistry combustion model predicts a maximum temperature of about 2300K occurring around a thin region of the stoichiometric mixture fraction surface. The flamelet model predicts a lifted flame zone that appears detached from the burner consequent of the high jet velocities at the burner. The characteristic of a smooth temperature profile is reasonably represented by the flamelet model. This is a fair representation of Wunning's [1] observation following his experimental study and simulations, in which he showed that preheated temperature and axial velocity of the air were important parameters to achieve flameless oxidation combustion in the test combustion chamber.

Validation II

This validation study entails the prediction of an experiment, .2.2, expressly built to study flameless oxidation NOx reduction in gas turbines [2]. Combustion products are generated in a premixed pre-combustor and then diluted with air. The vitiated air so produced supplies a second combustor, termed the ‘afterburner’, which has been simulated. Data were collected at the indicated Test Station. There is high shear between the fuel and the vitiated air streams at the edge of the baffle. As a result, the present combustion model provides better predictions than the fast global chemistry. The global NOx model predictions (Fig. 2.3) at test station is in agreement with the NOx concentration measurements.

Validation III

The final validation study simulated the experimental HITAC furnace of the International Flame Research Foundation [3]. Vitiated air from a pre-combustor is supplied to the main combustor where measurements are taken. Both the fast global chemistry and flamelet models give good temperature predictions (see Fig. 2.4). Local quenching due to supercritical strain does not occur because here the use of vitiated air obviates the need for high momentum entry. The global NOx model predictions in Fig. 2.5 gives a good representation of the measured NOx concentration field.

Conclusions

Three model validation studies are summarised. The present flamelet model successfully simulates the validation data. It is found that the extinction criterion is necessary for good simulation of the ‘classical’ FO condition where the fuel and oxidant combustor entry streams have high momenta. Where a pre-combustor is deployed, allowing FO to be achieved at lower injection momenta, the simple global reaction is sufficient provided that the mesh is suitably refined. The global NOx model predicts the measured NOx concentration field.

Acknowledgements

The WP2 express their gratitude to the EU for its financial support (contract ENK5-CT-2000-00114) and the multi-national contract partners for their technical contribution to the project.

References

1. Wunning J. G. *Flammlose Oxidation von Brennstoff*. PhD thesis, RWTH., 1996

2. Levy Y, Nadvany V. and Nekrusov A. *Experimental unit for investigation flameless oxidation No$_x$ reduction in gas turbines*. Report No. JTL001-02-2003, Technion Israel Institute of Technology, Haifa 32000, Israel

3. Weber R., Verlaan A. Orsino S., and Lallemant N. On emerging furnace design methodology that provides substantial energy savings and drastic reductions in co2, co and NOx emissions. *Journal of the Institute of Energy*., 72 (1999) 77-83.

4. Kandamby N. H. Lockwood F. C. and M. A. Yehia. FAFNIR - A Computer Code for 3-D Turbulent Reacting Flows. Mechanical Engineering Department, Imperial College, University of London, 1990.

5. Jones, W.P. and Launder B. E. The prediction of laminarization with a two-equation model of turbulence, *Int. J. Heat Mass Transfer* 15 (1972) 301-314

6. Arscott J. A. Gibb, J. and Jenner R. European Symposium. The combustion Institute, Sheffield, 1973, p.674

7. Lockwood F. C. and Naguib A. S. A united method for the prediction of turbulent diffusion and premixed flames. Combustion Institute, 2nd European Symposium, 2 (1975) 502.

8. Lindstedt R. P. and Vaos E. M. Modelling of premixed turbulent flames with second moment methods. *Combustion and Flame* 116 (1999) 461-485.

9. Peters N. Laminar Diffusion Flamelet Models in non-premixed turbulent combustion. *Prog. Energy Combust. Sci.* 10 (1984) 319-339

10. Seiser R. Pitsch H. and Varatharajan B. A guide to FlameMaster. 2001

11. Warnatz J. *Skeleton CH4 mechanism* lecture notes in physics. In N. Peters and B. Rogg, editors, Reduced Kinetic Mechanisms for Applications in Combustion Systems. Springer, 1993

12. Marinov, N. M., Pitz, W. J., Westbrook, C. K., Vincitore, A. M., Castaldi, M. J. and Senkan, S. M. Aromatic Hydrocarbon formation in a laminar premixed n-butane flame, Lawrence Livermore National Laboratory. *Combustion and Flame* .114 (1998). 192-213

13. De Soete, G. G., An overall mechanism of nitric oxide formation from ammonia and amines added to premixed hydrocarbon flames, European Symposium on combustion, The Combustion Institute, Sheffield, (1973) p.73

14. De Soete, G. G., Overall reaction rates of NO and N2 formation form fuel nitrogen, Fifteenth Symposium International on Combustion, The Combustion Institute, (1975) p1093-1102

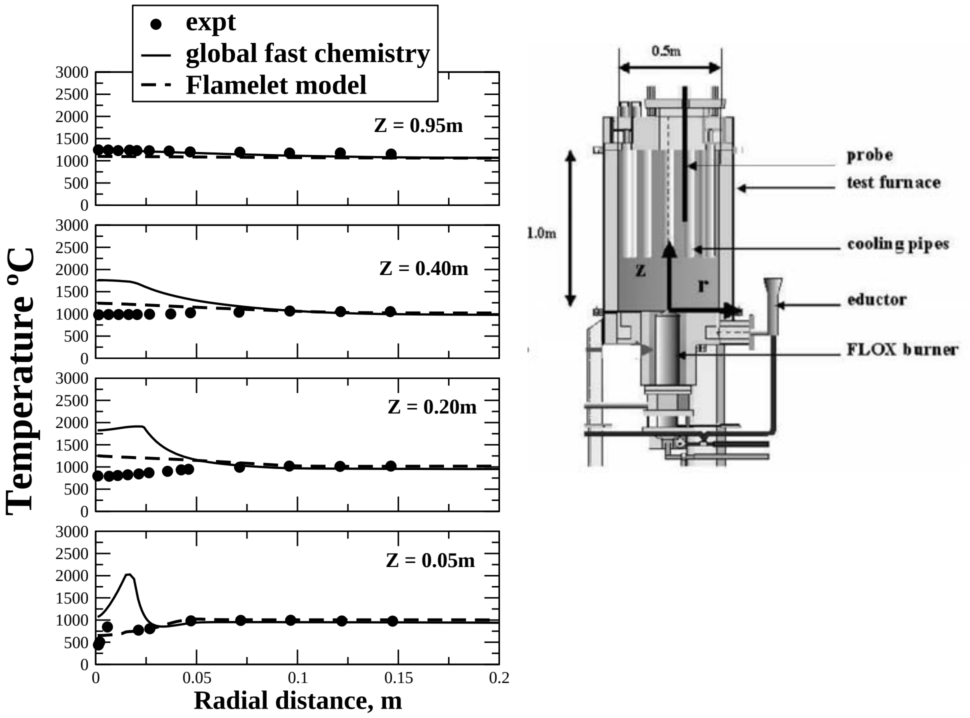


Fig 2.1 Flameless Oxidation test furnace by [1].

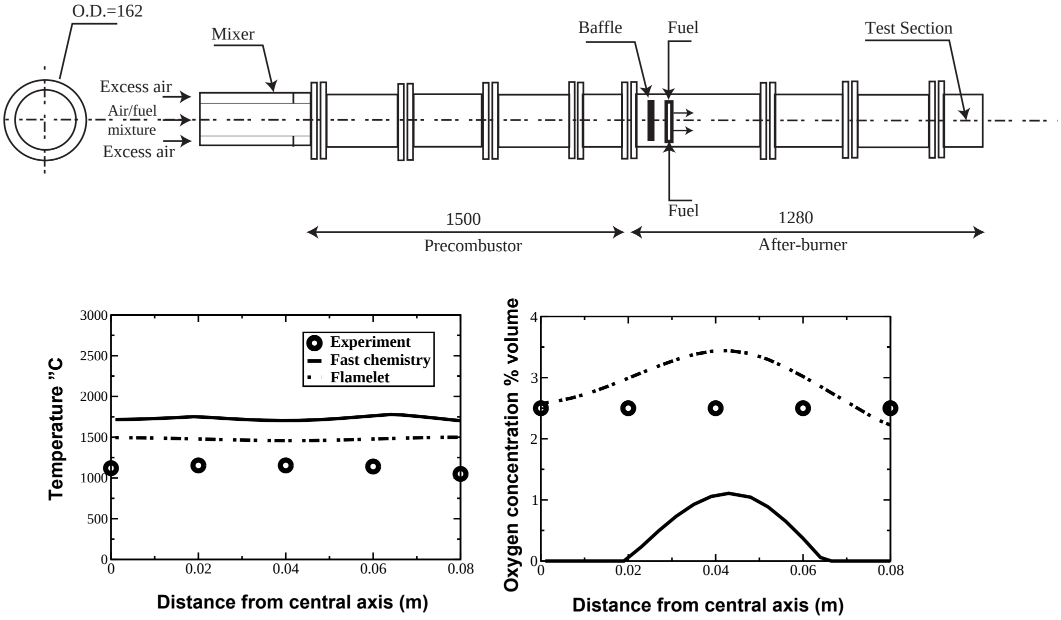


Fig 2.2 Flameless oxidation experimental device by [2].

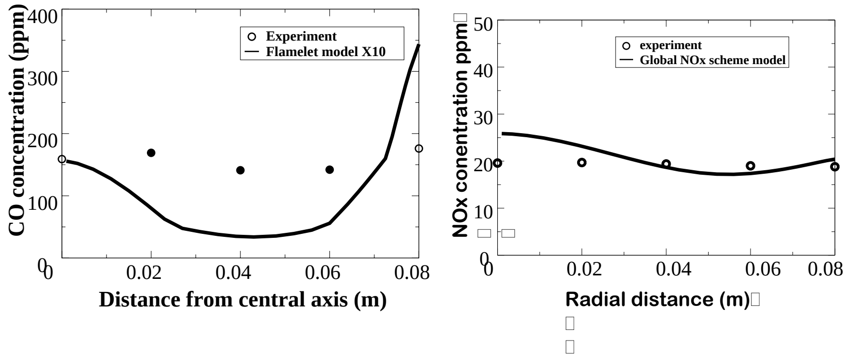


Fig 2.3 Predicted and measured carbon monoxide (CO) and NOx concentration field.

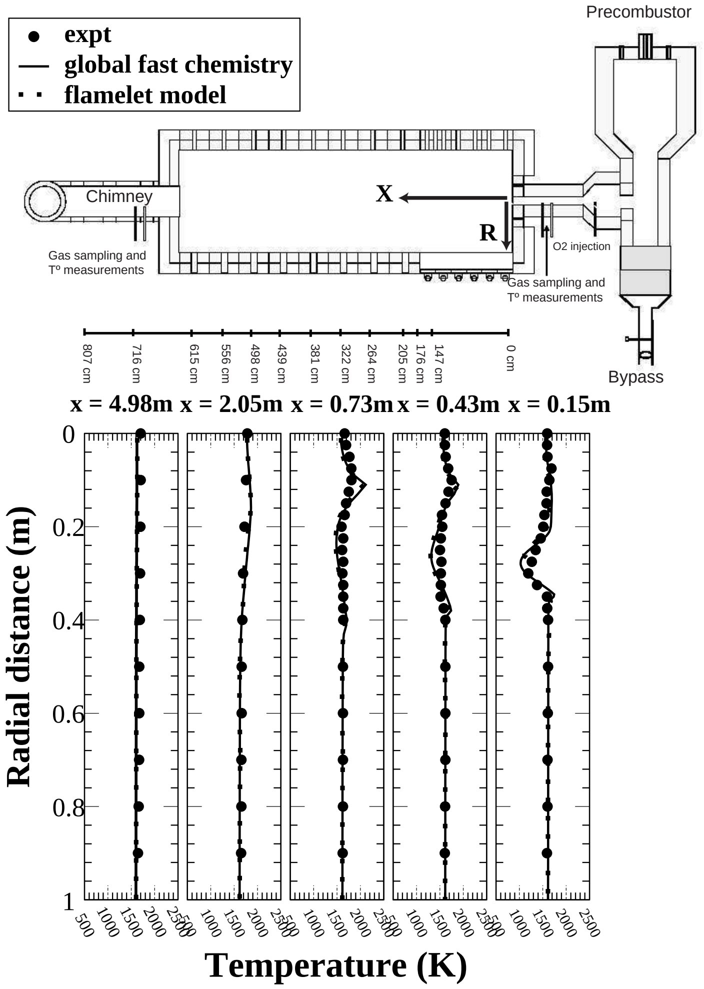


Fig 2.4 High temperature air combustion experiment by [3].

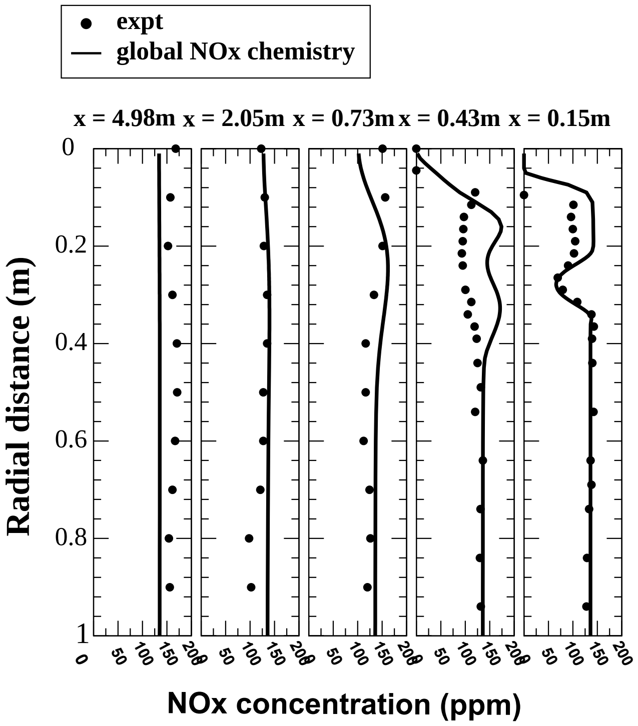


Fig 2.5 Predicted and measured NOx concentration field.

2.2.2.3 Experimental Research at the Technion (task 2.3, 2.4)

Flame dilution techniques were investigated in order to decrease NOx emissions. Dilution means that fuels and oxidizers are mixed locally with a mixture of inert gases before they react so that the oxygen concentration in the reactants is substantially reduced with respect to the 21% value of standard air. Recirculation of flue gases, or the products emanating from inside the combustion chamber, is the dilution mechanism for the present combustor . An increasing amount of recirculation of hot flue gases at 1200ºC to vitiate the combustion air has been assumed for achieving the Flameless Oxidation conditions. The present combustor was designed and developed to meet the performance required for investigation the proposed experimental unit for NOx reduction.

The experimental facility is organized as a series of components: premixer no. 1; two hollow cylindrical sections with total length L = 760 mm; measurement unit no. 1; hollow cylindrical section with total length L = 500 mm; bluff-body stabilized combustor; hollow cylindrical section with total length L = 500 mm and measurement unit no.2.

Pressures were measured by U-tube nanometers. Temperatures were measured by Chromel - Alumel thermocouples of the unshielded wedge type. Airflow rates were measured by orifice plates installed in accordance with ASME specifications. All the air and fuel flow meters were calibrated with the help of a Pitot tube.

2.2.2.4 Experimental Unit for Preliminary NOx Reduction Investigation

In order to adjust the measurement system with the present low-pressure source (air and fuel),the following a combustor with the following gaseous mixture was designed:

fuel $[C_4H_{10} + C_3H_8]$ commercial cooking gas, a mixture with varying proportions

$P_{\text{fuel inlet}}$	0.8 atm. gage
air flow rate:	50 ÷ 200 gram/sec
air access factor	$1 \div 3 = \dfrac{\text{air}}{15 \times \text{fuel}}$
$P_{\text{air inlet}}$	0.08 atm. gage

In order to control the mixing quality a long premixer was created. This premixer consists of three ducts with separate air and fuel supply. The center body of premixer is the pilot. The gas supply – a straight through nozzles with orifice diameter □0.6. The inner duct is equipped with swirler ($\alpha = 45°$) and is the main fuel-air duct. Fuel is supplied via four pipes with six holes each □ 1 mm for the fuel injection. The angle of the swirler blades can be changed in the range of $30° \div 60°$. The swirler outlet channel can be changed with the help of rings .

The outer duct is equipped with a swirler of $\alpha = 45°$ is the reserve fuel-air duct for operation with the large flow rates of air and for selecting a large excess air regime.

Fuel supply – eight pipes with six holes each one □ 1 mm for the fuel injection. The angle of the swirlers' blades can be changed inside within the range of $30° \div 60°$. The swirlers' outlet slot can be changed with the help of rings (1).

The inlet of the outer duct can be plugged with the help of rings with calibrated holes (2). The air duct to the pilot can be plugged with a plate (3) with a calibrated hole. The outer swirler has 11 blades and the inner swirler – 7 blades. The premixer flame tube is situated downstream. The following section is the measurement unit – a short piece of pipe with two openings for water-cooled probes. It is necessary to point out that currently, the way to cool combustion products after chamber no. 1 is by radiation heat flux from the liner walls. A gas-water heat exchanger was also assembled. In that heat exchanger, hot gases pass through 28 pipes of □ 15 mm inner diameter. The heat exchanger and premixer no. 2 are situated downstream. The design of this premixer, in general, imitates the design of premixer no. 1. The difference of premixer no. 2 is that its inner components are made of stainless steel. A bluff-body stabilized combustor (of the afterburner type) was also built. Fuel manifold consist of four pipes with diameter of 8mm with 24 holes of 1 mm in diameter each. The following premixer flame tube and measurement unit (measurement unit # 2) are the same as in the case of chamber # 1.

Fig 2.6 Modular Experimental Unit

It is necessary to point out that there was no control over the fuel composition and density. This is due to the fact the fuel supply was though domestic Butane bottles that are commercially available. The conclusion about their composition was reached by comparing the flow meters indication with the gas analysis results.

Static pressure at the combustor liner wall was measured by wall-static pressure taps. Liner wall temperature was measured by an IR camera.

Flue gas composition was measured by an emissions monitoring system Model 7000 – Measurement Technology Probe Controller Model 3000 E NOx – analyzer, model 400 CLD California Analytical Instruments CO/CO2 – analyzer, infrared analyzer model 100.

HC – analyzer, model 300 HFID (CE version) California Analytical Instrument SO2/O2 – analyzer, model 2010 NDIR, serial # 686, Liston Scientific.

Results and Discussion

It was clearly seen that the experimental unit allows one to reduce the combustion products temperature to the self-ignition range. It is seen that with the help of the additional hollow

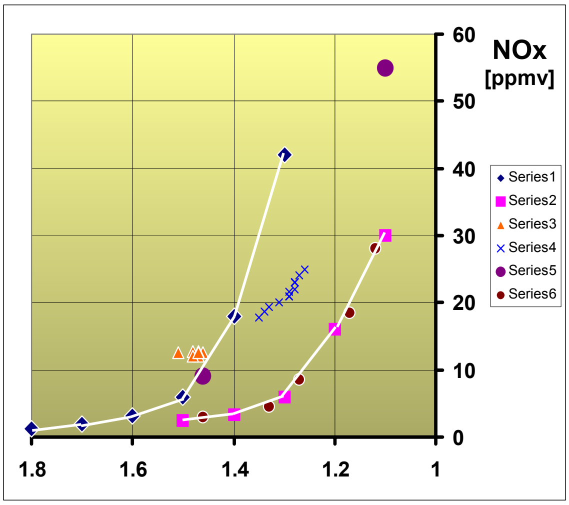


Fig 2.7 Gas Temperature Field Radial Distribution (section

cylindrical sections (before the second combustor), one can control the combustion products' temperature by the excess air.

Figure 2.7 shows the Gas Temperature Field. It is clear that one can obtain reliable data with the help of combustion no. 1. Correlation of NOx = NOx(λ), where λ=air/15 fuel comparison with the Lefebvre dependence was selected as a reference because of his emphasis on the influence of the residence time. This strong dependence for the emission of soot and NOx had been converted from theorem into a well-accepted theory long ago regarding the development of aircraft combustors [1,2]. Lefebvre's curve had been calculated for the conditions of present experiments: wcold ≅ 4 m/sec, where this velocity is calculated for the cold air movement inside the pipe of 160 mm internal diameter. Most of experimental data were obtained for the mixed domestic (C2H10 + C3H8) fuel. Short check-up test were also performed using clean (and expensive) Methane CH4 fuel.

2.2.3 WP 3: INTEGRATION OF DEVELOPED MATHEMATICAL MODELS &COMBUSTOR DESIGN

WP Leader: CINAR

2.2.3.1 Objectives and strategic aspects

The main objective for the Work Package 3 (WP3) in the FLOXCOM project is to integrate the developed mathematical models into a three-dimensional, multi-fuel flow and combustion computational procedure. The main objective is broken down in to five sub-objectives below:

1) To incorporate the newly developed combustion models in an existing three-dimensional, multi-fuel flow and combustion computational procedure.

2) To validate the resulting prediction procedure against well established literature on turbulent combustion data as well as against results from the experiments.

3) To perform a parametric study of the combustor's performance, in order to optimise its configuration, geometry and the inlet conditions.

4) To deduce from this parametric study generally applicable recommendations for the realization of a new gas turbine combustor with improved flame stability, combustion and emission performance.

2.2.3.2 Scientific and technical description

Work description and methodology

In this WP, the existing predictive quality of the computational fluid dynamics procedure is improved through incorporation of the developed mathematical models into a multi-fuel combustion solver. Validation against well-established combustion and NOx data from the literature as well as against results from experiments has been carried out. Optimisation of the operational conditions of the proposed gas turbine combustor design was performed to improve the mixing and combustion performance, and the pattern factor. The work package is divided into the following sub-tasks:

Task 3.1 Incorporation of the new flame extinction, atomisation and evaporation models into a computational unstructured grid and multi-fuel computational tool.

Task 3.2 Validation of the constructed code (Task 3.1) against literature as well as new experimental data.

Task 3.3 Parametric study of the gas turbine combustor operating under FLOX conditions covering a wide range of fuel and input variables.

Task 3.4. Deduction from a series of parametric studies of the process physics of a gas turbine combustor with particular regard to the good mixing, efficient combustion, and reduced NOx, CO and hydrocarbon emissions.

Methodology

Computational fluid dynamics (CFD) has evolved as an accepted tool in engineering design of fluid flow and combustion equipment, because of the enormous increase in the computational power. It is at present unavoidable to express complex physics, such as turbulence, turbulent combustion and radiation in flow/combustion dynamics, with approximate models. However, fundamental studies have shown that the available models are capable of well capturing the mean dynamics of a flow. The accuracy of a CFD procedure can be viewed as the precision with which the boundary conditions and geometry are specified/represented. Geometrical configurations of industrial flow or combustion equipment typically involve several topologically different boundaries (e.g. gas turbines). In order to improve the accuracy of a CFD procedure, the topologically unrestrained or free form meshes, usually termed as unstructured meshes, are often used. They extend the CFD methods to be applied for complex geometries with arbitrary topology.

The conservation equations

The fluid flow transport equations representing the conservation laws can, in general, be considered as the set of equations for the conservation of mass, momentum and for a general scalar variable. The generic form of the equations can be expressed as,

$$\frac{}{t}(\rho) + \nabla(\rho\underline{v}) = S_m$$

$$\frac{}{t}(\rho\underline{v}) + \nabla(\rho\underline{v}\otimes\underline{v} + p\underline{\underline{I}} - \underline{\underline{\tau}}) = \underline{S}_v$$

$$\frac{}{t}(\rho\phi) + \nabla(\rho\underline{v}\phi - \underline{q}) = S_\phi$$

where, for a Newtonian fluid with homogeneous and isotropic properties the stress tensor and the flux vector can be expressed as,

$$\underline{\underline{\tau}} = 2\mu\underline{\underline{S}} - \frac{2\mu}{3}\left(\nabla \cdot \underline{v}\right)\underline{\underline{I}} \quad \underline{\underline{S}} = \frac{1}{2}\left[\left(\nabla \otimes \underline{v}\right)^t + \left(\nabla \otimes \underline{v}\right)\right]$$

$$\underline{q} = \Gamma_\phi \nabla\phi \qquad \Gamma_\phi = \frac{\mu}{\sigma_\phi}$$

It can be shown that the occurrence of these curvature terms can be simply eliminated by expressing the vector and tensor components with respect to a spatially invariant base, thereby enabling a strong conservative formulation of the governing equations. Thus, using a Cartesian spatially invariant base to express the vector and tensor components a strong conservative form of the governing equations can be obtained by using tensor transformation laws in the contravariant or covariant differential forms, leading to,

$$\frac{}{t}\left(\rho\right) + \frac{1}{\sqrt{|G|}}\frac{}{\overline{x}^j}\left\{\sqrt{|G|}\,\rho J^j_m v^m\right\} = S_m$$

$$\frac{}{t}\left(\rho v^i\right) + \frac{1}{\sqrt{|G|}}\frac{}{\overline{x}^j}\left\{\sqrt{|G|}\left[\rho v^m v^i + p\delta^{mi} - \tau^{mi}\right]J^j_m\right\} = S^i_v$$

$$\frac{}{t}\left(\rho\phi\right) + \frac{1}{\sqrt{|G|}}\frac{}{\overline{x}^j}\left\{\sqrt{|G|}\left[\rho v^m\phi - q^m\right]J^j_m\right\} = S_\phi$$

where, the stress and the strain rate tensors, and the flux vector are given by,

$$\tau^{mi} = \mu\left\{J^\alpha_m \frac{v^i}{\overline{x}^\alpha} + J^\alpha_i \frac{v^m}{\overline{x}^\alpha}\right\} - \frac{2\mu}{3}\frac{1}{\sqrt{|G|}}\frac{}{\overline{x}^\alpha}\left\{\sqrt{|G|}\,J^\alpha_\beta v^\beta\right\}\delta^{mi}$$

$$S^{mi} = \frac{1}{2}\left\{J^\alpha_m \frac{v^i}{\overline{x}^\alpha} + J^\alpha_i \frac{v^m}{\overline{x}^\alpha}\right\}$$

$$q^m = \Gamma_\phi J^\alpha_m \frac{\phi}{\overline{x}^\alpha}$$

The CFD Solver

The Cartesian velocity based contravariant form of the steady incompressible state of the governing equations is employed in the block unstructured CFD solver. The conventional k-ε model [1] is deployed and, in standard form, the combustion model with presumed shape pdf is utilised. The thermal radiation transport is accommodated through a non-equilibrium diffusion approach [2], with absorption coefficients estimated from [3]. A non-staggered variable arrangement is deployed in the numerical procedure, and the pressure-velocity solution is obtained according to the SIMPLE algorithm [4], based on the Rhie and Chow technique [5]. Particular attention was paid to determine a reliable grid.

Mesh discretisation

In the unstructured domain flow solver, a coordinate frame oriented multi-block discretisation practice is deployed, with the mesh topology allowed to vary only at the boundaries between adjacent blocks. As such, the overall mesh topology, defined as the number of cells sharing a host cell, is not required to be uniform in the complete computational space. The method also allows non-hexahedral cells to be deployed, which are usually required when connecting two topologically different mesh regions, or for the purpose of inserting or removing cell layers in the overall mesh as required to tailor a mesh for a given geometry.

The structured blocks of the computational mesh can be constructed, in any of the available methods: either an elliptic/poisson differential equation based mesh generation technique or an algebraic method such as the transfinite interpolation technique. Considering computational efficiency, the later method is deployed in the present mesh construction procedures.

It must also be noted that the interior grid generation described above can also be analogously used to generate the mapping on boundary surfaces. This would typically require two dimensional mesh generation on a surface described in a three dimensional space. In this case a two dimensional surface mesh may be constructed between two pairs of boundary edges and the local curvature of the surface determined according the interior distribution of grid points on the surface. The overall mesh generation procedure can be summarised as follows: (a) selection of curves representing edges of given geometry, (b) define a mapping of grid points on the edges, (c) generate two-dimensional surface meshes of the physical and interface boundaries, (d) generate interior meshes between boundary surfaces, and (e) insert or remove cells or cell layers, or redefine the connections between cells in the adjoining regions as necessary. This procedure is used for generating the gas turbine combustor mesh into a block-structured mesh by use of transfinite interpolation technique, in which a two dimensional surface mesh is first constructed and the surface is rotated through an angle to form the three-dimensional FLOXCOM combustor sector shown in figure 3.1.

Assessment of results and conclusions

Original design (base case)

A number of runs were performed for the conditions inset in figure. 3.2. The base case represents the originally conceived design where the inlets were annular slots. The predicted inert flow pattern was validated by a flow visualisation experiment (figure 3.3). The combustor operating conditions are: methane fuel; operating pressure of 4.5 bars; and the fuel and air inlet flow rates and temperatures of 0.02kg/s at 293K and 2.0kg/s and 498K respectively. For this configuration, stable combustion was not achieved due to poor mixing such that combustion was incomplete, penetrating the exhaust section and resulting in a pattern factor in excess of 1.50.

Optimisation

The remaining runs were performed for a geometry where the annular inlet slots were replaced by evenly distributed holes, staggered with respect to the opposing jets. This arrangement resulted in significantly improved mixing and combustion characteristics.

The potential for the combustor design to be operated in the flameless oxidation mode was determined by plotting the combustion air temperature against the recirculation ratio, defined as the ratio of the total mass flow of recycled products to the sum of the inlet mass flows of air and fuel. The recirculation ratio was estimated through the calculation of a stream function characterising the mass flux within the recirculation zone. In general, combustion is considered to be stable under flameless oxidation conditions for recirculation ratios greater than 3.0 and reaction products above the auto-ignition temperature (circa 800ºC for methane) [6], [7], [8], [9]. The stability chart of figure 3.2 shows that several cases qualify. Since the combustor temperature always exceeds 800ºC, the former criterion controls. When inlet 2 is deployed, acting in opposition to the rotational sense of the large vortex, runs I, III and IV, the recirculation ratio is limited by the 'obstruction' so caused. Conversely, when inlet 1 is used, acting in harmony with the large vortex, runs II, V, VI and VII, the vortex is strong enough to ensure flameless oxidation. The results of run VII, the preferred case (figure 3.4), show that complete combustion occurs far upstream of the exhaust, with no trace of fuel in the exhaust.

Improved pattern factors and exhaust temperature profile shapes were achieved by introducing additional air holes: inlet 3 for upstream dilution; and inlet 5 to tailor the profile to achieve an optimum of maximum temperature at the blade mid-height and lowest temperatures at the blade root and tip. The pattern factors were lower than 0.25 for all cases except the base case. It was observed that the introduction of cooling air admitted through inlet 4 prevents the temperatures along the lower wall exceeding the metallurgical limit of the combustor.

Acknowledgements

The WP2 express their gratitude to the EU for its financial support (contract ENK5-CT-2000-00114) and the multi-national contract partners for their technical contribution to the project.

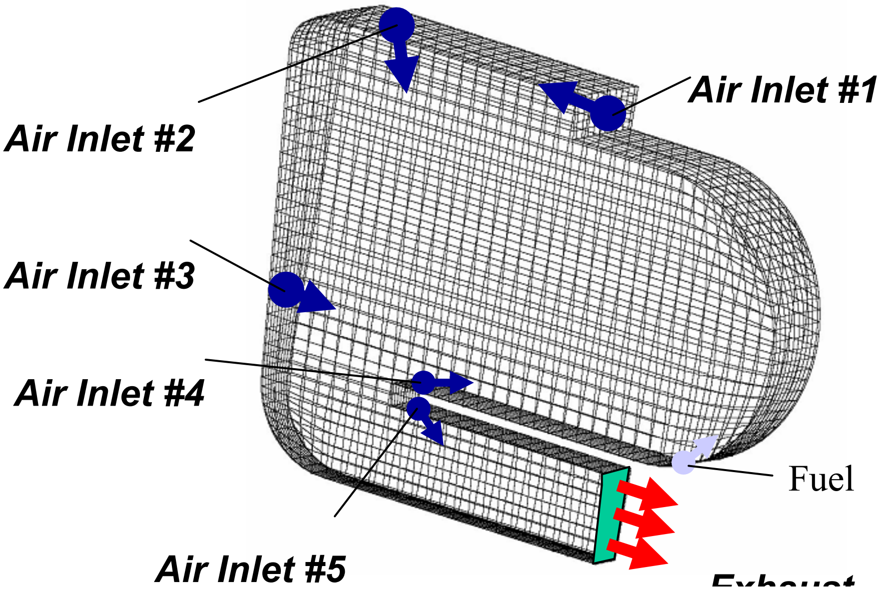
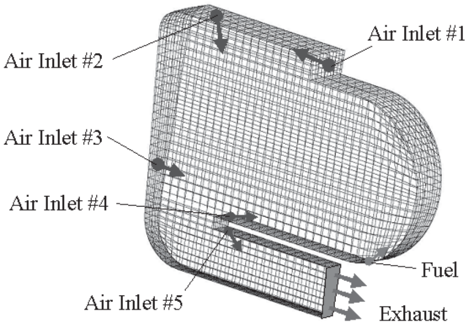


Fig 3.1 Discretised mesh of combustor



Simulation Code	Air Mass Flow Rate [kg/s] inlet #1	inlet #2	inlet #3	inlet #4	inlet #5
Base Case	1	1			
I	-	2	-	-	-
II	2	-	-	-	-
III	-	1.4	0.6	-	-
IV	-	1.4	-	0.4	0.2
V	1	-	0.5	0.5	-
VI	1	-	0.5	0.5	-
VII (Prefered Case)	0.8	0.8	-	0.3	0.1

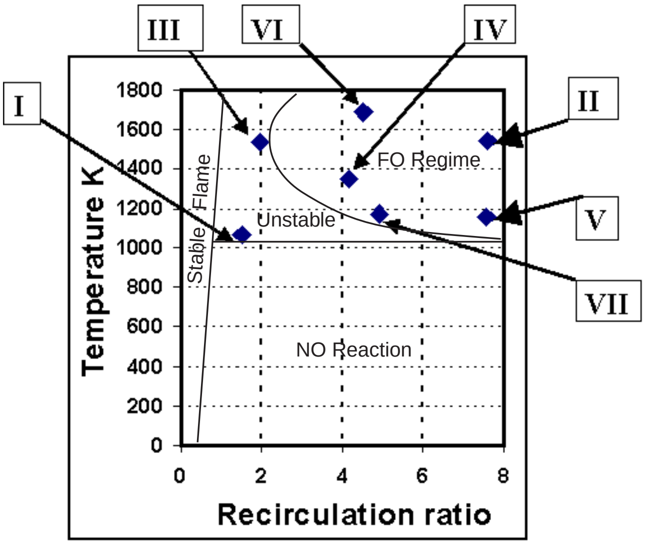
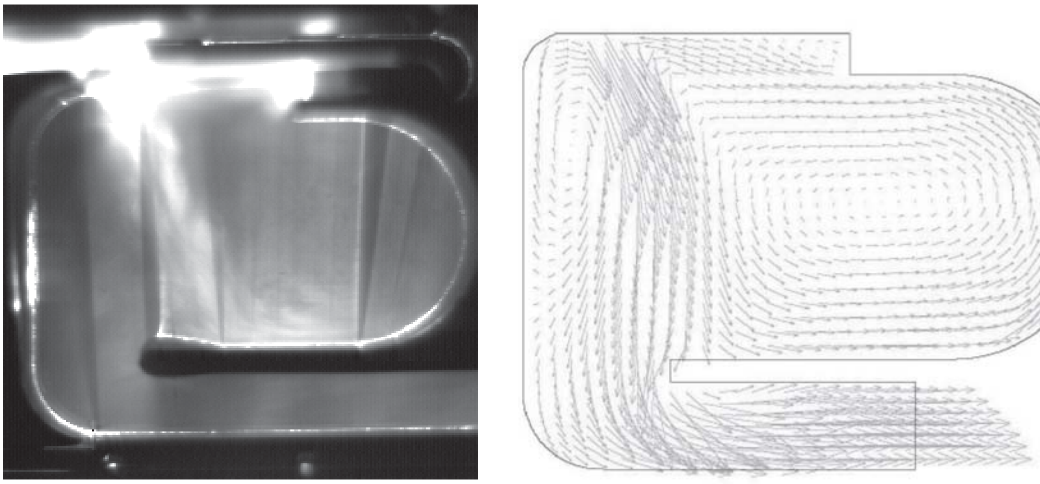


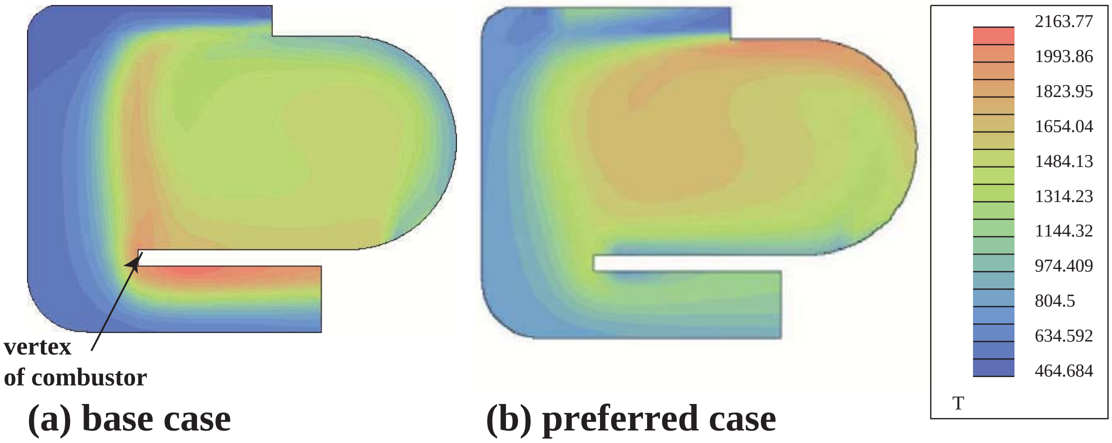
Fig 3.2 Discretised geometry, simulations and FO stability chart for proposed combustor.



▲ Cold flow visualisation experiment by Prof. T. A. Kowalewski (IPPT-PAN)

Predicted inert flow pattern ▲

Fig 3.3: Measured and predicted inert flow pattern.



(a) base case

(b) preferred case

Fig 3.4 Predicted temperature field of original geometry (base case) and modified design (preferred case) of proposed combustor.

References

[1] Jones, W.P. and Launder B. E. The prediction of laminarization with a two-equation model of turbulence, *Int. J. Heat Mass Transfer* 15 (1972) 301-314

[2] Arscott J. A. Gibb, J. and Jenner R. European Symposium. The combustion Institute, Sheffield, 1973, p.674

[3] Truelove J. S. A mixed grey gas model for flame radiation AERE HARWELL R-8494.

[4] Patankar S.V. Numerical Heat transfer and fluid flow. Hemisphere publishing corporation 1980

[5] Rhie C. M. and Chow W. L. Numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. AiAA-82-0998

[6] Wunning J. A. and Wunning J. G. Flameless oxidation to reduce thermal no-formation. *Progress in Energy Combustion Science*., 23, (1997) 81-94.

[7] Wunning J. G. *Flammlose Oxidation von Brennstoff*. PhD thesis, RWTH.
1996.

[8] Norbert P. Plessing T. and Wunning J. G. Laseroptical investigation of highly preheated combustion with strong exhaust gas recirculation. 27th Symposium on combustion, The Combustion Institute, Pittsburgh, PA., (1998). 3197-3204

[9] Wunning J. A. and Wunning J. G. *Ten years of flameless oxidation technical applications and potentials.* Proceedings of the 4th International symposium on High Temperature Air combustion and gasification 2001.

2.2.4 WP4: FUEL INJECTION SYSTEM DEVELOPMENT

WP Leader IST, Portugal

2.2.4.1 Objectives and Strategic Aspects

The present work is part of an European Commission project whose main objective was to develop a flameless oxidation combustor based on the principle of fuel injection in vitiated air, which is highly diluted in internally recirculated hot flue gases. Despite that most of the combustion equipment manufacturers already offer low-NO_x burners, there is still a great potential for pollutants reduction through design improvements. Recently, a special form of combustion called “flameless oxidation” has been presented as a means of reducing thermal NO formation (Wünning and Wünning, 1997). Apparently, the concept can be applied to a wide range of combustion equipment including small industrial gas turbines. Flameless oxidation conditions may have to incorporate a separate supply of fuel and air into the combustion chamber, so that the features of the fuel atomisation are of primary importance. The present report summarizes the work carried out at IST within the frame of this project, whose main objectives were: 1. design and construction of a wind tunnel and different sets of liquid atomisers and air injectors; 2 experimental investigations of the fuel supply system; and 3. testing of the combustor model.

2.2.4.2 Scientific and Technical Description of the Results

Design and construction of a wind tunnel and different liquid atomisers and air injectors

The FLOX combustor is based on the principle of fuel injection in vitiated air highly diluted in hot flue gases. Since the flue gases are recirculated internally, the combustor geometry and the flow field vortices are of prime importance. Preliminary numerical results have shown the critical importance of the fuel injection zone and, thus, this system must be optimized to enhance the jet momentum thereby allowing the recirculation pattern essential for FLOX mode. The former was the task being carried out at IST during the first year of the project.
With this objective in mind, IST has designed a wind tunnel capable of air velocities up to 40 m/s (to simulate the fuel injection zone in the combustor) and several nozzles that allowed detailed experimental studies of the fuel injection performance. Extensive tests were conducted with water, since its physical properties at 20 ºC are close to those of kerosene at 60 ºC. The water was injected into the wind tunnel through the nozzles, all them producing water jets. In order to achieve a good atomization and a contribution to the main vortex, several nozzles with different injection angles and different jet diameters have also been designed. The IST wind tunnel is shown in Figure 4.1.

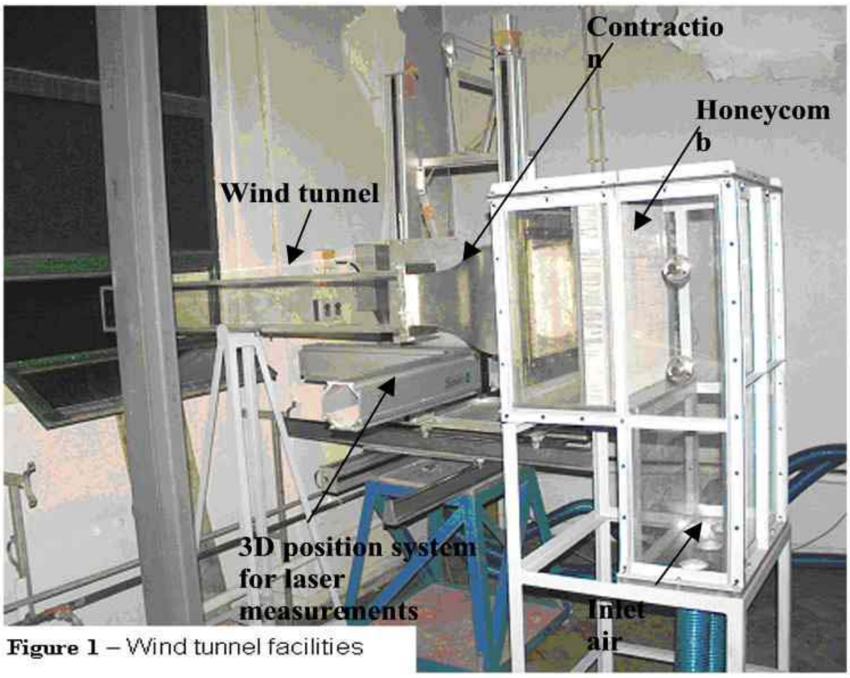


Fig 4.1 Wind Tunnel Facilities

Experimental investigation of the fuel supply system

The study reported in this section aimed to provide information on the influence of the main gas stream on fuel atomisation. To this end, the atomisation performance, under isothermal conditions, of a liquid jet emerging from a nozzle into the wind tunnel (crossflow) has been investigated. The atomisation data gathered during the project includes breakup length, droplet size and droplet velocities, and quantify not only the effect of the air velocity in the crossflow but also the effects of the nozzle injection angle and liquid (water) flow rate. Figures 4.2 and 4.3 show schematics of the test section and associated experimental techniques used in the present work. The experiments were conducted using water as the test liquid.

Figure 4.4 shows a series of exposures obtained during the visualisation procedure, which quantify the effect of the air velocity of the crossflow on the liquid jet disintegration process. It is seen that an increase in the air velocity of the crossflow results in a decrease in the stabilised spray angle with respect to the crossflow, accompanied by an increase in the width of the spray. Owing to the higher momentum of the air stream, the disintegration process promoted by the crossflow becomes more effective. This results in the formation of smaller droplets, which are more efficiently transported by the airflow, thus leading to a larger spread.

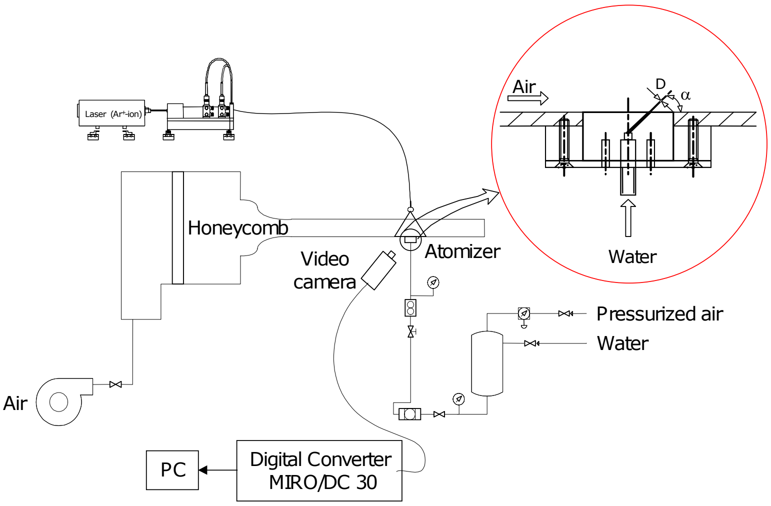


Fig 4.2 Flow visualisation arrangement and injection system.

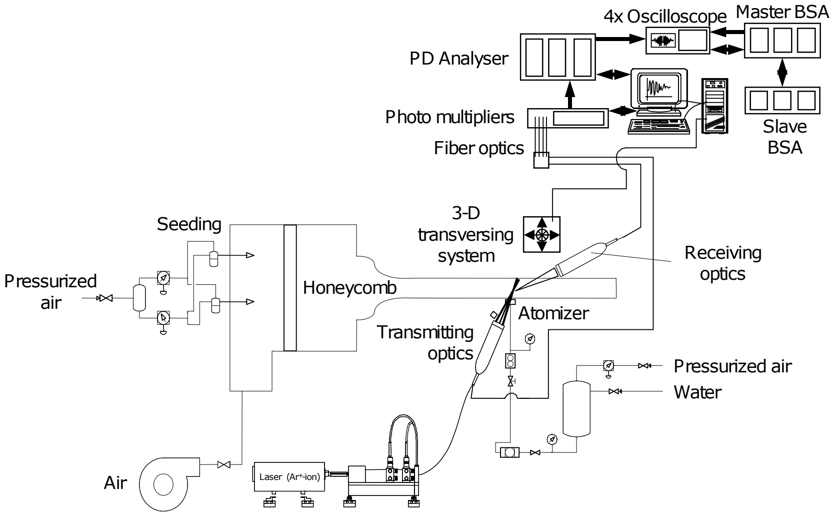


Fig 4.3 LDA and PDA measurement systems.

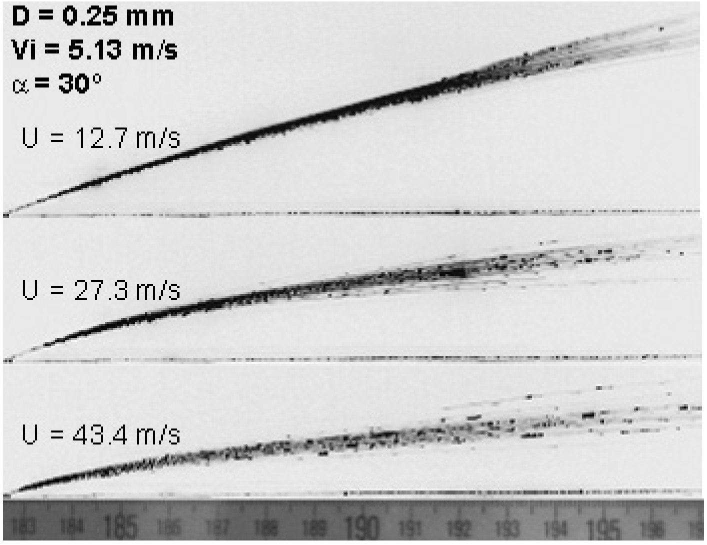


Fig 4.4 Flow visualization: effect of the air velocity of the crossflow in the sprays.

Figure 4.5 shows the droplet size and velocity distributions at *s*/*D* = 20 and 100 for a typical condition studied. There is an evident *U*-velocity/diameter correlation for all droplets, with small droplets being faster than larger droplets. In addition, the crossflow affects the droplet trajectories in such a way that the smaller droplets exhibit lower *V*-velocities as compared with the larger droplets. As for droplet diameters, Figure 4.5 shows a gradual increase in the Sauter Mean Diameter (SMD) in the *y*-direction, corresponding to a spectrum of droplet sizes produced in the flowing air stream that is skewed vertically. This clearly indicates that the larger droplets penetrate farther into the air stream. It can also be observed in Figure 4.5 that there is an increase on the values of SMD when moving downstream from *s*/*D* = 20 to 100. This can be attributed to the shorter lifetime of the smaller droplets due to evaporation and to the coalescence caused by droplet collisions. In the present flow configuration, droplet collisions are expected to be the most important phenomenon.

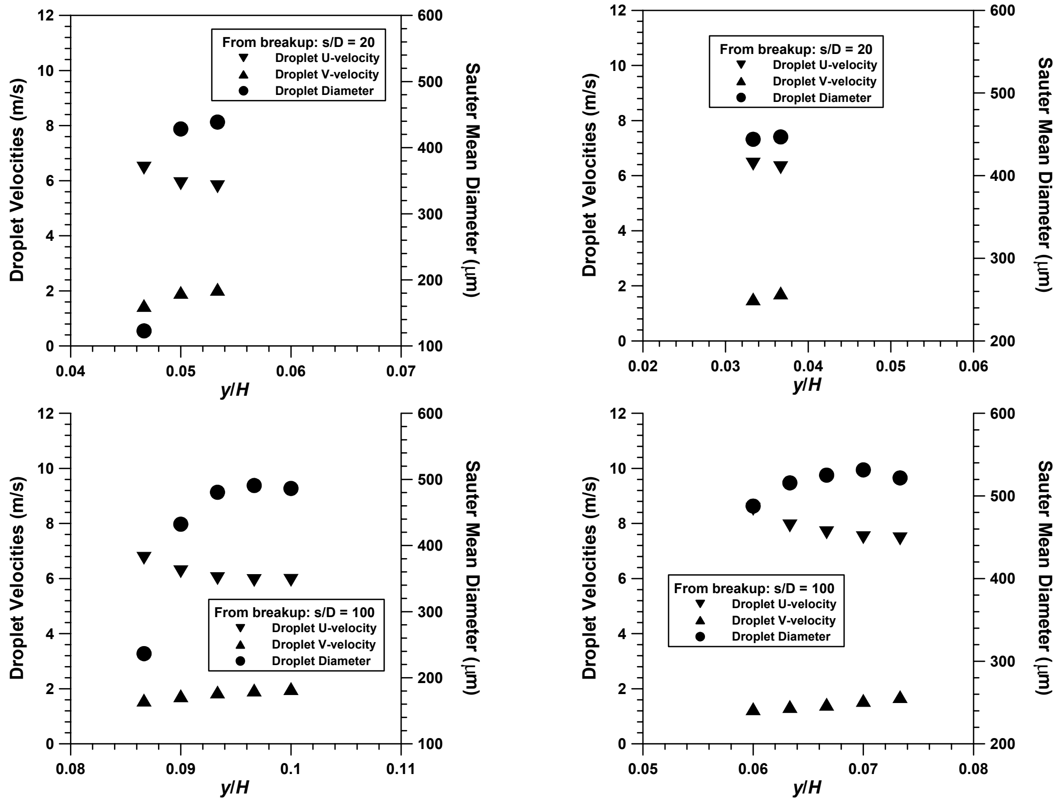


Figure 4.5. Effect of air velocity of the crossflow: U = 12.7 m/s (left) and U = 27.3 m/s (right).

Testing of the combustor model

The study reported in this section aimed to provide information on the performance of the combustor model both under non reacting and reacting conditions. The flow characteristics of the combustor model under non reacting conditions have been investigated using Laser-Doppler Anemometry. Data was gathered for mean and turbulent velocities as a function of the air mass flow rate and its preheating temperature. The isothermal flow characterization was followed by combustion measurements at the exit of the combustor model. Measurements of mean gas species concentration (O_2, CO_2, CO, HC and NO_x) were performed as a function of the equivalence ratio and thermal input for two different configurations of the main air inlets.

Figure 4.6 shows the effect of the air flow rate on the mean flow structure. The figure reveals that a common feature to all the conditions is the establishment of a large recirculation zone and that the mean velocities increase within the recirculation zone as the air mass flow rate increases.

Figure 4.7 shows the measured flue-gas data for the combustor model as a function of the equivalence ratio and of the thermal input. The figure reveals that combustion performance is higher for both lower values of thermal input and equivalence ratio. The NO_x emissions are very low regardless of the combustor operating conditions. This demonstrates the effectiveness of the combustor on the reduction of the NO_x emissions, which was one of the main objectives of the present project.

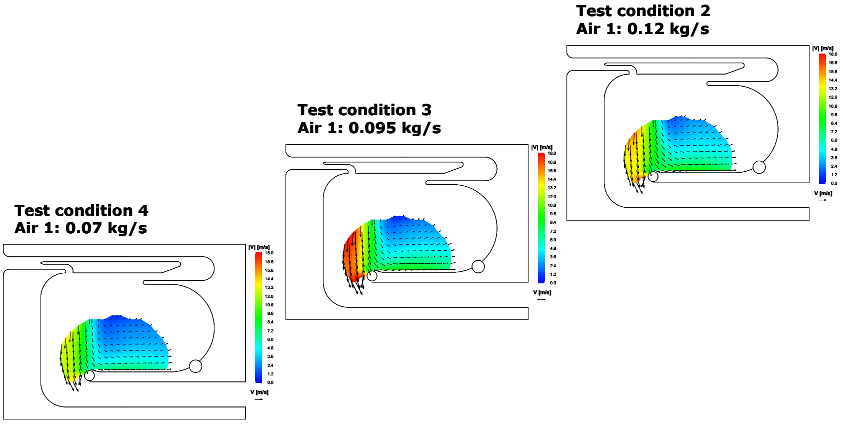


Figure 4.6. Effect of the Air 1 Flow Rate on the Mean Flow Structure

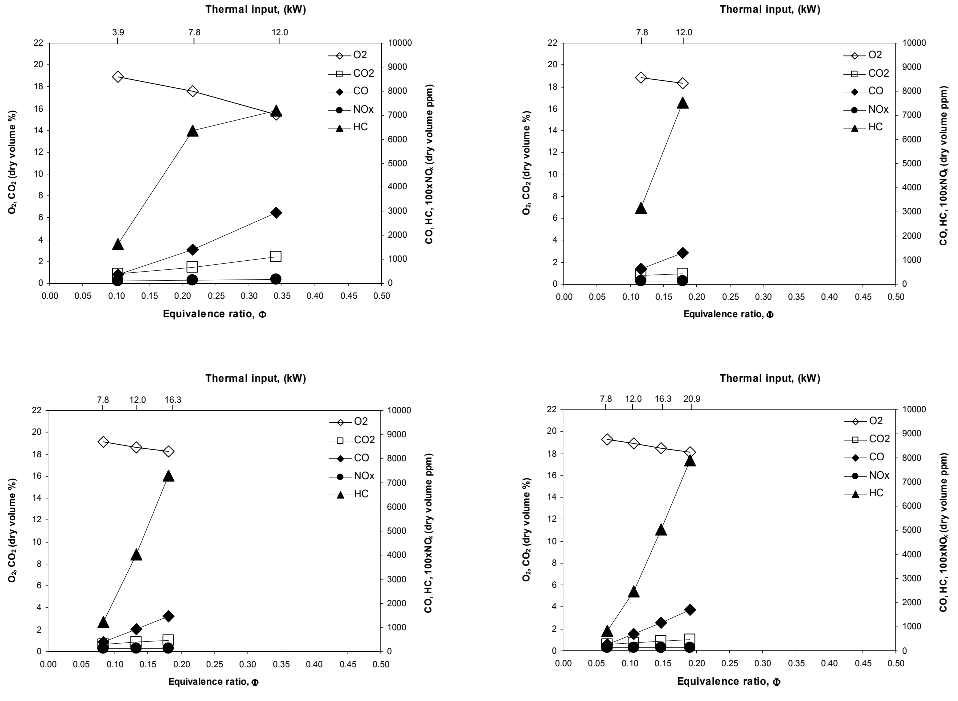


Fig 4.7 Flue-gas data for the combustor model

Experiments under reacting conditions

Table 2 summarises the operating conditions for the experiments under reacting conditions, which allow for an assessment of the effects of the equivalence ratio and thermal input on pollutant emissions for both configurations studied (see Figures 4.8b and 4.8c) Plate 2 shows a photograph of a typical flame and Figures 4.11 show the measured flue-gas data for the original model configuration, shown in Figure 4.8b, as a function of the equivalence ratio and of the thermal input. The figure reveals that combustion performance is higher for both lower values of thermal input and equivalence ratio. It is interesting to note that NO_x emissions are very low regardless of the combustor operating conditions.

In order to try to improve the combustion performance of the combustor, the configuration of the air inlets has been modified, as shown in Figure 4.8c). The measured flue-gas data for this new configuration is represented in Figure 4.12. As can be seen, the effect of the air inlet configuration on both combustor performance and pollutant emissions is marginal.

CONCLUDING REMARKS

The isothermal flow characterization was followed by combustion measurements at the exit of the combustor model. Measurements of mean gas species concentration (O_2, CO_2, CO, HC and NO_x) are reported as a function of the equivalence ratio and thermal input for two different configurations of the air inlets. The main conclusions are as follows: i) combustion performance is higher for both lower values of thermal input and equivalence ratio; ii) NO_x emissions are very low regardless of the combustor operating conditions and iii) the effect of the air inlet configuration on both combustor performance and pollutant emissions is marginal.

Table 2. Operating Conditions for the Experiments under Reacting Conditions for both Model Configurations

Test Condition	**Air 1**		**Fuel (Methane)**	**Equivalence ratio, Φ**
	Flow rate [kg/s]	**Temperature [ºC]**	**Flow rate [kg/s]**	
Original Model Configuration				
1	0.012	380	7.79E-05	0.10
2		387	1.56E-04	0.22
3		387	2.39E-04	0.34
4	0.023	361	1.56E-04	0.12
5		361	2.39E-04	0.18
6	0.031	369	1.56E-04	0.08
7		368	2.39E-04	0.013
8		368	3.27E-04	0.18
9	0.038	368	1.56E-04	0.07
10		368	2.39E-04	0.11
11		368	3.27E-04	0.15
12		369	4.18E-04	0.19
Modified Model configuration				
1	0.012	378	1.17E-04	0.17
2		378	1.56E-04	0.22
3		384	2.39E-04	0.34
4		391	3.27E-04	0.49
5	0.020	403	1.56E-04	0.13
6		402	2.39E-04	0.20
7		403	3.27E-04	0.28
8		405	4.18E-04	0.35
9	0.027	409	2.39E-04	0.15
10		407	3.27E-04	0.20
11		407	4.18E-04	0.27
12		409	5.12E-04	0.33
13	0.035	411	3.27E-04	0.16
14		411	4.18E-04	0.21
15		414	5.12E-04	0.25
16		417	6.22E-04	0.30

Plate 1. Photograph of the experimental set-up

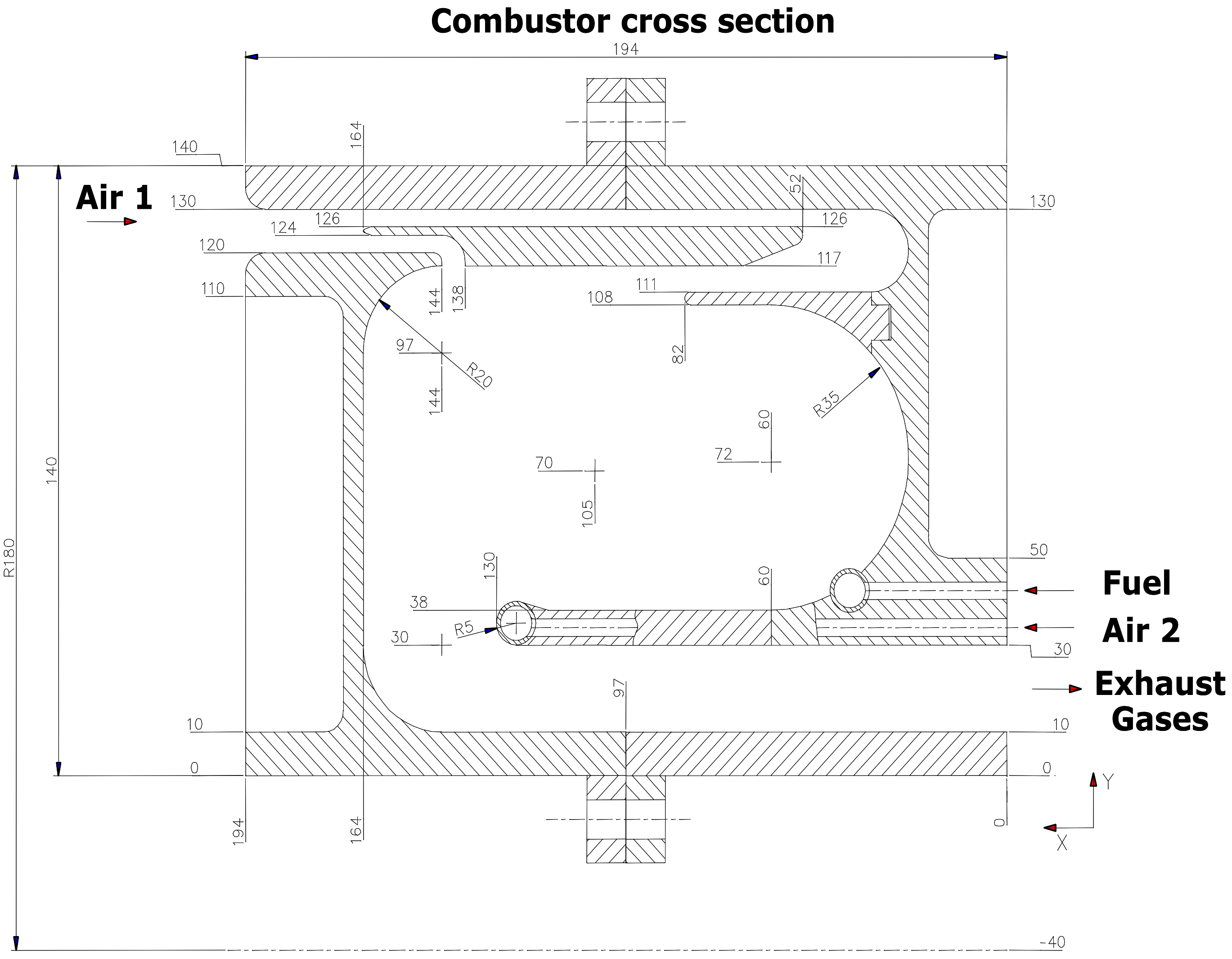


Fig 4.8a. Schematic of the Combustor Model

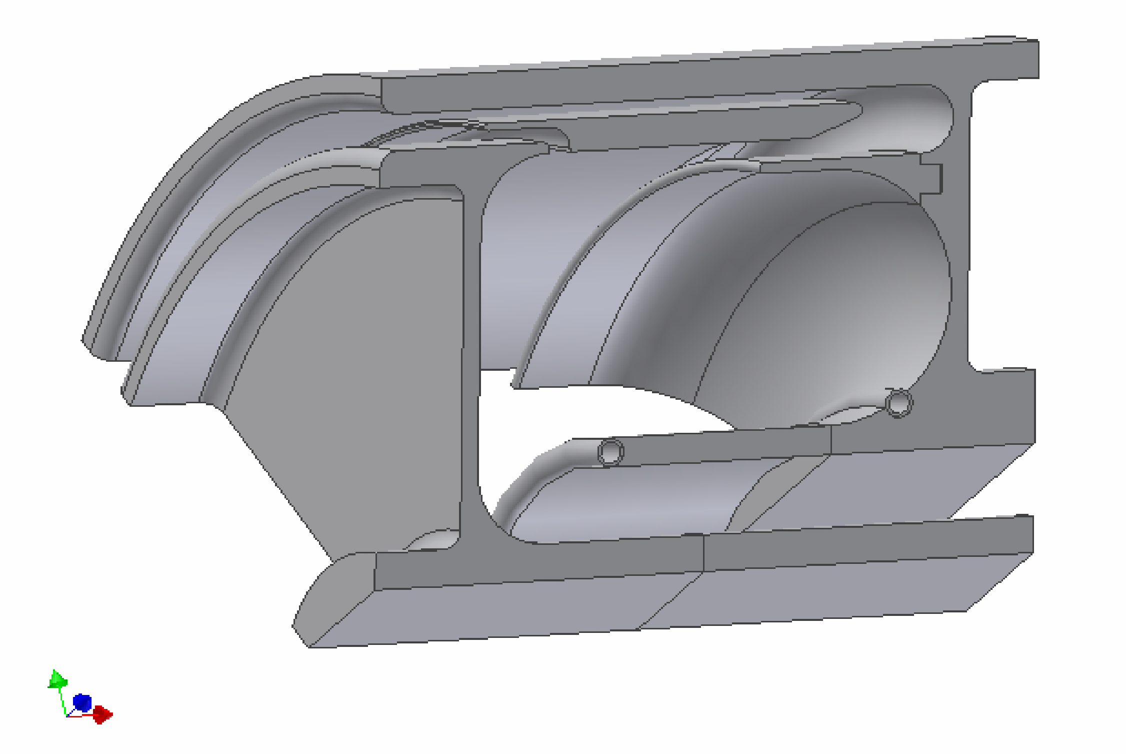


Fig 4.8b. Original Model Configuration

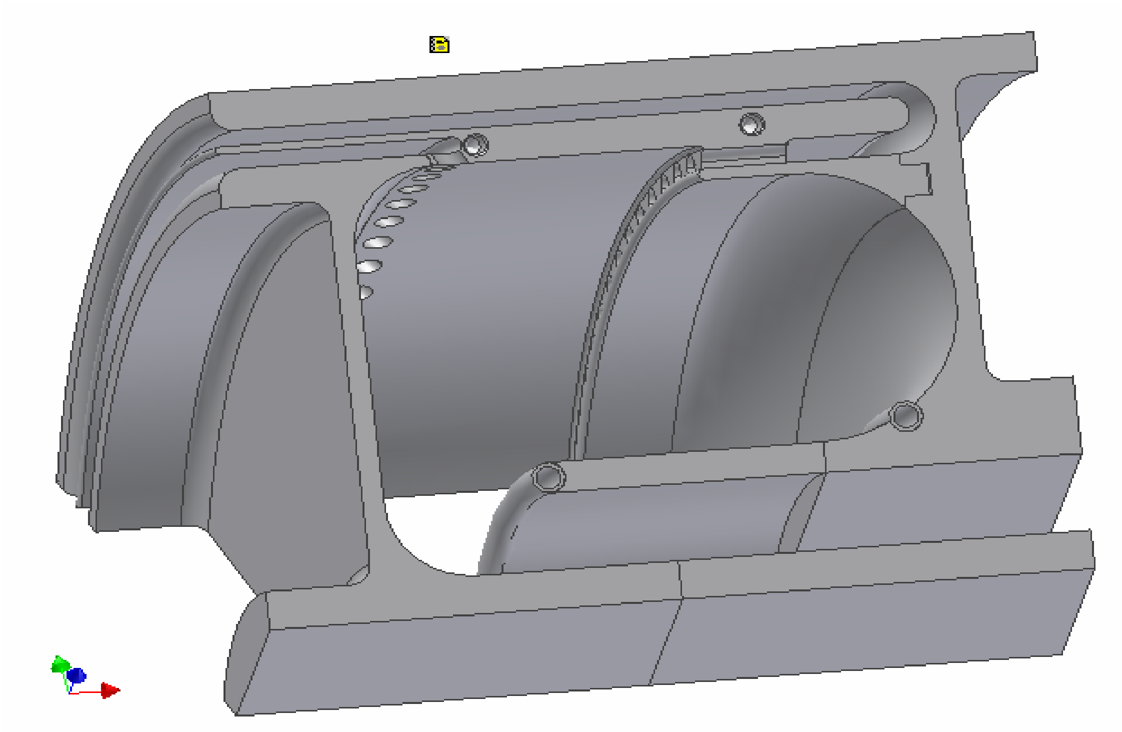


Fig 4.8c Modified Model Configuration

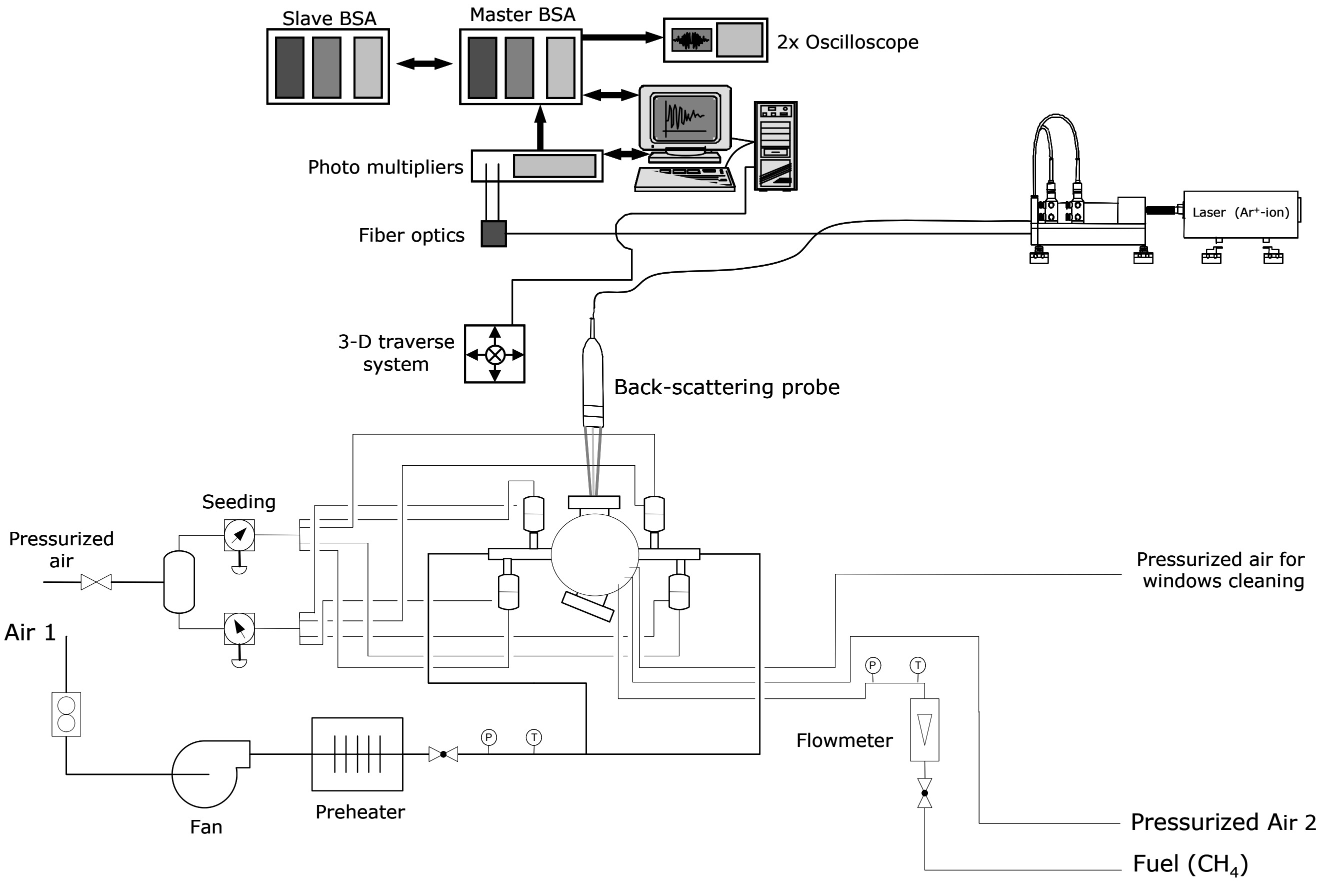


Fig 4.9 LDA Measurement System

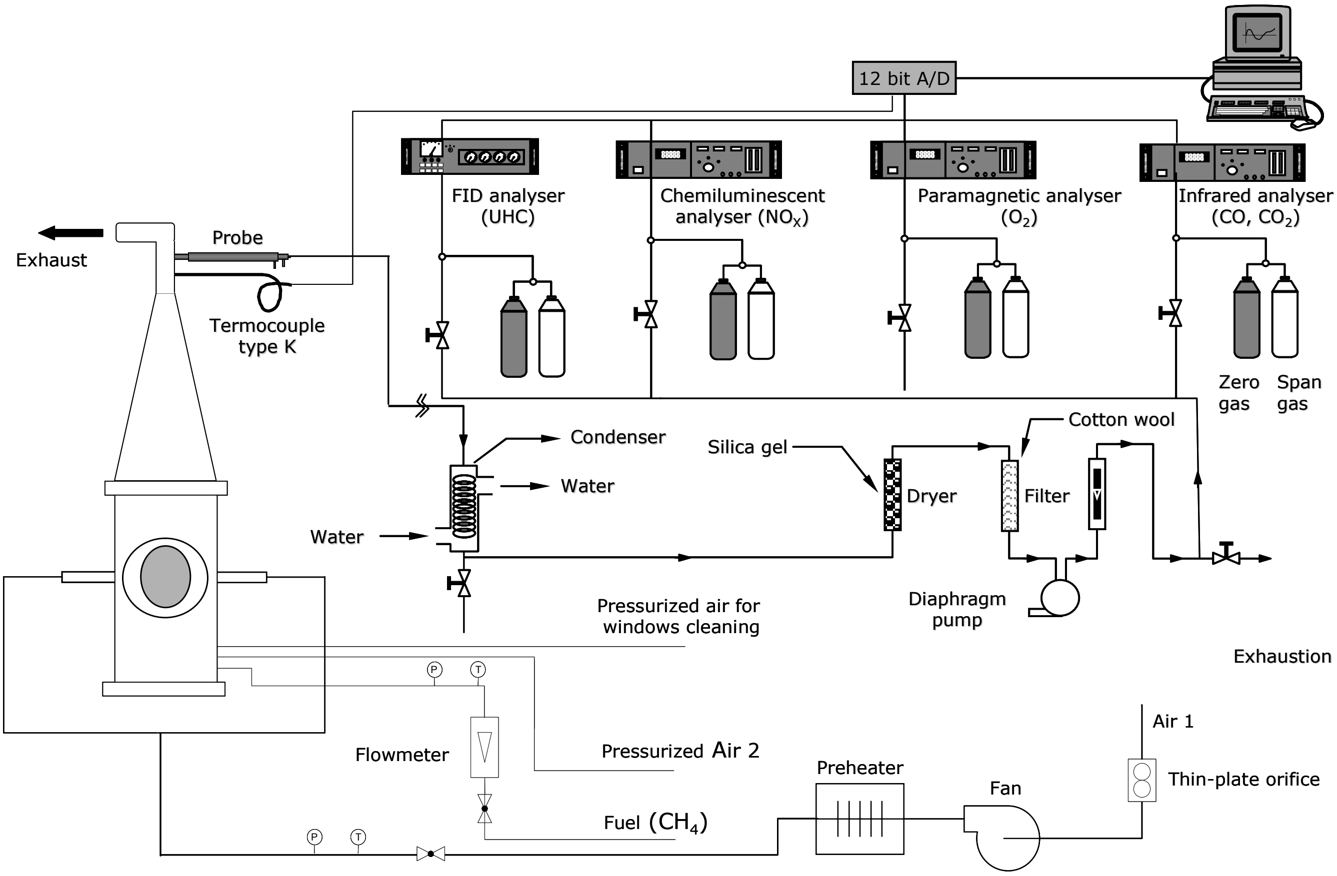


Fig 4.10 Combustion Measurement System

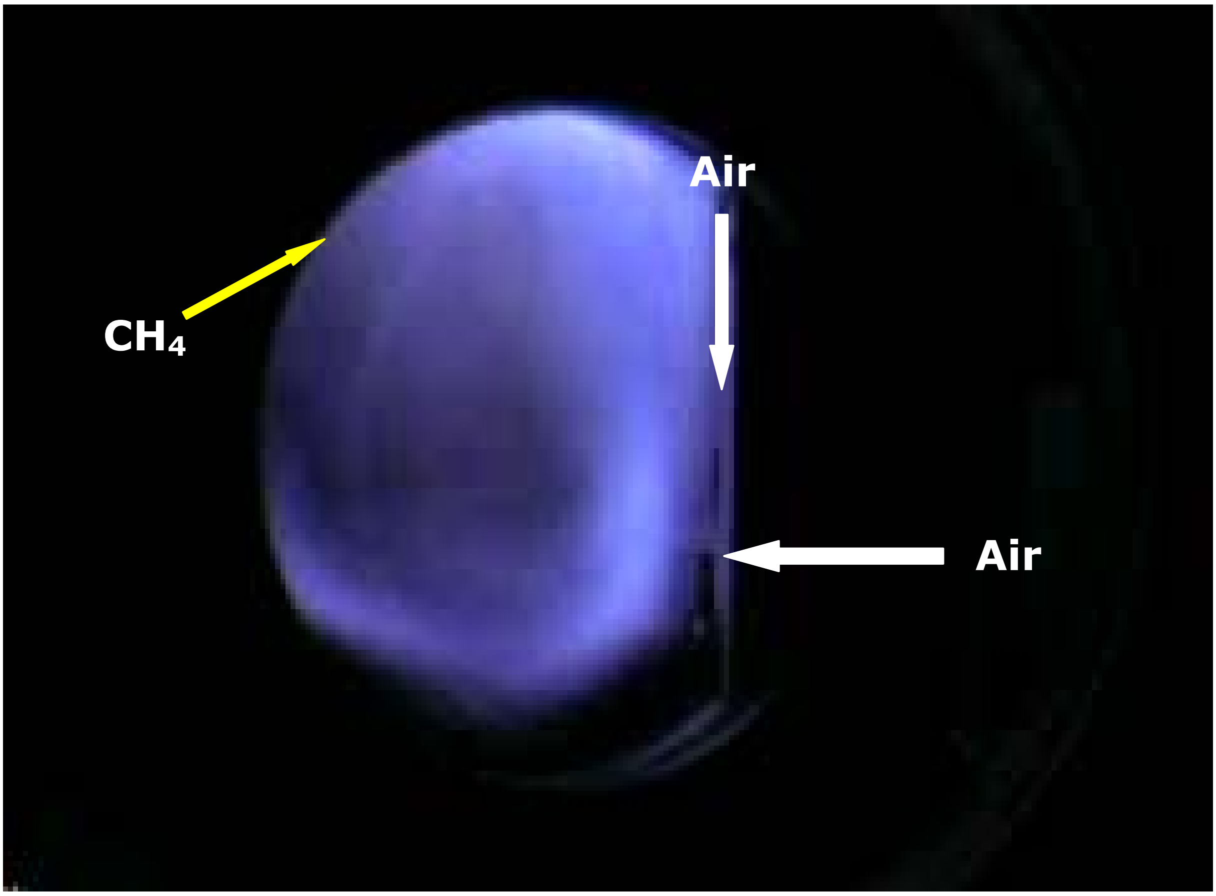


Plate 2 Photograph of a typical flame for the original model configuration shown in Figure 4.8b.

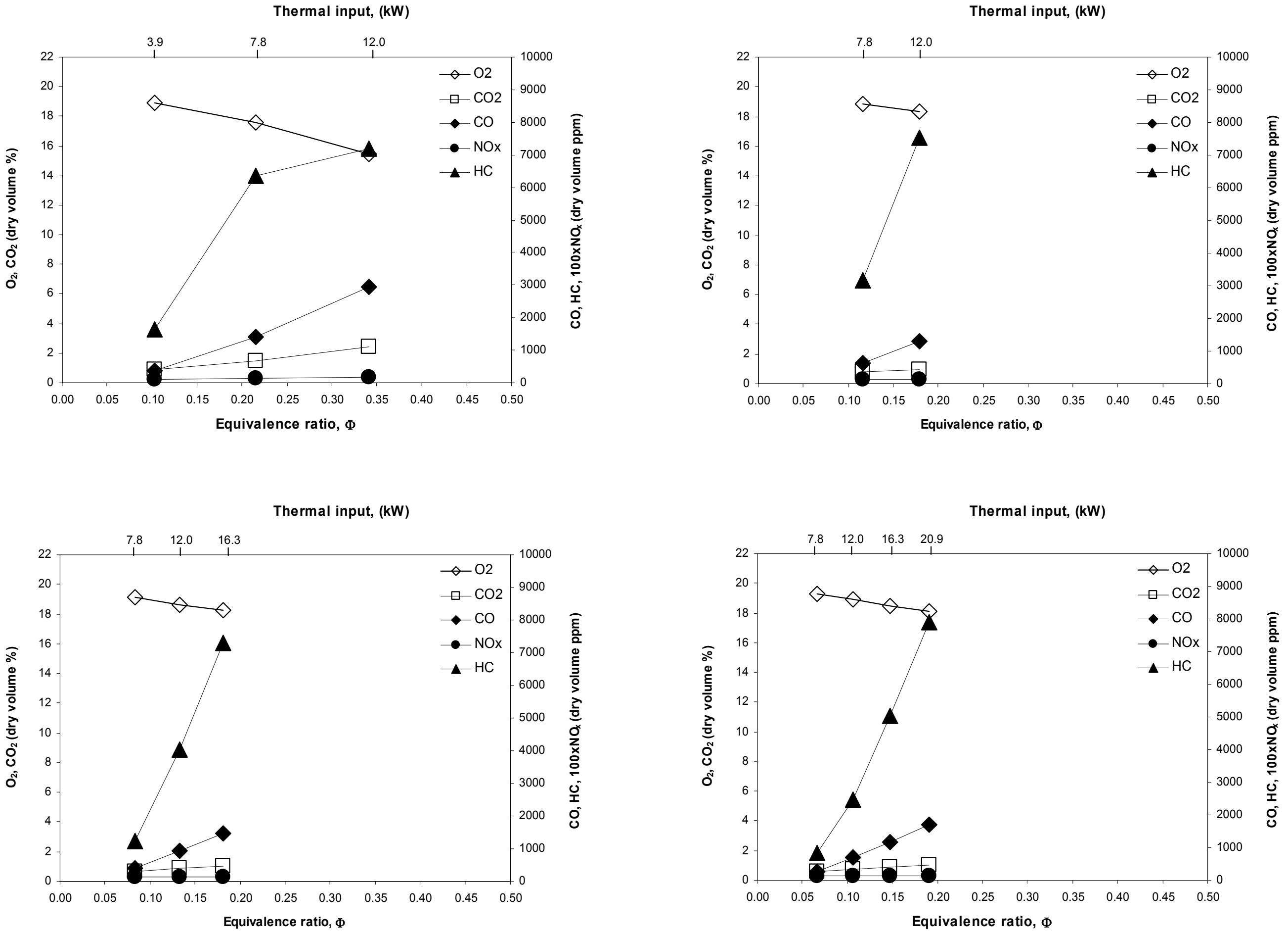


Fig 4.11- Flue-gas data for the original model configuration shown in Figure 4.8b.

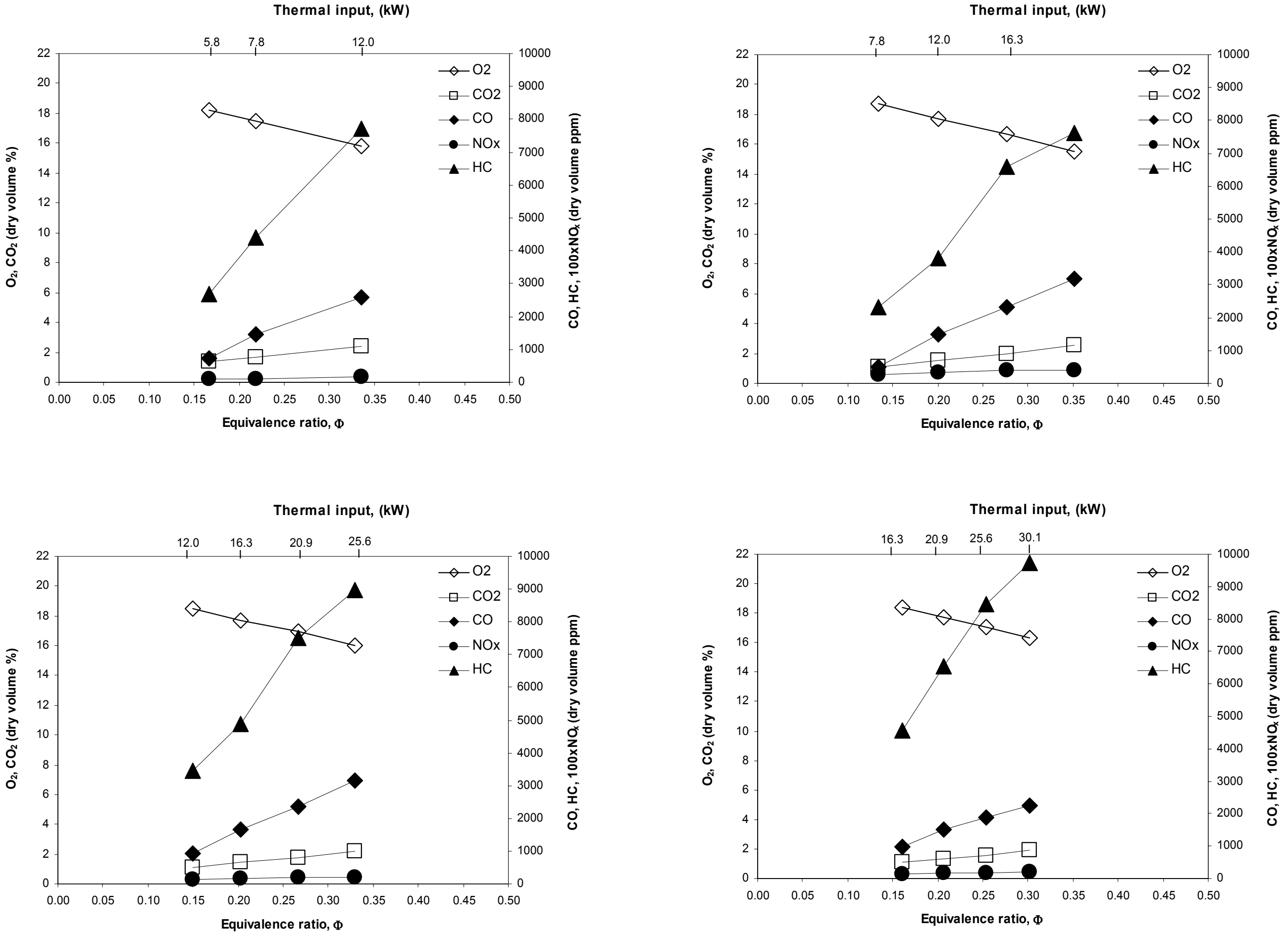


Fig 4.12 Flue-gas data for the modified model configuration shown in Figure 4.8c.

2.2.4.3 Assessment of Results and Conclusions

The isothermal spray characteristics of a liquid jet emerging from a nozzle into a wind tunnel (crossflow) has been investigated for a range of air velocities in the crossflow, nozzle injection angles and liquid (water) flow rates. The results include spray images, breakup length, droplet size and droplet velocities. The main conclusions of this work are as follows: i) droplet trajectories are significantly affected by the nozzle injection angle and, to a lesser extent, by both the air velocity in the crossflow and the liquid injection velocity. In particular, there is a strong relationship between the spray width and the nozzle injection angle; ii) jet breakup lengths decrease significantly with the air velocity in the crossflow over the entire range of investigated velocities and with the nozzle injection angle up to a value of 30º, beyond which only marginal variations are observed; iii) the droplet diameters increase gradually in the vertical direction indicating that the larger droplets penetrate farther into the air stream; iv) there is an evident *U*-velocity/diameter correlation for all droplets, with small droplets being faster than larger droplets and v) a common feature to all sprays is that the droplet *U*-velocities increase significantly on increasing the air velocity of the crossflow, exhibiting values above the injection liquid velocity but well below the air velocity.

The flow characteristics of a combustor model under non reacting conditions have been investigated using Laser-Doppler Anemometry. Data is reported for mean and turbulent velocities as a function of the air mass flow rate and its preheating temperature. The main conclusions are as follows: i) a common feature to all test conditions is the establishment of a large recirculation zone; ii) mean and turbulent velocities increase within the recirculation zone as the air mass flow rate increases; iii) near the combustor outlet the mean and turbulent velocities are higher for the intermediate air mass flow rate tested and iv) the effect of the air inlet preheated temperature on the flow field is marginal.

The isothermal flow characterization was followed by combustion measurements at the exit of the combustor model. Measurements of mean gas species concentration (O_2, CO_2, CO, HC and NO_x) are reported as a function of the equivalence ratio and thermal input for two different configurations of the air inlets. The main conclusions are as follows: i) combustion performance is higher for both lower values of thermal input and equivalence ratio; ii) NO_x emissions are very low regardless of the combustor operating conditions and iii) the effect of the air inlet configuration on both combustor performance and pollutant emissions is marginal.

References

1. Wünning, J. A. and Wünning, J. G. (1997). “Flameless oxidation to reduce thermal NO-formation”, Prog. Energy Combust. Sci., 23, pp. 81-94.

Levy, Y. (2002). Personal communication.

2. Sousa, J. M. M. and Pereira, J. C. F. (2000). “Rollup region of a turbulent trailing vortex issued from a blade with flow separation”, Exp. Thermal Fluid Sci., 20, pp. 150-161.

2.2.5 WP5: LASER DIAGNOSTICS OF VORTICAL FLOW

WP Leader IPPT-PAN

2.2.5.1 Objectives and strategic aspects

The experimental validation of the numerical model is a primary strategy necessary to obtain credible predictions for industrial problems. The selected strategy of using simplified, non-combusting flow offers unique possibility for detailed flow field analysis.

The main target of the WP 5 is to get physical insight of the main vortex, responsible for the efficient mixing of fuel and air in the combustor. This process is essential for optimisation of the combustor and for the global strategy of the project. Measurements performed for cold flow in a transparent model of a combustor delivered detailed description of the flow structure, impossible to obtain in a working combustor with high temperature, high pressure and inevitable soot deposition on the observation window. The data on the flow structure (visualisation movies) and instantaneous velocity fields were delivered to compare with the numerical simulations generated using Fluent at IPPT, and by partners (CINAR, Technion, RWTH, B&B AGEMA). The cold flow information obtained was used to verify numerical models used and to optimise combustor geometry.

Deliverable D5.1 Transparent combustor model. Modern optical methods of qualitative and quantitative flow field analysis were employed to achieve the WP goals. To secure good insight to the flow several transparent models of combustor were assembled from glass plates or machined form Plexiglas. Air under atmospheric conditions was used as a working fluid. The construction of the cavity allows for flexible changes of geometry by movable sliding plates. The geometry has two inlet channels and one outlet. Beside geometry changes, variation of the flow rate ratio through the inlets, as well as enhancement of the inlet turbulence were chosen as the main optimization parameters. The basic geometry was several times modified according to the preliminary flow measurements and suggestions of the coordinator. Figure 5.1 illustrates Geometry 2 with two parallel walls used for primary optimization tests. Such construction allowed easy optical access and detailed measurements of the flow in the sector model.

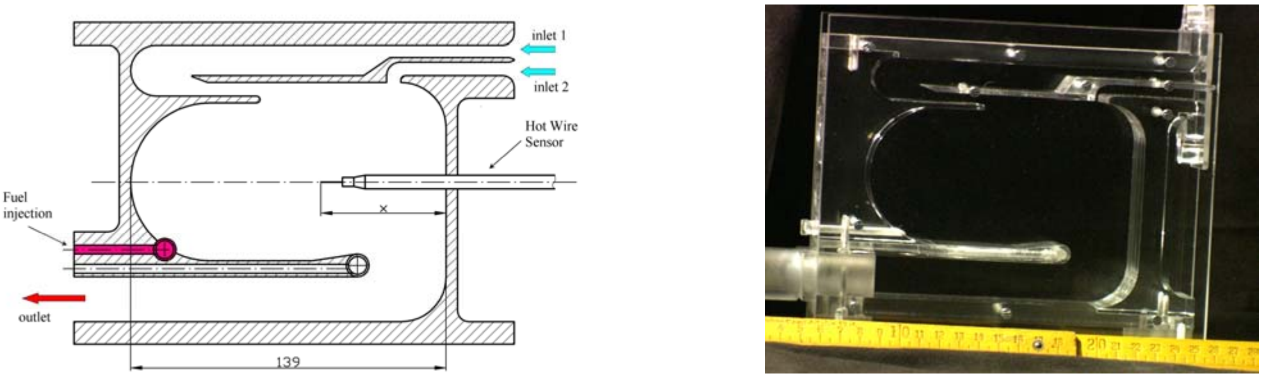


Fig 5.1 Schematic and Plexiglas model for Geometry 2 with two parallel walls, used for the cold flow visualization

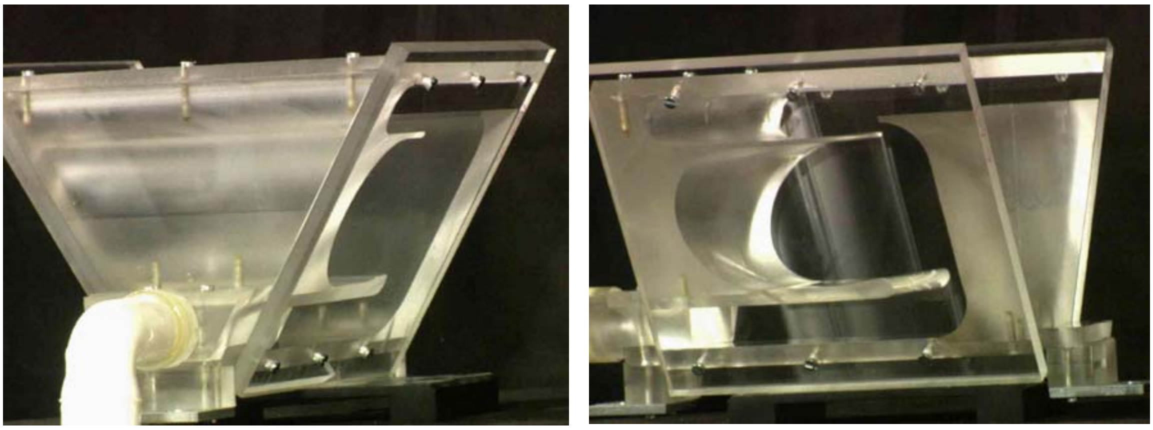


Fig 5.2 Side views of the Plexiglas model for Geometry 3 used for the cold flow visualization. It replicates main features of the final combustor model.

Figure 5.2 gives an overview of the final three-dimensional model (Geometry 3), built in accordance with knowledge gained with help of a simpler preliminary Geometry 1 and Geometry 2. The Geometry 3 consists of two inclined transparent sidewalls with inner details analogue to the real version of the sector investigated by the FLOXCOM consortium. Advantage of the Geometry 3 is its direct geometrical relation to the investigated combustor sector. However, wall inclination and complexity of the three-dimensional inner details (multiple curvatures of the channels and divisions) caused additional obstacles for the laser light and limited accuracy of the optical measurements. Therefore, for the flow optimisation both Geometry 2 and Geometry 3 cold flow data were used.

Flow visualisation using high speed imaging was used for detailed flow structure analyses and quantitative measurements of the flow velocity turbulent fluctuations (high speed Particle Image Velocimetry). High resolution Particle Image Velocimetry was applied for detailed evaluation of temporary flow velocity fields. Four channel one component and three components hot wire anemometer was applied to obtain high frequency measurements of turbulent fluctuations and to verify Particle Image Velocimetry measurements. For better understanding of flow mixing properties, the hot air jet of temperature about 300°C was introduced through the side wall simulating fuel injection. The flow in the transparent combustor was observed using Schlieren setup and Mach-Zender interferometer.

The image acquisition experimental setup consisted of FASTCAM – Ultima 40K high speed video system with maximum rate of 40000 frames/s and high resolution PIV system with 12bit SensiCam CCD camera. Illumination was based on light sheet technique using a CW 5W Ar laser for high speed imaging and a double pulse NdYag 35mW (minimum time interval 200ns) laser for the high resolution PIV measurements. The typical range of investigated flow velocities is 10-25m/s. The optical arrangement allows for flow visualization using fog generator and PIV measurements using tracer particles. The schematic of the optical setup shows Figure 5.3. It consists of several mirrors and the optical arrangement to form thin (about 2mm) light sheet penetrating the investigated cavity from a direction perpendicular to the observation direction.

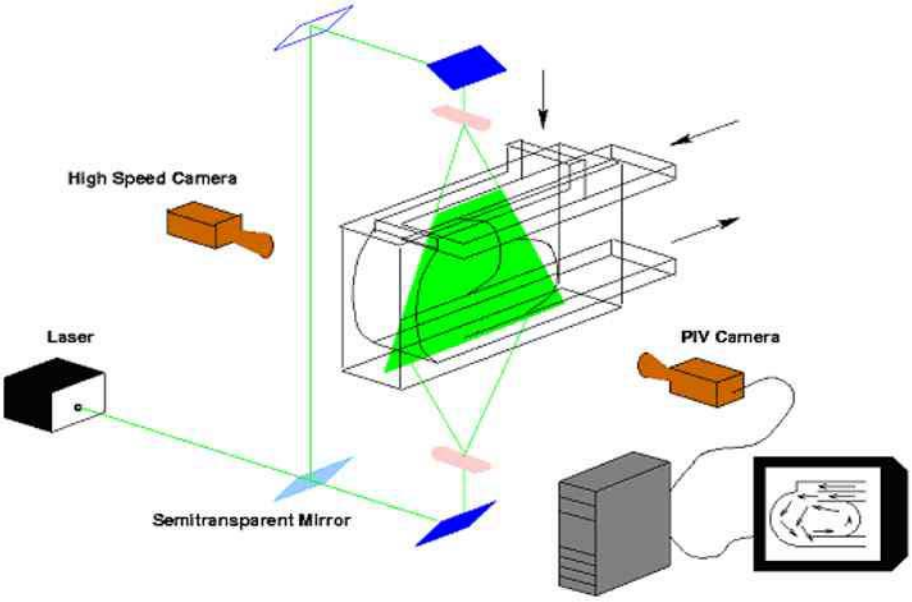
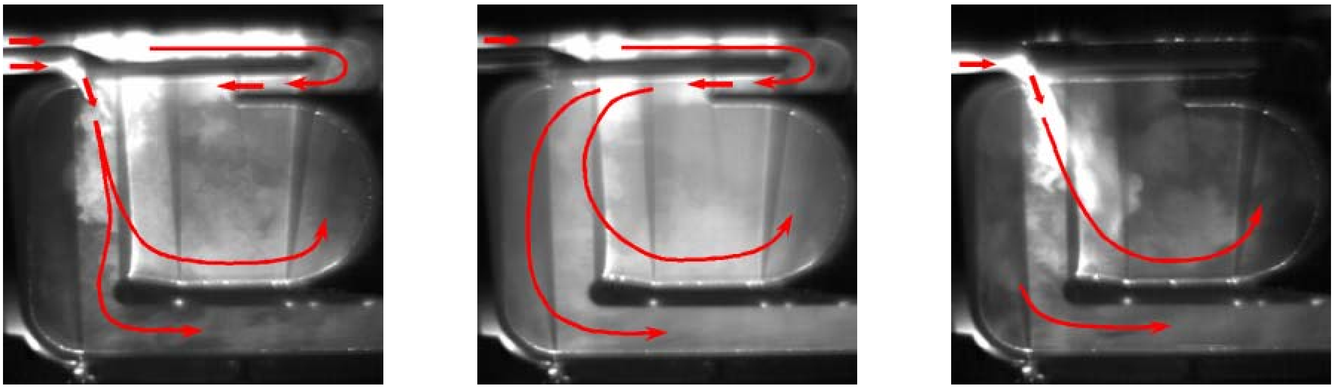


Fig 5.3 Schematic of the experimental setup.

About 40 outlet-inlet modifications of the inner geometry and flow have been investigated for three different base geometries in purpose to understand and to optimize the flow structure, so that the flow return to the region of combustion prevails. The flow return is the main target of the FLOXCOM project. The cold flow experimental results obtained were compared with numerical simulations performed in framework of WP5. The numerical runs were performed on unstructured grids (up to 1mln elements) with commercial code Fluent 6 using k-e turbulence model (steady state) and Large Eddy Simulation (LES) approach (unsteady flow). The flow velocity fields and turbulence statistics were compared with the experimental results.

2.2.5.2 Scientific and technical description of the results

Deliverable D5.2. The interaction of the fuel injection aerodynamics and the main vortex. The parametric experimental study of the cold flow combustor clearly indicated that the main goal of the project, namely flow return in the inner vortex can be achieved using properly chosen flow conditions at two inlets or using only lower inlet. Typical images of the flow structure for Geometry 2 are given in Figure 5.4. It appeared that the tangential flow generated by the upper inlet (configuration B) does not guarantee penetration of the flow into the main vortex (inner combustion region). The most efficient flow mixing was present for the configurations A (both inlets open) and configuration C (only lower inlet with direct flow into the lower part of the combustor open).



Configuration „A” Configuration „B” Configuration „C”

Fig 5.4 Flow structure observed in the cold flow model for three investigated inlet configurations: A – both inlets with inlet velocity u=10m/s, B – upper inlet open, tangential flow with inlet velocity u=20m/s, C – lower inlet open, inlet velocity u=17m/s.

High resolution Particle Image Velocimetry was used to obtain detailed 2D velocity fields for the central cross-section of the model (Fig. 5.5).

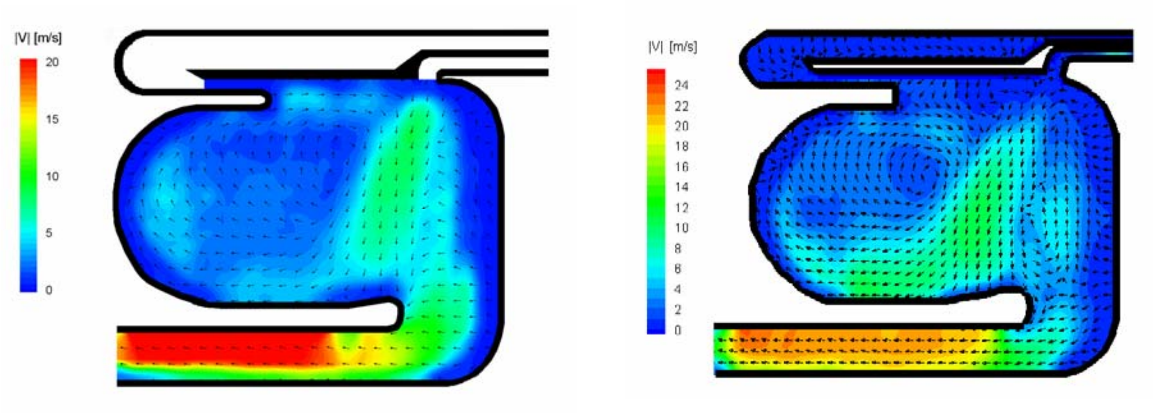
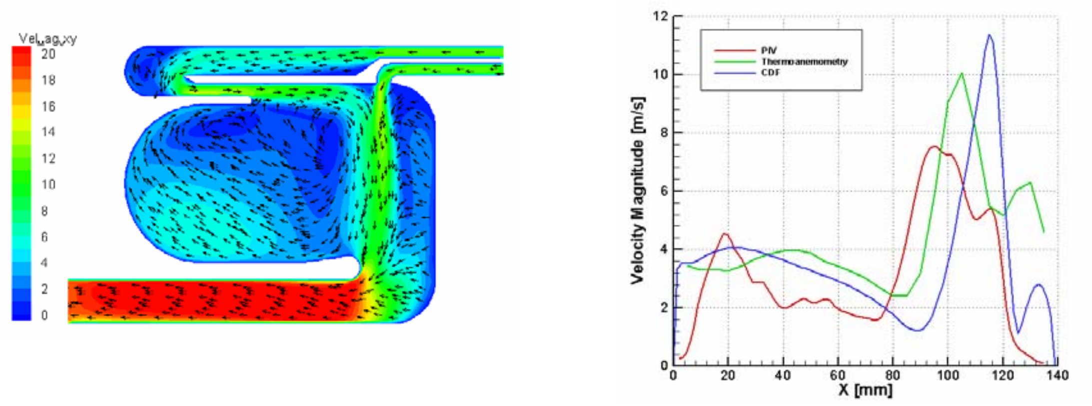


Fig 5.5 Averaged velocity fields measured by PIV method for configuration A (right) and C (left). Velocity magnitude indicated by colour contours.

To obtain additional information about sensitivity of the flow structure numerical simulation for the investigated experimentally flow and geometry parameters were performed. A commercial finite volume code Fluent 6 was used to generated velocity fields, to collect turbulence statistics and to visualize the flow paths. In the model a three-dimensional, incompressible turbulent flow of viscous fluid was investigated with k-e turbulence approach and Large Eddy Simulation, using about 822000 tetrahedral elements. The main futures of the flow, the velocity field and details of the velocity profiles obtained in the experiments from PIV and hotwire measurements were compared with their numerical counterparts. There is general agreement between experiment and numerical simulations, however details of the velocity profile close to the wall as well as turbulence intensity may differ up to 30%. It is partly due to the limited resolution of PIV method close to the walls but also it suggests necessity to verify physical modelling implemented into turbulence model used in the numerical code. Figure 6a shows an example of the numerical result obtained for the cold flow conditions.



Numerical simulation: velocity field

Velocity profiles

Fig 5.6. Numerical simulation, velocity vectors and contours of the velocity magnitude (left); numerical velocity profile at the center line compared with experimental data obtained from PIV and hotwire anemometry (right). Configuration A of Geometry 3

The full field measurements with PIV were accompanied by point measurements using hotwire anemometer. To validate numerical simulations, selected velocity profiles obtained from PIV velocity measurements and from hotwire anemometry were compared. Figure 5.6b displays an example of this validation for the centre-line of the combustor.

The main conclusion of the cold flow experimental and numerical tests is that only configuration A (both inlets open) and configuration C (lower inlet open) offers conditions for sufficient fuel mixing in the internal vortex. In purpose to intensify this process several tests were performed introducing turbulence enhancers at both inlets (see Fig. 5.7). Both experimental and numerical tests positively confirmed increased turbulent mixing in the inner part of the vortex and this modification of the combustor geometry was recommended to the consortium.

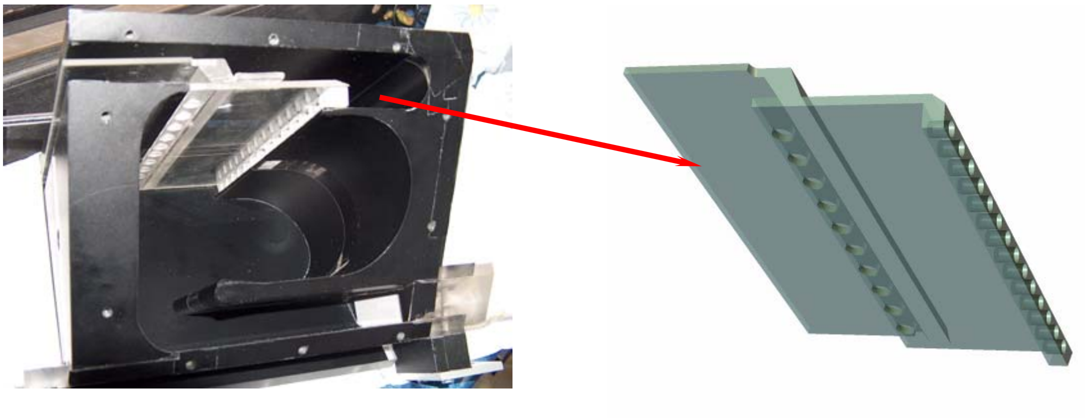


Fig 5.7 Plexiglas model (Geometry 3) with a turbulence enhancer (perforate plates inside inlets).

To quantify mixing properties of the flow high speed Particle Image Velocimetry was introduced. For this purpose 500 images taken at 4500fps were analyzed by PIV method to obtained sequence of temporary velocity fields. At the first glance, comparing selected velocity profiles one may already notice increased fluctuations for the flow with the turbulence enhancer (compare Fig. 5.8). Power spectrum density (PSD) of the full field fluctuations obtained for PIV measured velocity field clearly confirms this observation (Fig. 5.9).

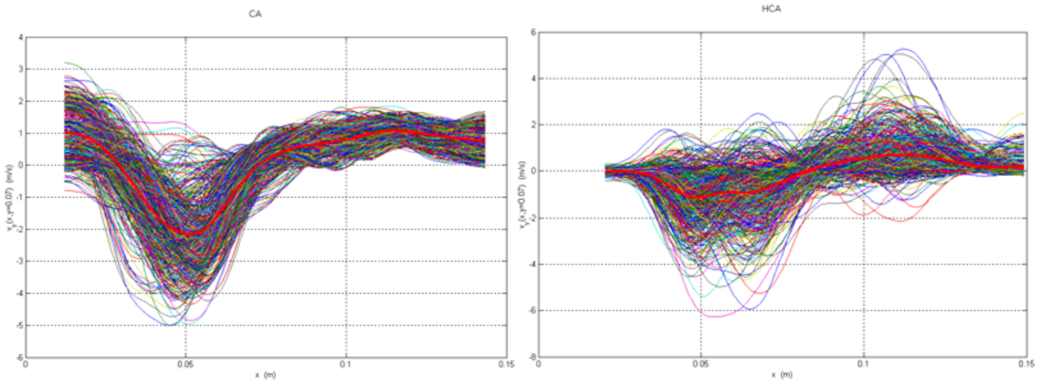


Fig 5.8. Superposition of temporary velocity profiles obtained at 4500fps for cold flow model (Geometry 3) without (left) and with (right) the turbulence enhancer. Vertical velocity profile for the center combustor line was extracted from 500 PIV measurements.

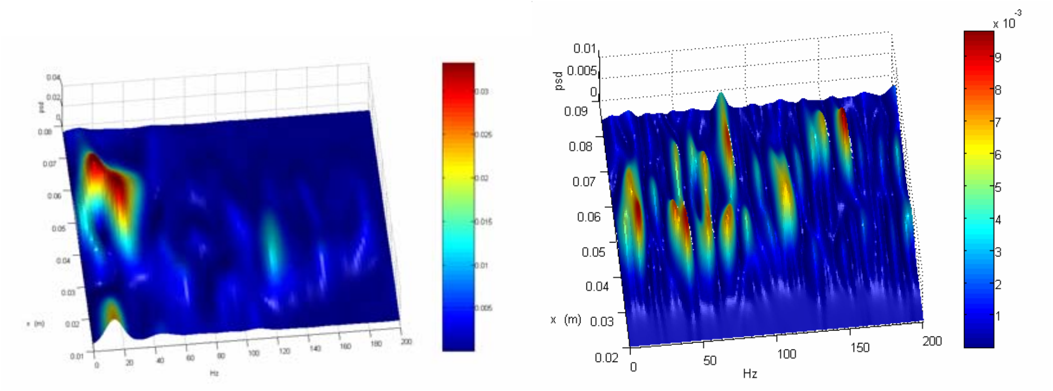


Fig 5.9. Power spectrum (PSD) of the turbulent fluctuations measured from the full field vertical velocity component for the cold flow model without (left) and with (right) turbulence enhancer. Vertical velocity for the center combustor line were extracted from 500 PIV measurements taken at 4500fps.

Deliverable D5.3. The interaction of the cooling jets with the main vortex flow. For better understanding mixing of a simulated fuel injection, the flow was observed using Schlieren apparatus and Mach-Zender interferometer. As a fuel hot air of temperature about 300°C was introduced through small opening in the combustor. Measurements performed indicated very short range of interaction of the jet injected with the main flow. Schlieren images of the injected hot air show that the jet is fully mixed after entrainment about 2-3cm into the combustor. This result has been verified using interferometer, however due to the low optical quality of the plexiglas walls only quantitative measurements were performed. To evaluate effect of jets cooling the cold air was introduced through a slit opening in the lower part of the combustor partition. Observations showed that the cold air injection has small effect and practically was negligible in Schlieren images. It appears that the main flow velocity, being one order of magnitude higher immediately redirects cooling jet into the exhaust channel, minimizing cooling effects on the wall. This observation were verified using numerical model with hot air inject in the location of the fuel injector and the cold flow jets located in small opening in the lower partition of the combustor. Figure 5.10 shows results of the simulation. It can be found that cooling effect of the cold jets is very local. However, in real combustion process the cooling jets can play important role screening lower part of the combustor form the hot combustion gases.

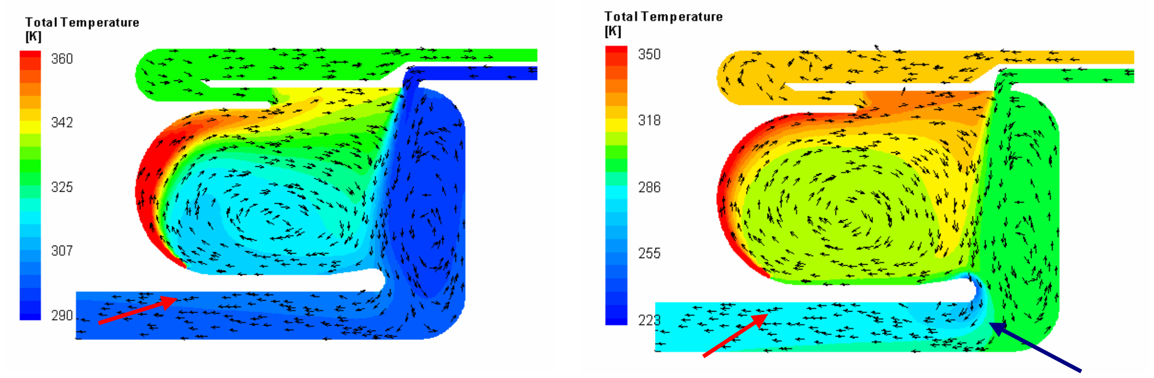


Fig 5.10 Simulation of the fuel injection – (left) and cooling jets - (right). Velocity field and temperature contours calculated for Geometry 3, configuration A (both inlets open) without turbulence enhancer.

2.2.5.3 Assessment of results and conclusions

Milestone M5.1 Experimental demonstrations of the main recirculation pattern, fuel injection and wall cooling characteristics. The cold flow measurements allowed for prediction of the flow structure, flow velocity, turbulent mixing and temperature distribution for several flow configurations and three basic geometries of combustor. The main outcome of the investigation is confirmation of the initial project hypothesis, assuming possibility to generate recirculating flow with the central vortex responsible for prolonged combustion time. Application of the turbulence enhancers is advisable at the inlet. Flow interaction with the fuel injector and cooling jets appears to me relatively week in the cold flow model. However, it can be expected that in real combustor prolonged combustion will be responsible for more uniform temperature distribution in the main vortex, decreasing sooth and NO_x contents of the combustion products.

Milestone M5.2 Quantitative comparison with numerical predictions of the recirculation patterns and effects of fuel injection and wall cooling. Validation of the numerical models is one of the important tasks of the project. Experimental data for the cold flow combustor were used for validation of turbulence models, answering number of questions concerning optimisation of the combustor geometry. Present experimental results show only qualitative agreement. Differences present mainly in the recirculation regions reach over 20% of the velocity magnitude. Large Eddy Simulation applied in the modelling improved its output and what is even more important, allowed to resolve unsteady fluctuation of the velocity field. These simulations clearly show presence of strong mixing zones, periodically injecting fresh air into the combustor zone. Similar effects of pulsating flow are well visible in the experimental flow fields and flow visualizations.

2.2.5.4 Acknowledgements

Involvement of Prof. F. Lusseyran (LIMSI-CNRS) in spectral analysis of the PIV images is gratefully acknowledged. Engagement of several our PhD students, specifically of S. Blonski and T. Michalek in main parts of the project is acknowledged.

References

[1] T. Michalek, S. Blonski, T. Kowalewski, “Cold flow model for gas turbine combustor sector”, Polish Fluid Mechanics Conference, Sept. 23-26, 2002, CD-ROM Proceedings.

[2] S. Blonski, „Analysis of of cold flow and turbulent mixing in a model of gas turbine combustor”, Warsaw University of Technolgy, Diploma Oct. 2002, awarded by FIAT (Italy).

[3] S. Blonski, „Cold flow in transparent gas turbine combustor model ”, PIVNET2 Workshop, Goettingen 6-7 Nov. 2002.

[4] S. Blonski, T.A. Kowalewski „FLOXCOM project for efficient gas turbine combustor”, seminar presentation at Warsaw University of Technolgy, Plock, 2003.

[5] T.A. Kowalewski, F. Lusseyran, S. Blonski, Flow structure identification with PIV and high speed imaging in cold flow combustor model, 4th Euromech Fluid Mechanics Conference, Toulouse 2003.

2.2.6 WP6:HOT PRESSURIZED TESTS OF THE SECTOR COMBUSTOR

WP leader: Ansaldo Ricerche Srl

Additional Participant: Ansaldo Caldaie

2.2.6.1. Objectives and strategic aspects

Following Technical Annex to the contract, Ansaldo Ricerche activities are focused on Work package 6: “Hot Pressurised Tests on the Combustor Sector”. The related objective consists in the measurements of the FLOXCOM combustor performances in pressurised conditions.
A numerical activity was also performed in order to assess numerical models and to interpret experimental data.

These objectives have been reached through:

- Experimental tests performed at pressurised burner condition by Ansaldo Boiler at its research combustion centre.
- Numerical simulation carried out by means of a thermo-fluid-dynamic code suitable for comparison with experimental data (FLUENT).

Strategic aspects are straightforward. Experimental activities at nominal condition provide a set of results concerning both performances and safety of the burner. On such a basis it is possible to give an assessment of the proposed technology against of state-of-the-art for gas turbine applications.
The performances of the combustor system have a great relevance in the gas turbine market competitiveness, affecting overall efficiency, reliability and compliance to the emission legislation.

2.2.6.2. Scientific and technical description of results

Activities can be subdivided in the two following groups (and related tasks of Technical Annex):

- ***Experimental testing (Tasks 6.1 – 6.4);***
- ***Numerical simulation and verification of the results (Task 6.5).***

Experimental testing

Research approach

Experimental testing was addressed to assessment of the burner; so the measured data were:

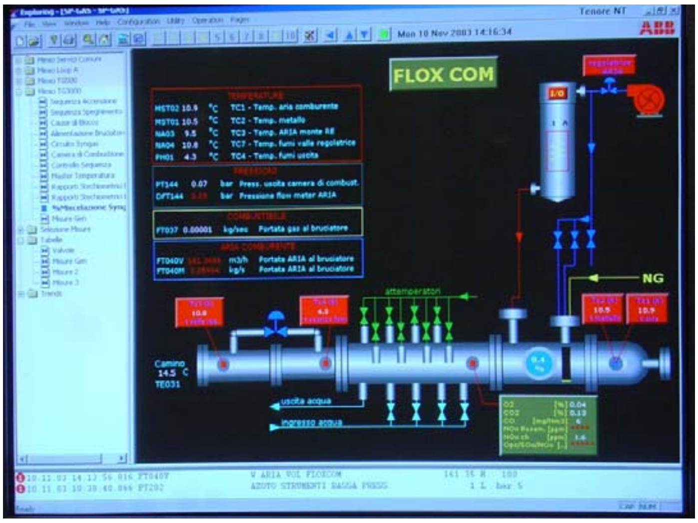
- The easiness to have a regular ignition;
- The efficiency of the burner, i.e. the rate of fuel conversion at different operative condition;
- The rich and lean limits;
- The overall NOx, CO and UHC emissions;

Providing an insight of the phenomena occurring into the combustor is another fundamental issue. Combustor sector was supplied with two optical windows that allow both videos and optical local measurements. The velocity field inside the burner was measured by LDV (laser Doppler anemometry) technique. The optical measurements in such combustion regime has a scientific relevance as only a few literature is available world-wide in these issues. A powerful argon laser equipment have been used; great care has been directed booth towards system calibration activity and insemination system, in order to maximise date rate and to lower noise and non-valid data. Than, we assessed also fundamental safety characteristics of the combustor, monitoring pressure values during test and by an exam of the combustor structure after measurements. A complete suction line was designed in order to bring the exhaust to the conditioning of the gas analysis system. In particular, a specific water-cooled gas sampling probe was designed and built-up. Thermo-graphic equipment has been used to identify hot spot that could have a negative impact on the safety of the test (Fig 6.3).

Work performed

In order to test the combustor sector (Fig 6.1), a design of the lines supplying pressurised pre-heated combustion air, fuel, cooling air, cooling water and quenching water have been set-up. The combustion-air line was assembled by mounting a 200kW electrical pre-heater and two ducts that guarantee the right balance to the combustor inlets in mass flow rate and temperature. The quenching line and the cooling water line were designed from scratch. A new configuration of the DCS control system and data acquisition was prepared in accordance with the needs of the tests, in order to carry out both measurements and operational setting in remote mode by a computer (Fig 6.2). Combustor sector is mounted in a closed room dedicated to pressurised combustion testing (Fig 6.4); the room was equipped with safety devices like gas detector and flame extinguish systems.

Fig 6.1 Pressurised test rig. Detail of discharge section (left) and air inlet tubes (right)



Mon 10 Nov 2003 14:16:34

MST02 10.9 °C TC1 - Temp. aria comburente
MST01 10.5 °C TC2 - Temp. metallo
NA03 9.5 °C TC3 - Temp. ARIA monte RE
NA04 10.8 °C TC7 - Temp. fumi valle regolatrice
PH01 4.3 °C TC4 - Temp. fumi uscita

PT144 0.07 bar Press. uscita camera di combust.
DPT144 [illegible] bar Pressione flow meter ARIA

FT037 0.00001 kg/sec Portata gas al bruciatore

FT040V [illegible] m3/h Portata ARIA al bruciatore
FT040M [illegible] kg/s Portata ARIA al bruciatore

O2 [%] 0.04
CO2 [%] 0.13
CO [mg/Nm3] 6

Fig 6.2 Picture of the remote control system

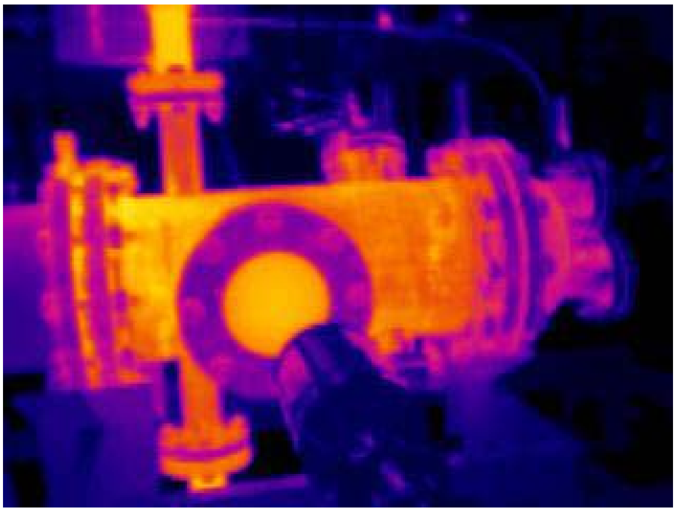


Fig 6.3 Thermography of the rig

Fig 6.4 Laser room

Laser ray was transmitted by optical fiber, which allows placing laser equipment in a separated room (Fig 6.4).

The main data of the test rig are reported in the Tab. 1.

Table:1 Test rig main data

FLOXCOM Combustion test rig Main Capability Data
Air line Maximum pressure: = 7 bar Maximum mass flow: = 2350 Nm3/h Compressor power: = 273 kW
Fuel line Maximum pressure: = 16 bar Maximum mass flow: = 400 Nm3/h Compressor power: = 37 kW
Air preheating. A 200 kW electrical heater will be placed close to combustor sector
Control systems: Air: pressure, temperature, and flow rate automatically controlled by a closed loop; Fuel: pressure automatically controlled by a closed loop, mass flow rate controlled remotely; Cooling system temperature and flow rate were automatically checked for safety reasons; Two separate water line, for quenching and for discharge tube cooling were designed; Safety: fuel line is automatically closed by stop-valves without the consensus of UV sensor. Control system also takes into account the pressure of quenching water and the flow rate of cooling water, it automatically inhibit the fuel flow if a hazardous condition occurs.
Diagnostic Thermocouples for metal and gas temperature measurements; Gas sampling and analysis of the exhaust before quenching; Pressure transducers for inlet and outlet pressure measurements; Optical measurements by laser system (carried out by Ansaldo Ricerche)

Results

After a characterization of pressure drop across different section of the test-rig, a first series of experiments were carried out at atmospheric pressure in order to assess the ignition and a general behavior of the combustor. Ignition occurred regularly and combustion also took place silent and completely. Potential dangerous explosion condition due to the presence of unburned fuel could be avoided by taking care of the ignition system.

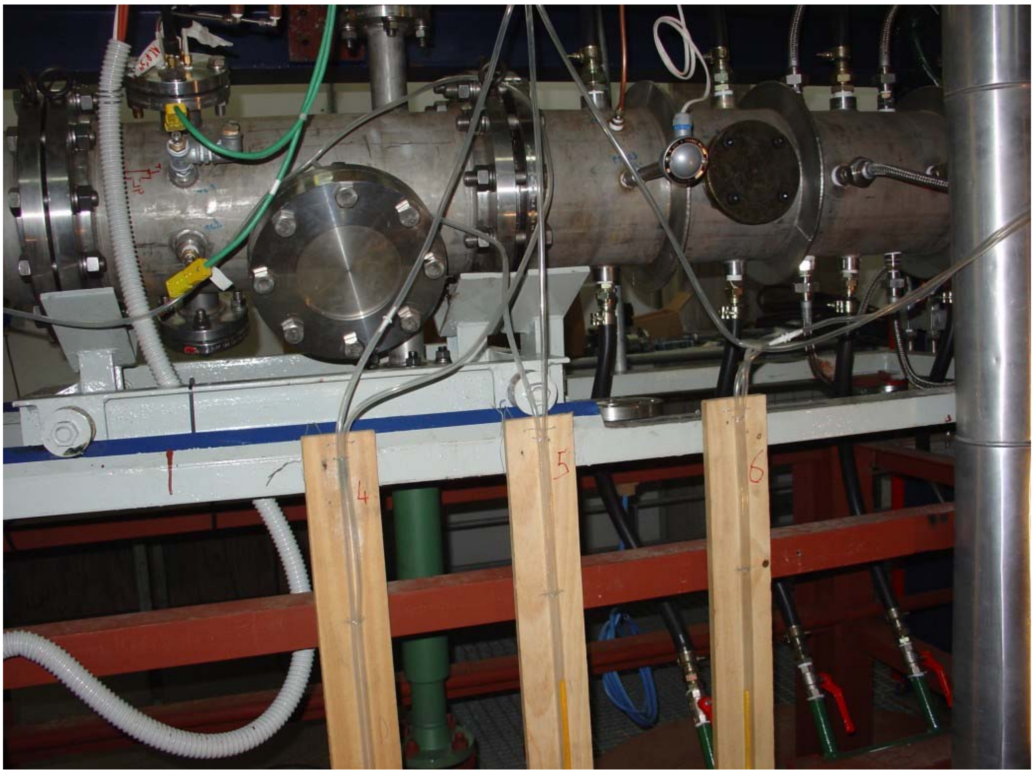


Fig 6.5 Pressure drop across the rig: characterization

Fig 6.6 Internal view of the combustor section

Tab. 2. Main experimental results of FLOXCOM sector

As far as pressurised condition, we had the following overall results:

Pressurised operative condition	
General flame behavior (video analysis)	Flame shape and colors appreciably changed with the air-to-fuel ratio (AFR). Two different regions could be identified. The first characterized by blue flame and stoichiometric combustion at all AFR, the latter changing with AFR.
Lean – Rich limit	The combustor bore a wide range of AFR (from 17 to 80) without showing any stability problem. However, Floxcom sector produced a big amount of CO when it was managed at high AFR.
Emissions	in pressurised condition 2.5 bar
Normalised NOx	17 ppm
Normalised CO	846 ppm
Normalised UHC	Negligible
Noise / vibration	Combustion noise was negligible
Wall temperatures	During the test wall temperature looks always low enough; after the tests, the sector was inspected even the surfaces exposed to high temperature were found in good condition.

The FLOXCOM sector behaves quite good if we look at the ignition easiness and at mechanical and thermal stresses. NO_x emissions are also very low, conversely CO emissions are always much higher than the state-of-the-art combustors.

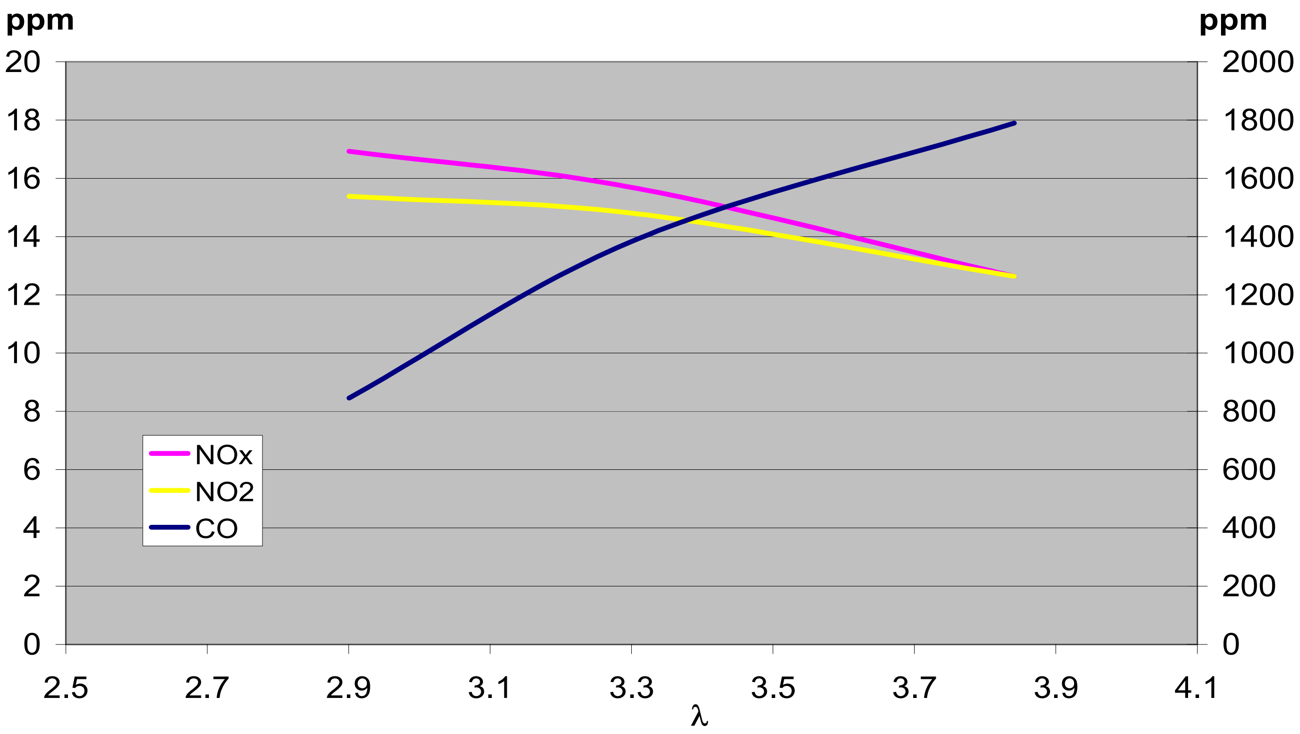


Fig 6.7 Emission test diagram.

During the test, often the CO emissions approached the limit of 2000 ppm and even if we did not find unburned hydro-carbon, we can say that the combustion efficiency needs to be improved. In addition, pressure drops inside the burner look too high to use this kind of burner in standard gas turbine.

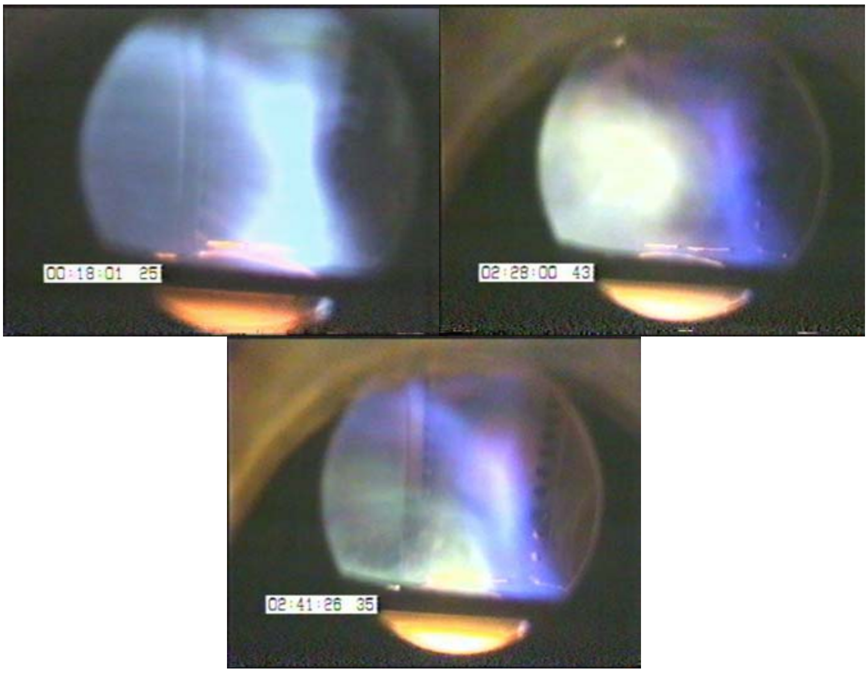


Fig 6.8 - Flame pictures: λ=2.3, λ=2.1, and λ=3.0 respectively.

On the other hand was an unbelievable surprise to found so wide stability region for a non-diffusive flame. In some conditions, the flame shape was typical of a premixed flame, but in some other, the rich combustion zone slowly vanished. It is difficult to say if it is a real flameless or a very lean premixed flame, otherwise, in our humble opinion, this burner shows the possibility to be improved to reach the aim to bring the flameless oxidation technology in gas turbine combustion.

Numerical simulation

Research approach

Combustion simulation is addressed to the following aims:

- to aid the interpretation of the experimental data;
- to validate models and thus to use numerical simulation for further scaling or modification of FLOXCOM combustor;

The following approach has been used;

- use of a state-of-art commercial code (in this case FLUENT);
- sub models for combustion simulation (i.e. reduced mechanism we developed for lean pressurised combustion of natural gas) tuned against experimental data in on gas turbine conditions.

Since "flameless or MILD combustion" regime does not lay in the premixed regime nor in pure diffusion one, we avoided to use combustion models expressively built for these conditions (such as flamelets models, G equation model, diffusion models,…).

We used, instead, an Eddy Break Up (EBU) models; despite its physical weakness [1] it is successfully used in gas turbine research [2]. The major advantages of EBU is that it can takes into account both turbulence and chemical kinetics phenomena.

The major drawback is that it requires a tuning of both turbulence constants as stated by [3]. In such case tuning is possible through comparison with experimental data.

Work performed

We performed both two and three dimensional simulations; the former were only utilised to aid the combustor test section safety (i.e. wall temperatures were calculated), the latter addressed to model validation and experimental data interpretation. Three dimensional simulations were required because of the design of both air and fuel injection systems.

Grids have been built in different modules. The zones of the combustor affected by wall boundary layers are represented by structured grid; non-structured or hybrid methods are used far from combustor walls in order to reduce cells number and the related computer effort.

Each calculation reached convergence regularly; EBU combustion submodel has been utilised with both two and three step mechanism. We use respectively the two step mechanism by Westrbrook and Dryer [3] and three step mechanism Ansaldo Ricerche proprietary, tuned for lean pressurised condition.

Radiation heat transfer is considered through FLUENT sub-model suitable for non-transparent gases.

The three dimensional analysis were focused on two type of burner, say “Type A” and “Type B”, which are the final step of the design process carried out into the project. The air streams are introduced from a series of holes, which affect mixing and turbulence in the reaction zone. Therefore a grid refining is provided in the zone in front of such holes in order to have a “grid independent” simulations.

Solid walls of the combustor is also included in the domain and related conjugate heat transfer is calculated.

Results

The results confirm the main conceptual issues of the burner.

- A recirculation zone is well established inside the burner, mainly composed, in stationary conditions, by flue gas produced by combustion. This allows to mix fuel with hot combusted products before fuel injection (air rate is low in this zone) and therefore to prepare a mixture suitable for MILD combustion condition (high temperature, high fuel dilution). This feature is confirmed by temperature contours fields reported in the Fig. 6.9.

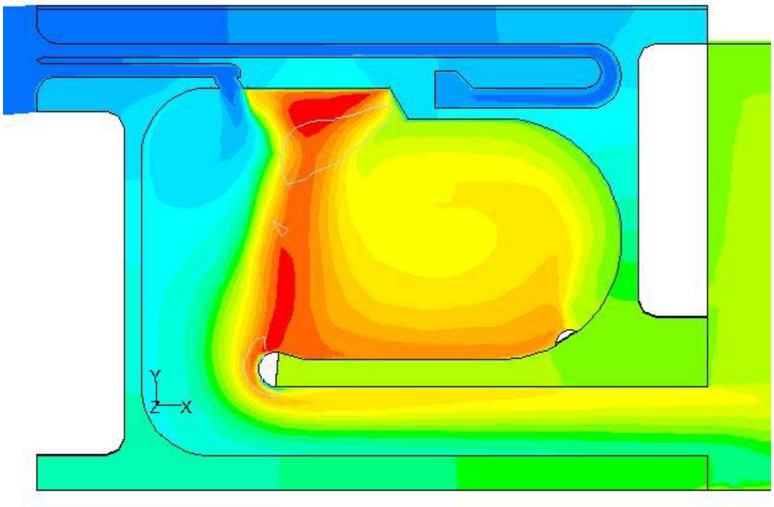


Fig 6.9 Temperature field in axial plane

Flame takes place in the zone of Fig. 6.9 where temperature contours have higher values. The shape of the flame is similar to the one shown in the Fig. 6.8 (left figure). Higher temperature zone is placed far from fuel injection. It confirms that the design is suitable for the right balance between mixing time (between fuel and recirculated flue gas) and injection delay of the mixture. Combustion is complete being no unburned hydrocarbon nor CO yield at the combustor exit. This last result is different from experimental data and it requires further investigations. Wall temperature is below of the material threshold.

2.2.6.3 Assessment of Results and Conclusions

A totally novel concept burner FLOXCOM has been tested in the operative condition; diagnostic were arranged to provide for burner performances and to give an insight of combustion phenomena.

Tests reveals some fundamental data for gas turbine application.

- Ignition occurred regularly and complete combustion takes place without noise.
- Stability: the burner shows no tendencies to produce flash-back or blow-off in a wide range of air to fuel ratio.
- Emission: NO_x emissions are quite low, CO emissions are instead very high thus suggesting to review the design internal recirculation.

A quite interesting feature of the burner that fuel is injected directly inside of combustion chamber; therefore, no any fuel preparation is required. Thus, the FLOXCOM concept is quite promising for non standard fuel, i.e. characterised by high flame velocity and not suitable for premixing; among this all fuel that have an high hydrogen content.

Numerical simulation associated to test is also provided. The comparison with experimental data allow to tune and validate models.
Since a good agreement is obtained, it is possible to perform design modification and/or scaling with the aid of numerical simulation.

References

[1] N. Peters, Four Lectures on Turbulent Combustion, ERCOFTAC Summer School, 1997, Aachen;
[2] M. Brandt, W. Polifke, B. Ivancic, P. Flohr, B. Paikert, "Auto Ignition in Gas Turbine Burner at Elevated Temperature", ASME TE 2003, GT2003 – 38224.
[3] *Ibid. 1*
[4] Westbrook, C. K.; Dryer, F. L., Simplified Reaction Mechanisn-for the Oxidation of Hydrocarbon Fuels in Flames, Combustion Sience and Technology, 1981

2.2.7 WP7: MODELING AND OPTIMISATION OF CONVECTIVE WALL COOLING, WALL TEMPERATURE AND STRESS ANALYSIS, TURBINE INTERFACE

WP Leader:	B&B AGEMA
Additional Participant:	RWTH

2.2.7.1 Objectives and strategic aspects

The thermal design of a combustion chamber is one of the most important issues for the reliable and secure operation of a combustion chamber. Thus, the work package includes the numerical calculation of the thermal load level for the FLOXCOM combustion chamber under basic operating conditions. Based on the results, the detection of possible deficiencies in the thermal design is possible and improvements can be obtained. Therefore, it is the aim of the calculations within this work package to optimise thermal loads in order to allow a combustion chamber design, which is basically able to reach high annual basis availability and reliability.

Further aspects of the calculations deal with the development of innovative cooling technologies, which improve thermal protection of the combustion chamber walls (innovative film-cooling technologies), and the design of a cooling technology, which is able to support the central vortex of the combustion chamber. The central vortex is of great importance for operating the combustion chamber in the flameless oxidation condition.

In order to reach the objectives, 2-D and 3-D computational fluid dynamics are used. The prediction of the thermal load is achieved by the conjugate calculation technology. This is an innovative calculation strategy, which allows the simultaneous calculation of the fluid flow region and the temperature field in the solid walls. The main advantages are that it is not necessary to apply heat transfer coefficients for calculating the heat transfer and that the interaction of fluid flow and heat transfer is taken into account.

In order to be able to reach basic results for the design specifications of the cooling model of the combustion chamber, a basic cooling study with different parameters has been performed. Furthermore, a small-scale model has been derived from the basic geometry of the combustion chamber. The model has been tested by a partner (IDG of RWTH Aachen) of the FLOXCOM consortium. Based on the testing, the reliable operation of the film-cooling technology has been validated. Finally, the final geometry of the combustion chamber has been modelled completely and a full 3-D simulation has been performed.

2.2.7.2 Scientific and technical description of the results

The WP contains 3 Deliverables. Major results of the Deliverables are summarised.

2.2.7.2.1 D7.1 – Design specifications for the pilot combustor from thermal load point of view

Basic wall cooling studies (numerical approach) have been performed in order to achieve the design specifications for FLOXCOM combustion chamber. Main aspects of the calculations deal with film cooling efficiencies of different film-cooling hole geometry and interaction of cooling jets and main flow. Following numerical studies have been performed (details in deliverable **report D7.1**):

7.2.1.1 2-D basic model of FLOXCOM combustion chamber with film cooling

7.2.1.2 2-D design study for the small scale experiments including the calculation on the film cooling performance of a thin vane in a hot gas duct

7.2.1.3 3-D study on the film cooling performance through a row of holes (different hole geometries) in a rectangular hot gas duct

7.2.1.1 Two-dimensional calculations with wall cooling ejection have been performed for the initial geometry obtained from the project partner TECHNION. A 2-D numerical grid consisting of 10 blocks with 14000 grid points has been used for flow simulation in that geometry including cooling air injection. Within the calculation the fuel injection has been replaced by the cooling air injection at the same location.

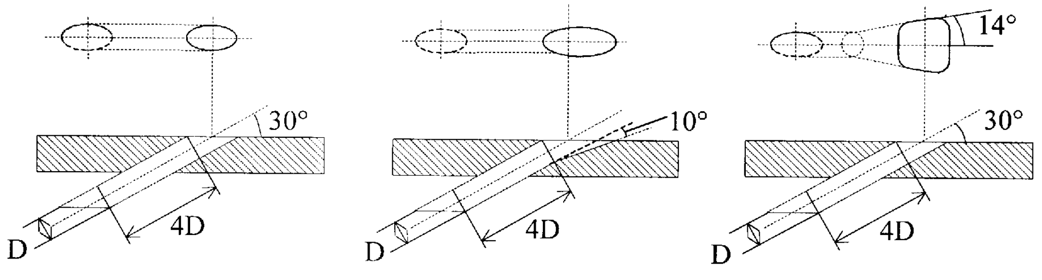
Due to the concave surface and the circulation of the central mixing vortex the cooling fluid keeps attached to the wall. An effective cooling of the combustor wall is established. As the interaction with the main flow is very low (the central vortex is weak), an improvement is necessary. From the total pressure distribution it becomes obvious that a large recirculating area at the inlet of the exhaust duct is existent and an improvement of the geometry design becomes necessary, too. Detailed description of the results can be obtained from deliverable report D7.1.

7.2.1.2 Numerical calculations have been performed for the geometry of the small-scale model. Results are summarised in 7.2.2.3.

7.2.1.3 In this study, a hot gas duct flow with cooling fluid injection through one row of 8 cooling holes is investigated. The hole geometry comprises different configurations with cylindrical holes and shaped exits (diffuser and fan-shaped, figure 7.1). The duct wall with cooling fluid ejection is calculated with two different heat transfer boundary conditions within the same calculation. The first part (holes no. 1 to no. 4) is calculated with the conjugate heat transfer condition. The second half of the wall (holes no. 5 to no. 8) is calculated with the adiabatic wall condition. The cooling fluid is supplied to the holes by a rectangular duct from one side with a 90° angle to the streamwise direction.

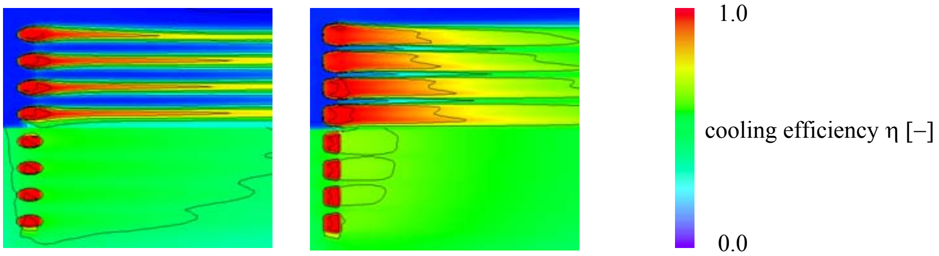
The secondary flow vectors for the cylindrical hole configuration reveal the counter-rotating vortices of the kidney-vortex system. For the diffuser hole configuration, it can be shown that the secondary flows are significantly reduced. In the case of the fan-shaped hole, the momentum of the injected cooling fluid is decreased significantly and, thus, the interaction of the main flow and the cooling jet is reduced leading to weak secondary flow structures. Furthermore, the cooling fluid spreads out into the lateral direction.

Figure 7.2 shows for the adiabatic surface of the duct wall (upper half) the adiabatic cooling effectiveness. In the case of the conjugate part of the duct wall (lower half), the values of the effectiveness give a dimensionless surface wall temperature. The adiabatic effectiveness shows the significant improvement of the cooling performance for the shaped configurations whereas the lift-off in the cylindrical case leads to a very poor performance. Furthermore, the improved lateral extension of the single jets for the fan-shaped configuration keeps only a very narrow region of reduced performance between the jets.



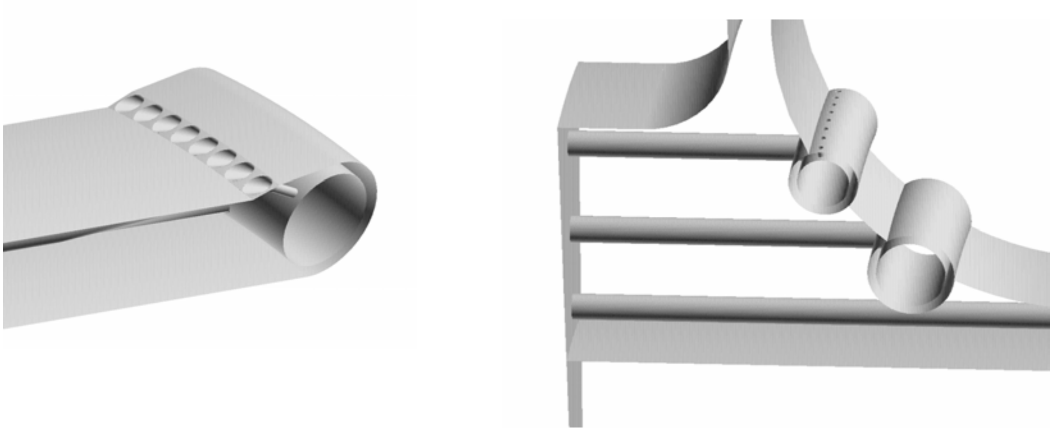
Cylindrical hole Diffuser hole Fanshaped hole

Fig 7.1 Investigated hole geometries



a) cylindrical holes b) diffuser holes c) fan-shaped holes

Fig 7.2 Calculated film-cooling efficiencies



a) Innovative film-cooling design b) Coolant injector system (CIS)

Fig 7.3 3-D CAD models of cooling concepts

7.1.2.4 Major design specifications

On basis of these studies, the design specifications for the combustor were derived. The main specifications can be listed as following:

- Film cooling of the "planar" region in the lower part of the combustor, which is separating the chamber itself from the exhaust duct. The cooling air is ejected along the inner (hot) side of that region through a row of holes. Based on the 3-D cooling study it has been decided that a shaped hole exit is necessary in order to establish a direct wall contact of the cooling film (CAD model in Fig 7.3a). An alternative cooling geometry with small cooling slots has also been considered.
- Film cooling of the convex part of the chamber is established by an additional row of cylindrical cooling holes. The ejection angle of the holes is designed to establish a cooling

film for region behind the ejection and to support the main vortex in the chamber (CAD model in Fig 7.3b).

- Additional convective cooling is established by the cooling air supply system.

A detailed description of the geometric design specifications is given in the report on the deliverable (D7.1).

7.2.2 D7.2 – Evaluation of the results from operation of the small-scale model

Detail description of experiment and numerical results can be obtained from deliverable **report D7.2**.

7.2.2.1 Design of hot gas experiment

A film-cooled vane configuration has been designed. It has been tested in a rectangular duct with hot gas flow. The shape of the vane is designed to lead to similar flow conditions on its pressure side as in the combustion chamber. Thus, it is possible to investigate the cooling fluid injection and its influence on the flow field. The injection configuration (Coolant Injector System - CIS) has been designed for very high blowing ratios and, thus, leading to very high velocities for the ejected cooling fluid. The purpose of this technique is to support the main flow by the additional momentum of the cooling fluid and, thus, to accelerate the main flow. Therefore, this configuration is likely able to enhance and to stabilise the main vortex in the FLOXCOM combustion chamber.

A rectangular hot gas duct test rig exists at RWTH Aachen (Partner 8), where the experiments have been performed. The original hot gas duct has been designed for investigations on a steam-cooled test vane. It has a rectangular shape and 3 vanes in a cascade are installed. The central vane can be supplied by cooling steam or air. For the new test rig, the central vane has been replaced by a new configuration in order to establish similar flow conditions as in the main vortex region of the combustor.

Within the design of the test geometry different configurations have been considered:

a) turbine vane with pressure side ejection (design I, Fig 7.4a)
b) central thin vane with straight long leading edge (design II, Fig 7.4b)
c) same as b) but in a cascade (design IV, Fig 7.4c)
d) central thin vane with incidence (design IV, Fig 7.4d)

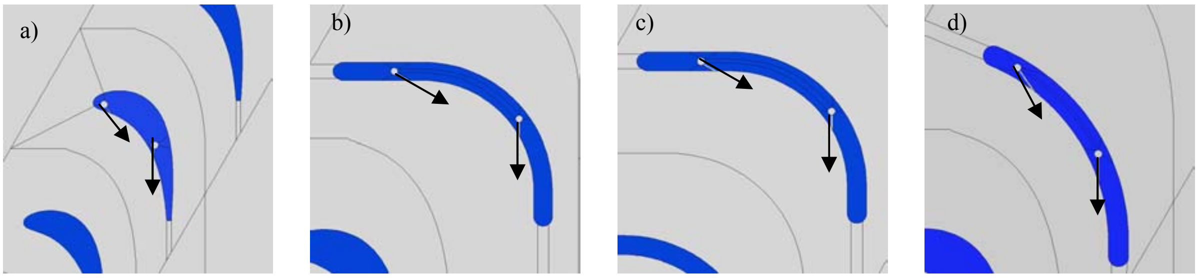


Fig 7.4 Design configurations for cooling experiment

Due to aerodynamic results of the numerical simulations and due to geometric restrictions of the test rig, design IV has been applied for testing.

7.2.2.2 Test facility and experimental conditions

The test rig was built up in the Institute of Steam- and Gas Turbines at Aachen University. The compressed air is delivered by the IDG Compressor plant consisting of 2 compressors with a maximum electrical power of more than 2 MW. For the current tests, the air comes into the test rig with a pressure of 2 bars and a temperature of about 350 K. The air is heated up by an oil driven combustor to a temperature of 750-850 K. After this, the hot air pressure is regulated by a hot gas throttle and the mass flow is determined by pressure and temperature measurements before and after an orifice according to DIN 1952. The hot air enters the test section with a pressure between 1.13 and 1.32 bar, a temperature between 723 and 823 K and a mass flow of 550 resp. 750 g/s. Because of the pressure losses inside the test section, the pressure between the test section and the exhaust drops down to values between 1.10 and 1.26 bar. The cooling air is delivered to the test section by and external, separate compressor with a pressure between 1.6 and 2.9 bar, a temperature between 475 and 575 K and a mass flow of 2, 3 or 4 g/s. The cooling air mass flow enters the test section through two holes with a diameter of 2.5 mm and is divided into the cooling jets by 17 holes of 0.8 mm diameter each in two rows. The distance between two cooling holes is 3.2 mm.

The measurement technique at the test rig can be divided into three parts:

At first the mass flow of the hot gas and the cooling air are determined by using orifices according to DIN 1952. Therefore the pressure before and behind the orifice as well as the temperature of the hot gas is measured and the mass flow is calculated within the data acquisition system using an iterative code. The second type of measurement is the temperature and pressure measurement before and after the test section, which builds together with the mass flow the boundary conditions for the numerical simulation of B&B-Agema. The temperatures inside the cooled wall section a measured by 11 thermocouples, 4 on the outer / or "suction" side (SS) and 7 at the inner / or "pressure" side (PS) of the cooled wall section.

The measurements were made for two different hot gas temperatures, 723 and 823 K, two different hot gas mass flows, 550 and 750 g/s and for three different cooling gases mass flows (2, 3 and 4 g/s).

7.2.2.3 Experimental and numerical results

The temperature depends mostly on its axial position and more or less not on the fact if the measurement is on the outer or inner side of the wall section. The slopes of the temperature curves are very similar for all twelve operation points. Between the first and second row of cooling holes the temperature drops by approx. 50 K. A second drop behind the second row of cooling gas holes, which could maybe been expected was not observed. The level of the temperature slopes depends on the boundary conditions. As expected, the level of temperature inside the wall decreases with increasing cooling gas mass flow. In addition, what is to be expected too, the level increases with increasing hot gas mass flows and hot gas temperatures. The difference in the temperature levels between similar operation points at different hot gas temperatures (approx. 85-90 K) is slightly lower than the difference of the hot gas temperatures itself (100 K), because the cooling air temperature is not increased as well.

Two-dimensional numerical analyses for several of the operating points of the test configuration have been carried out in order to get information of the momentum augmentation effect and for comparison of the thermal load of the configuration.

The comparison of the 2D-calculation and the measurement reveals a deviation in the results. As discussed in detail in deliverable report D7.3, this difference is not unexpected. Although the blowing ratios have the same values in experiment and calculation, the mass flow ratios are quite different, because it is not possible to have similar mass flow ratios in a 2-D calculation for a 3-D configuration with holes. The mass flow ratio in the experiment is only 0.27 % whereas for the 2-D calculation it is approximately 3.8 % because the holes are to be assumed to be slots over the full height of the passage. Thus, much lower temperatures are to be found in the calculation. Furthermore, the cooling film in the test configuration is only established for the mid-part of the vane. Therefore, heat conduction from the upper and lower part of the vane will increase the temperatures, too. This effect can also not be included in the 2-D calculation.

7.2.3 D7.3 – Numerical simulation of wall cooling including combustion aerodynamics

7.2.3.1 3-D Model I

For the complex 3-D calculation of the designed cooling ejection system and the combustion chamber itself, it has been necessary to generate a huge 3-D grid. The 3-D grid is a multi-block grid with 174 blocks containing over 2.6 million grid points. With respect to the thermal load prediction a conjugate calculation has been performed. Thus, the numerical grid contains not only the fluid flow region but also the solid walls.

Figs 7.5a-b give an overview on the achieved results. As it can be seen clearly the cooling air ejection leads to an effective cooling film along the combustor walls as indicated. As a result, thermal load level for the walls (green, yellow colours) is much lower than the hot gas in the chamber (red colour). Temperatures in the flow inlet region are significant cooler as the supplied fresh air is of low temperature (blue colour). Therefore, in this region no critical thermal load can be observed. High thermal load in the combustor wall at the right hand side (red colour) is only the result of the model, because heat for combustion is introduced directly by a homogeneous fill-up of the chamber with a constant high temperature. In reality, the wall on the right hand side will be covered by cool air coming from inlet.

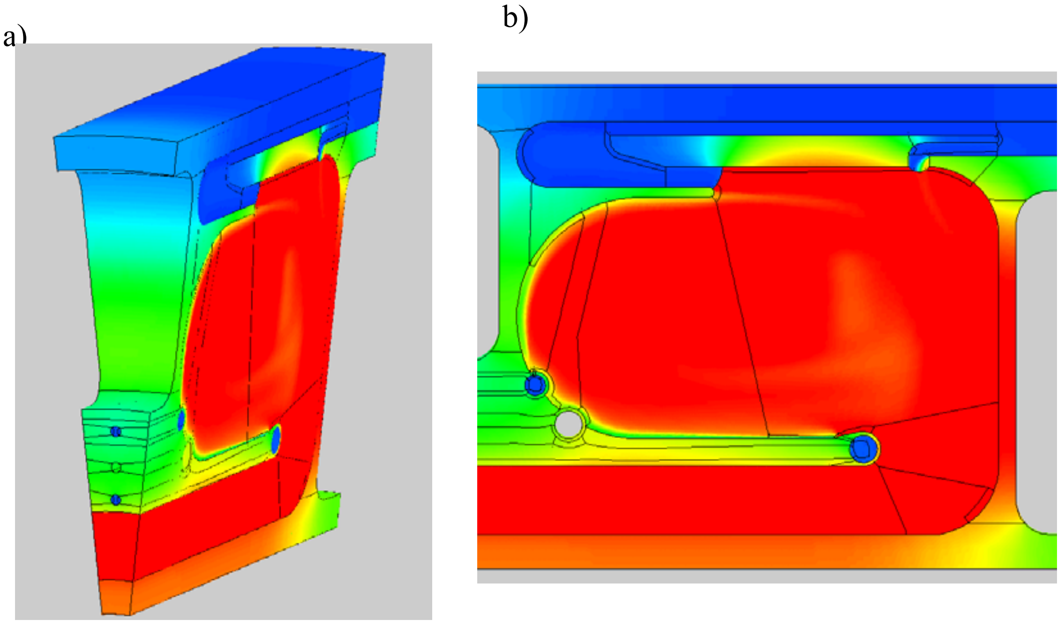
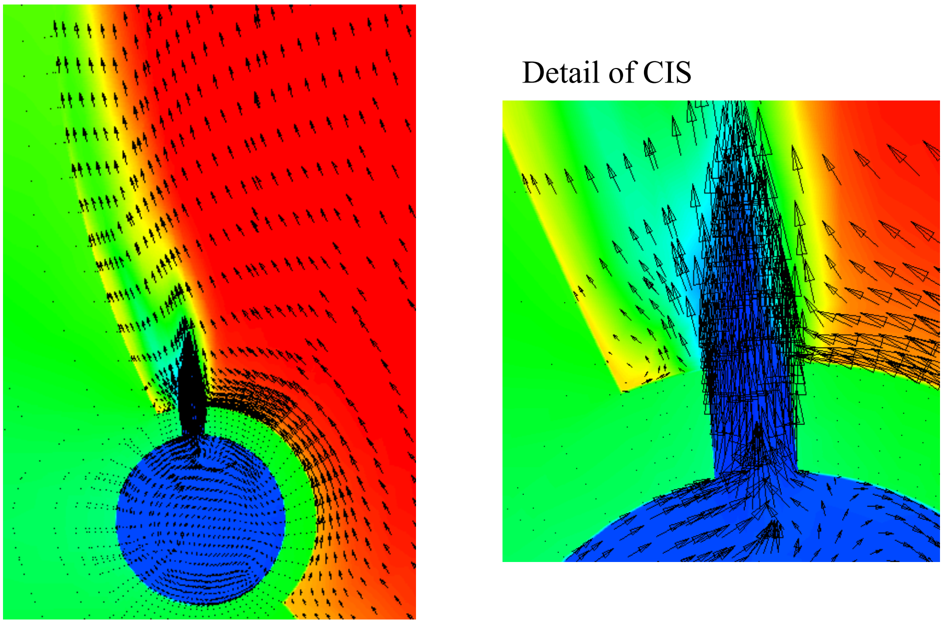


Fig 7.5 Full 3-D conjugate thermal load prediction (temperatures)

Figure 7.6 shows the cooling ejection results for the CIS. Here, it also can be seen that a cooling film can be established along the wall. Nevertheless, some improvement is necessary because the ejection leads to some mixing with the hot combustion air. An improvement is possible by a change of the ejection angle. More detailed results can be obtained from deliverable report D7.3



Detail of CIS

Fig 7.6 Cooling air injection from CIS

7.2.3.2 3-D Model II

Full 3-D conjugate calculations have also been performed for the final geometry of the FLOXCOM combustion chamber. The calculation contains the chamber, the complete fresh air supply system and the exhaust gas system. Thus, it has been possible to detect the aerodynamic behaviour and main pressure losses. Figure 7.7 shows the numerical grid for the calculation model. It contains 3.6 million grid points.

Figure 7.8 is an example result for the flow field (velocity) in a 2-D section. Thus, it is possible to detect several vortices, which increase pressure losses in the supply system for the combustion chamber. Detailed results can be obtained from an internal report (see reference list).

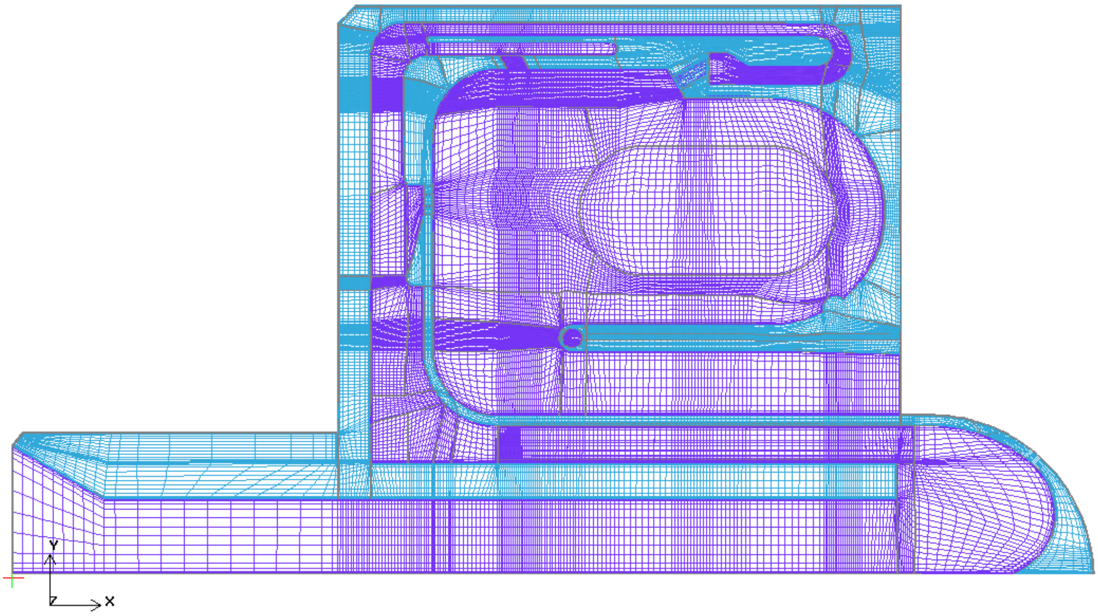


Fig 7.7 Numerical grid for 3-D Model II (2-D section).

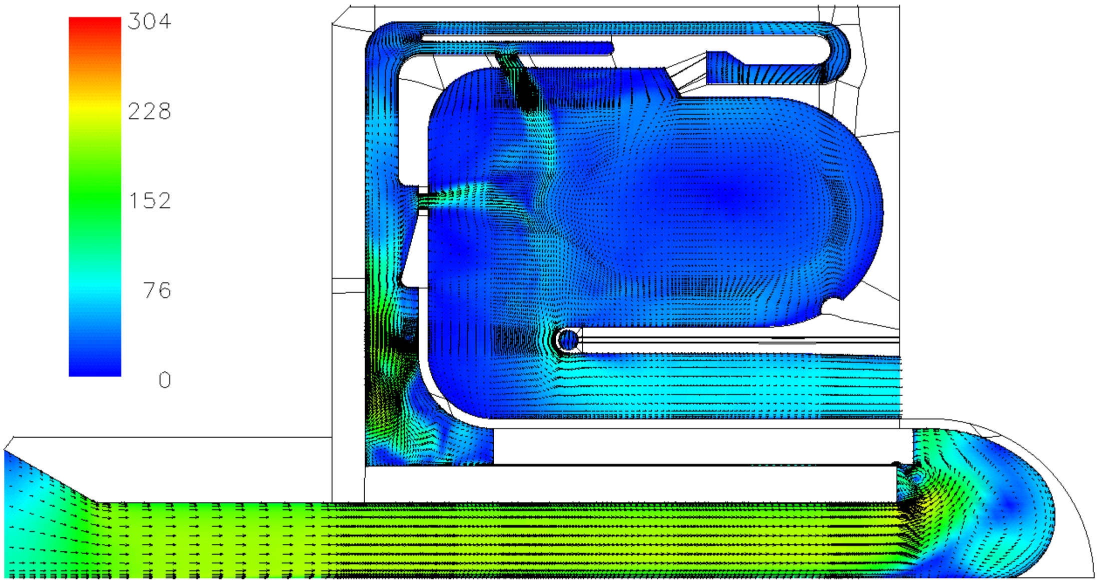


Fig 7.8 Velocity field (m/s) in 2-D section.

7.3 Assessment of results and conclusions

The numerical method has been used successfully for the prediction of the thermal load of the combustion chamber walls. Due to intensive convective cooling no general thermal problem has been detected for the test operating of the combustor. Nevertheless, additional effective film-cooling is recommended for some special parts of the combustion chamber (see 7.1.2.4 with respect to innovative film-cooling). An increase of film-cooling efficiency can clearly be achieved by application of shaped holes instead of simple cylindrical holes. The basic cooling study on different cooling hole geometry has detected a significant increase in cooling efficiency. The experience achieved within the numerical calculations provides an increase in knowledge for B&B-AGEMA, which is also of great interest for other cooling applications.

Furthermore, the effect of a Coolant Injector System (see also 7.1.2.4) on the main vortex of the combustion chamber has been investigated. Despite very high blowing ratios and momentum ratios, the cooling fluid is able to protect the chamber wall without separation of the coolant flow. The results demonstrate that the CIS is able to support the vorticity of the central vortex and, thus, is helpful to stabilise the vortex, which is essential for the flameless oxidation condition in the chamber. Nevertheless, the interaction of additional cooling air and the injection of the fuel might lead to local hot spot regions as the oxygen level is increased. Thus, it has to be avoided that large amounts of fuel injected in front of the cooling air comes into direct contact with the cooling air. With respect to the fuel injector, higher injection angles are recommended whereas the injection of cooling air is more or less tangential to the combustion chamber walls.

References

More detailed description of the results can be obtained from the deliverable reports and from following publications within this work package:

/1/ D. Bohn, J. Ren, K. Kusterer, “ Conjugate Heat Transfer Analysis of Cooling Jets Ejected from a Row of Shaped Holes”, 6th ISAIF Conference, Apr. 7 - 11, Shanghai, China

/2/ D. Bohn, J. Ren, K. Kusterer, “ Influence of Conjugate Heat Transfer on Film Cooling”, 5th European Conference on Turbomachinery, 18-21 March, Prague, Czech Republic, pp. 475 - 485 of the proceedings

/3/ K. Kusterer, D. Bohn, J. Ren, “ Conjugate Heat Transfer Analysis for Film Cooling Configurations with Different Hole Geometries”, ASME Turbo EXPO 2003, paper-No. GT2003-38369, June 16-19, Atlanta, USA

/4/ D. Bohn, J. Ren, K. Kusterer, “Systematic Investigations on Conjugate Heat Transfer Rates of Film Cooling Configurations”, Proceedings of the 10th International Symposium on Rotating Machinery, paper-no. ISROMAC10-2004-138, March 7- 11, Honolulu, USA

/5/ T. Hagedorn, K. Kusterer, “Gekoppelte numerische Simulation von Strömung und Wärmeübergang zur thermischen Auslegung einer Brennkammer”, B&B-AGEMA report (in German)

Contribution of the RWTH (Institute of Steam and Gas Turbines, IDG) to WP 7

The objectives of WP7 are to analyse theoretically wall temperature and stresses and to validate numerical and experimental data in order to optimise thermal loads, to minimise the cooling flow to the walls and to allow a gas turbine design with high annual basis availability (> 95 %) and reliability (90 %).

The leader of this work package is B&B AGEMA doing the theoretical analysis of the thermal wall loading by using an advanced numerical method.

Nevertheless experimental work has to be done in task 7.2 by IDG to verify the wall-cooling model. Therefore a small-scale 2-D test section was set up as shown in Fig. 7.9

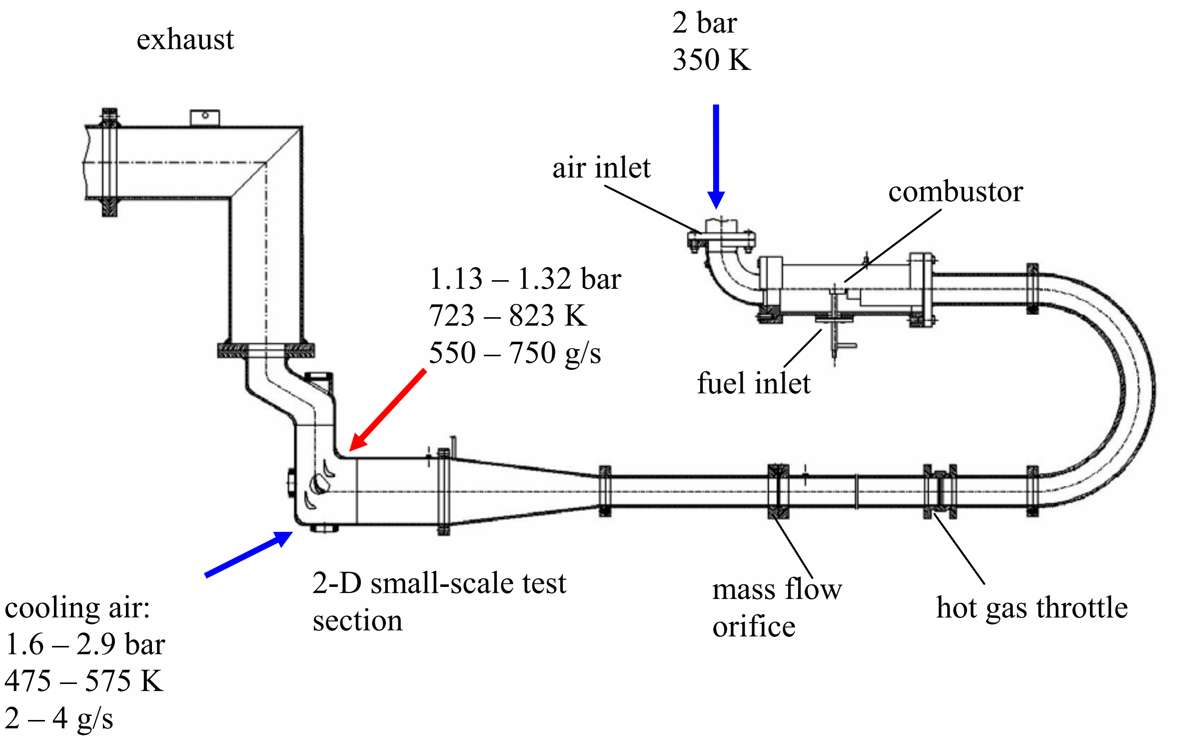


Fig 7.9 Hot-Gas Test Section

Test Rig Setup

Two turbo compressors provide the test rig with air at a pressure of 2 bar and a temperature of about 350 K. The air is heated by an oil fired combustor up to temperatures of 723 and 823 K. Downstream of the combustor, the pressure level of the gas is controlled by a hot gas throttle and the mass flow is determined by pressure and temperature measurements before and behind an orifice according to DIN 1952 /1/ (German Industry Standard).

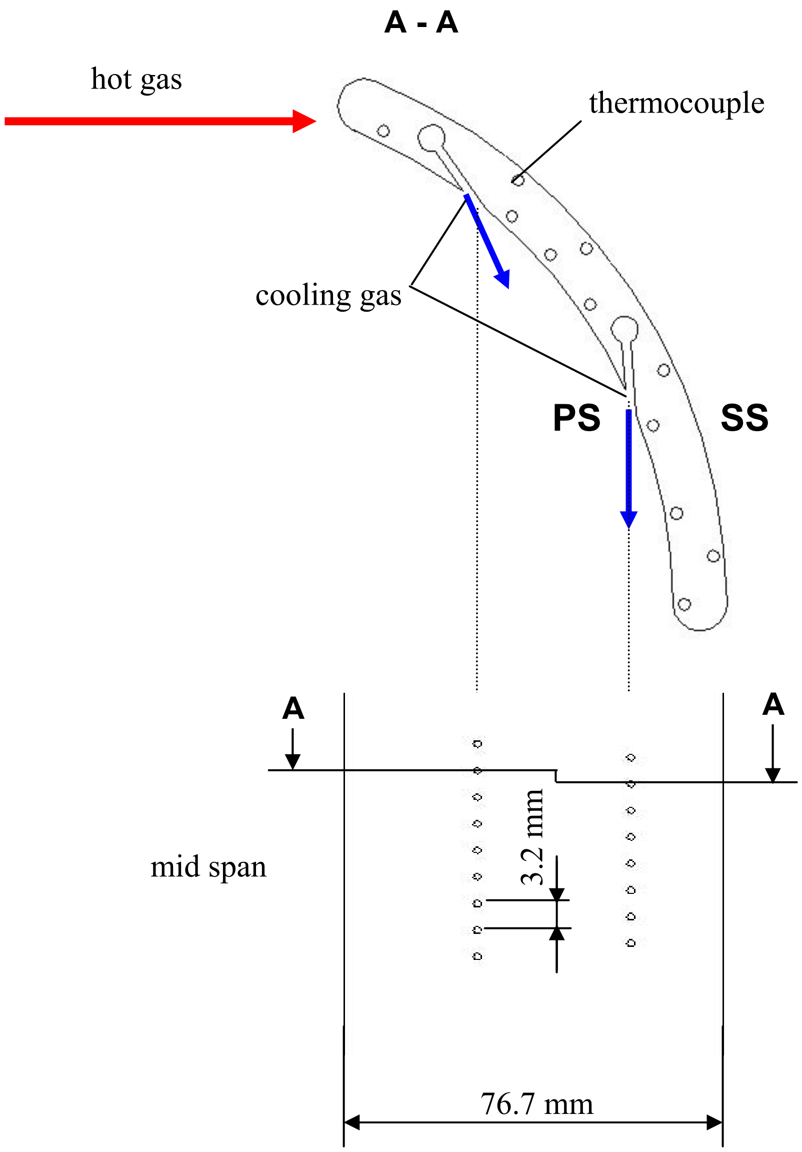


Fig 7.10 Cooled wall section

The hot air enters the test section with a pressure of 1.13 to 1.32 bar, a temperature of 723 K and 823 K and a mass flow of 550 g/s resp. 750 g/s. Because of pressure losses inside the test section the pressure between the test section and the exhaust decrease to values between 1.10

and 1.26 bar. The cooling air is delivered to the test section by an third compressor with a pressure between 1.6 and 2.9 bar, a temperature between 475 and 575 K and a mass flow of 2, 3 and 4 g/s.

The geometry of the investigated cooled wall section is shown in Fig . The cooling air mass flow enters the test section through two holes with a diameter of 2.5 mm and is divided into the cooling jets by 17 holes of 0.8 mm diameter each in two rows. Each row is shifted about half-length and the distance between two cooling holes is 3.2 mm.

Measurement Setup

The measurement technique at the test rig can be classified into three parts.

In the first part the mass flow of the hot gas and the cooling air was determined by using orifices according to DIN 1952 /1/. Therefore the pressures in front of and behind the orifice as well as the temperatures of the hot gas and the cooling air were measured. The mass flow was calculated in the data acquisition and data reduction system using an iterative code.

The second part were the temperature and pressure measurements in front of and behind the test section, which gave together with the mass flow the boundary conditions for the numerical simulation of B&B AGEMA.

Third the temperatures inside the cooled wall section were measured by 11 thermocouples, 4 of them on the outer or “suction” side (SS) and 7 at the inner or “pressure” side (PS) of the cooled wall section.

All signals of the sensors were digitalised by an analogue to digital converter and evaluated by the data reduction system. The complete system can be seen in Fig . In addition to the measurement system a complex safety system for the combustor was installed to protect against deflagrations and damages inside the test rig.

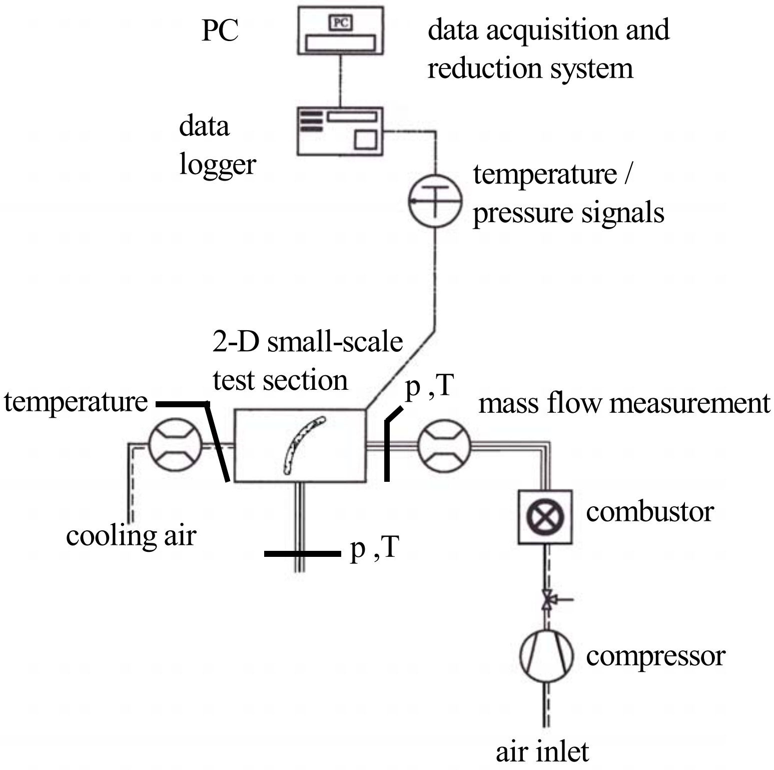


Fig 7.11 Measurement and data acquisition system

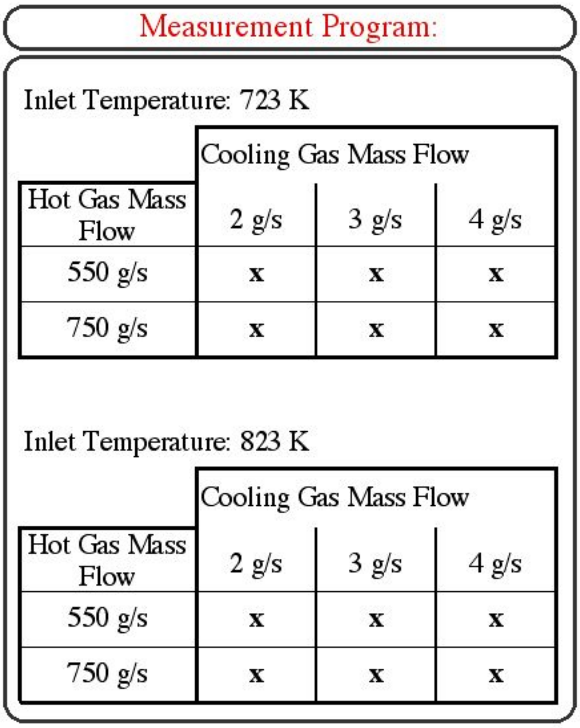
Experimental Results

In cooperation with the B&B AGEMA the following operation for the investigation were defined, see Fig 7.12:

The measurements were made for two different hot gas temperatures: 723 K and 823 K, two different hot gas flows: 550 g/s and 750 g/s and for three different cooling gas mass flows: 2 g/s, 3 g/s and 4 g/s.

The results of the measurements are shown in Fig - Fig .

In each diagram the temperatures of the thermocouples are shown in dependence on the relative length L/L_{Max} and the cooling gas flow. The relative length is defined as the quotient of the developed distance between the thermocouple position and the leading edge of the wall L in relation to the developed total length of the wall section L_{Max}.



Measurement Program:

Inlet Temperature: 723 K

Hot Gas Mass Flow	Cooling Gas Mass Flow 2 g/s	3 g/s	4 g/s
550 g/s	x	x	x
750 g/s	x	x	x

Inlet Temperature: 823 K

Hot Gas Mass Flow	Cooling Gas Mass Flow 2 g/s	3 g/s	4 g/s
550 g/s	x	x	x
750 g/s	x	x	x

Fig 7.12 Measurement program

The measurement results show that the temperatures depend mostly on the axial position. Compared with this the influence of the different flow and pressure field on the outer or inner side of the wall section on the temperature is negligible. The characteristic of the temperature curves are very similar for all 12-operation points. Between the first and the second row of cooling holes the temperature drops by approx. 50 K.

A further drop behind the second row of cooling gas holes, which could be expected, was not observed. The level of the temperature characteristic depends on the hot gas temperature. Furthermore the level of temperature inside the wall decreases with increasing cooling gas mass flow. In addition, what has been expected too, the level increases with increasing hot gas mass flow. The difference in the wall temperature levels between similar operation points at different hot gas temperatures (approx. 85-90 K) is slightly lower than the difference of the hot gas temperatures itself (100 K), because the cooling air temperature is not increased as well.

Summarising, the results are a good base to validate the numerical results of the project partner B&B AGEMA.

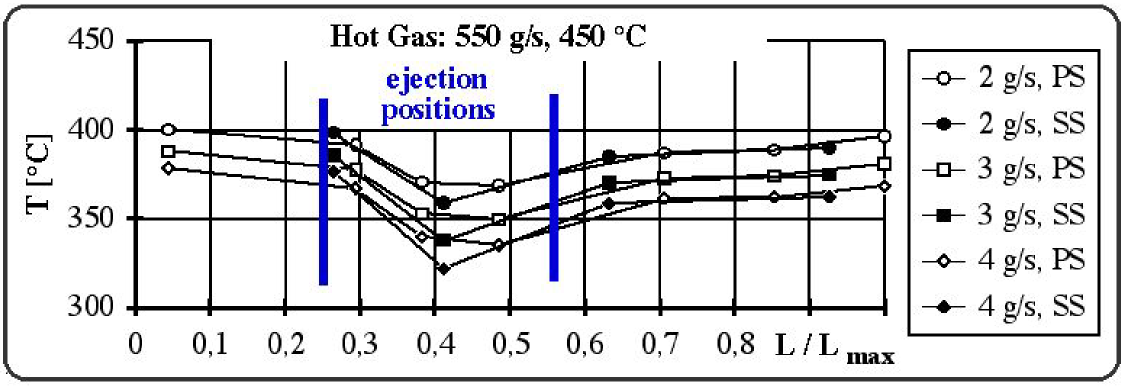


Fig 7.13 Temperatures inside the cooled wall section for hot gas conditions of 550 g/s and 450°C

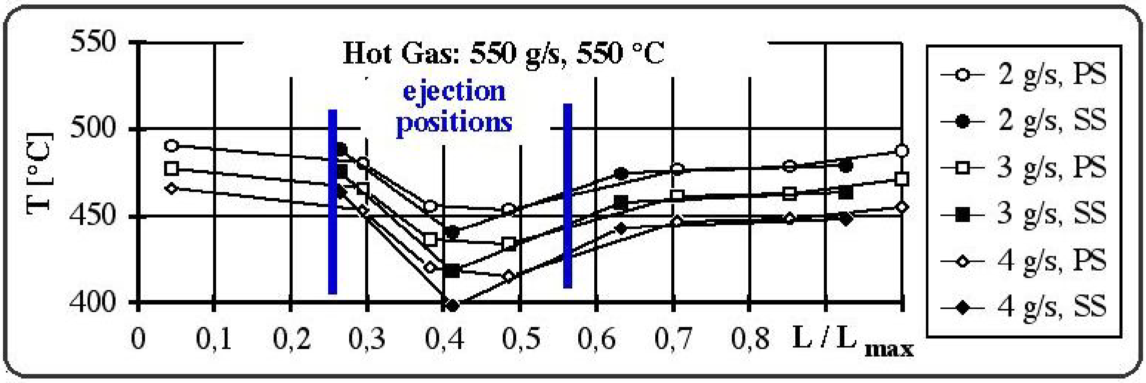


Fig 7.14 Temperatures inside the cooled wall section for hot gas conditions of 550 g/s and 550°C

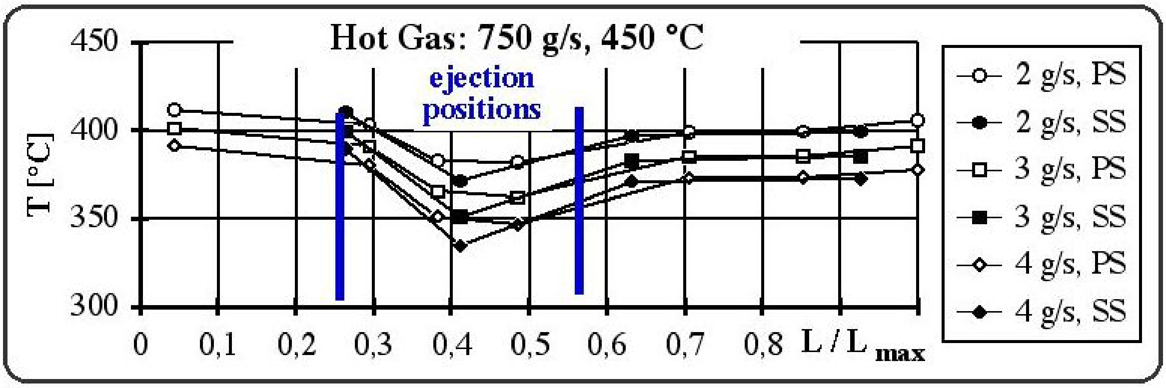


Fig 7.15 Temperatures inside the cooled wall section for hot gas conditions of 750 g/s and 450°C

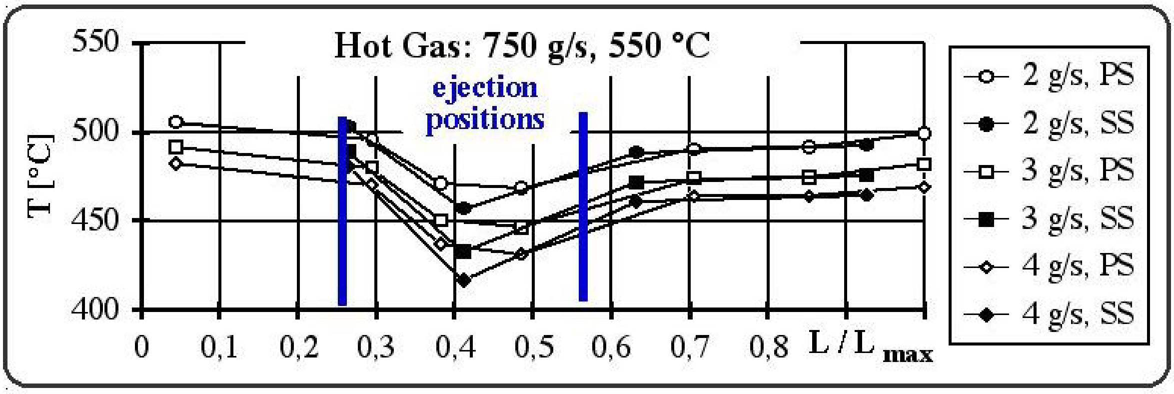


Fig 7.16 Temperatures inside the cooled wall section for hot gas conditions of 750 g/s and 550°C

2.2.8 WP8 PILOT COMBUSTOR TESTING

WP Leader:	RWTH
Additional Participant:	B&B AGEMA

2.2.8.1 Project Overview

The FLOXCOM project was intended to develop a technology for clean and efficient gas turbines. It was based on a technologically innovative combustion solution – Flameless Oxidation. The overall objective of the project was the development of an operating pilot combustor to demonstrate an improved performance relating to low NO_x levels, to maintain uniform combustor wall temperatures and increase “Mean Time Between Failure” (MTBF) and reliability of gas turbines.

In work package 8 a full-scaled pilot combustor should be designed and tested under realistic temperature and pressure conditions at a test facility of the IDG. Two radial compressors operated by IDG were provided for air supply. A lot of effort was done by IDG to take part in the follow-up of the development of the combustor in close cooperation to project partner Technion, who was responsible for the design of the pilot combustor. On the basis of drafts of the combustor IDG started to design connectors for air inlet and exhaust gas into the chimney. A safety concept with flame detection, fast closing valve, rupture disc and gas warning system was developed. The interfaces for flame detection and observation by boroscope were also done in close cooperation to Technion. In addition, different measurement techniques to control the operation status, to measure the pattern factor and a gas analysing system were set up. The test facility was equipped with a high accuracy real time data acquisition system.

A liquid fuel supply system had been prepared to provide the test rig with light fuel oil or kerosene when the decision was made on the request of Technion to use natural gas instead of liquid fuels for a more homogeneous mixing. Because it was not possible to provide the pressurised combustor with the institute’s internal natural gas pipe system, a mobile tube trailer was adopted to realise the fuel supply.

The design and the operation of the pressurised combustor must be done according to the German regulations, which means that high quality materials have to be used and high safety standards are prescribed. Additionally the pressurised combustor needs an authorisation by the German TÜV. During the project it turned out, that the abidance of these terms would have caused an unconsidered rise of the costs. Due to the limited funding, Technion had no other choice than to change the experimental parameters and to run the combustor under atmospheric conditions.

Due to the modifications of the new design of the atmospheric combustor a complete redesign of all pipe-connecting systems in the test field was necessary. With the reduced air pressure a preheater had to be used to raise the inlet temperature up to 140°C as foreseen for the pressurised combustion. In consequence to the reduced air mass flow the fuel mass flow was also reduced, to keep the air/fuel ratio constant. Hence, instead of the tube trailer a natural gas bottle package was adopted for fuel supply.

The safety concept was revised according to the change of parameters and the components were integrated in the test facility. Also the measurement techniques were adjusted to the new parameters.

The changes caused a delay in the project and the combustor was not delivered up to the official end of the FLOXCOM project. Nevertheless IDG worked out a plan specifying the test conditions and operating points for later investigations. As soon as the combustor arrives the experimental investigations will be accomplished and an addendum report will be send to the EU.

Due to the change from a pressurised to an atmospheric combustor the primary objective was not completely fulfilled, hence on the basis of the exploitation of the expected results of the investigations in the FLOXCOM project a follow-on project, regarding the design and operation of a pressurised combustor for further experimental investigations under realistic conditions, is recommended

2.2.8.2 Main Objectives

The following technical and scientific tasks were done by IDG:

1. to design the interface between combustor and the test facility,
2. to follow-up pilot combustor design, construction and preliminary testing,
3. to perform extensive pilot combustor testing and
4. to deduce conclusions from comparison between predictions and tests.

According to these points IDG took part in the follow-up of the pilot combustor design in close coordination to the project partner Technion. IDG developed and designed the instrumentation and mounting of the pressurised 360° combustor.

Furthermore a safety concept with flame detection, fast closing valve, rupture disc, gas detector, and gas warning system was developed and additionally a data acquisition unit was set up. All necessary connecting pipes were designed and implemented into the test field.

The maximum operating conditions for the pilot combustor were submitted to Technion at the beginning of the design process. The design parameters were set as follows:

Operation inlet pressure:	5.5 bar (absolute)
Inlet temperature:	75°C ≤ T ≤ 140°C
Operation outlet pressure:	4.5 bar (absolute)
Design outlet temperature at the exhaust:	500°C
Operation outlet temperature at the exhaust:	400°C

In the proposal a well considered concept of complementary tests of the partners was developed and submitted to the EU, starting with experiments at a 60° sector combustor at atmospheric pressure (IST), continuing with investigations at a 60° sector combustor under pressurised conditions (Ansaldo) and coming up to the final stage by a full scale testing under pressurised conditions (IDG) (see
Table 1). The latter was planned to show the functioning of the combustor under conditions as close to real engine conditions as possible.

Due to the upcoming costs of controlling and certification of the pilot combustor during the production, the financial situation of the project partner Technion leads to a change from pressurised combustion to atmospheric combustion (see chapter 0).

Table 1: Experimental Overview

Combustor Angle [°]	Pressure	Temperature	Institute
60	ambient pressure p_a	hot	IST (P)
60	$p > p_a$	hot	Ansaldo (I)
360	$p > p_a$	hot	IDG (D)

Until the end of the project the combustor was not delivered. After the arrive of the combustor the whole planned test program will be accomplished and the results will be analysed as in chapter 0 is described. An addendum report will be submitted to the EU as soon as the measurements are finished.

Nomenclature

Symbol	Description	SI-Unit
A	area	[m^2]
c	velocity	[m/s]
c	specific heat capacity	[kJ/kgK]
$\bar{c}$	averaged specific heat capacity	[kJ/kgK]
D	diameter	[m]
f	frequency	[Hz]
h	height	[m]
L	length	[m]
m	mass	[kg]
$\dot{m}$	mass flow	[kg/s]
P	power	[W]
p	pressure	[bar]
$\dot{q}$	specific heat flux	[kJ/kgs]
r	radius	[m]
R	ideal gas constant	[J/kgK]
t	time	[s]
T	temperature	[K]
Δt	time interval	[s]
$\dot{V}_{in}$	volume inlet flow	[m^3/s]
$\dot{V}_{out}$	volume outlet flow	[m^3/s]

Greek Letters

Symbol	Description	SI-Unit
α	flow rate number	[-]
Δ	difference	[-]
ε	expansion number	[-]
ρ	density of the fluid	[kg/m^3]
ν	cinematic viscosity	[m^2/s]
π	pressure ratio	[-]
π	circle number	[-]
Φ	equivalence ratio	[-]

Shortcuts

Symbol	Description
(DG-RL)	Pressure Equipment Standard (Germany)
DIN	German Industry Norm
DN	nominal diameter
eff.	effective
F	fluid
G	gas
id	ideal
HP	high pressure

LP	low pressure
Max	maximum
Min	minimum
PN	nominal pressure
PS	pressure side
SS	suction side
th	theoretical
tot	total
TÜV	Safety Standards Authority (Germany)

Indices

Symbol	**Description**
b	boil
e	exit
eoe	enthalpy of evaporation
el	electrical
G	gas
ges.	complete
in	inlet
out	outlet
m	mass
Max	Maximum
Min	Minimum
meas	measured
p	isobar
st	steam
v	vapor
W	water
0	coefficient number
1	coefficient number

INTRODUCTION

In the Kyoto Protocol the EU agreed to limit their greenhouse gas emissions, relative to the levels emitted in 1990, by 5.2 % until 2012.

A significant contribution can be done by lowering the emissions of gas turbines. The main objective of the FLOXCOM project is the development of an innovative combustor for gas turbines. The combustion technology developed within the FLOXCOM project will support this in achieving the following objectives:

- to obtain low NO_x emission levels (below 20 ppm)
- to generate electricity and heat with reduced CO_2 from gaseous or liquid fuels
- to improve the efficiency of gas turbines (to above 35 % for small gas turbines)
- to utilise lower grade fuel (with LHV lower by 25 % than that of natural gas)
- to retrofit existing power plants
- to reduce global and local environmental impact while reducing cost

The method proposed in the FLOXCOM project is based on a technologically innovative combustion solution, the FLamless OXidation (FLOX) method. FLOX is being currently used in industrial furnaces with regenerative burners in non-adiabatic cycles, indicating extremely low NO_x emission levels (typically less than 5 ppm).

The distinct advantage of this innovative design arises from the new ability to apply the FLOX combustion concept to gas turbine combustors. The FLOX combustor design results in lower NO_x emission levels in comparison to existing alternative dry-low NOx combustion techniques, running under similar operation conditions.

The technological objectives of the FLOXCOM project are to design, build and test a pilot combustor for gas turbines using the FLOX combustion concept. For the completion of these objectives basic studies in the field of combustion theory and design optimization have to be performed. In addition, due to the unique internal flow structure inside the combustor that is characterised by a large vortex, an improved wall cooling and fuel injection method can be integrated into its design.

To reach the scientific and technological aim, studies have to be done to improve combustion modeling according to engineering recommendations for enhanced mixing and combustion and emission performance of gas turbine combustors. This is to be achieved by improving the turbulence/chemistry interaction models developed for diffusion and partially premixed flames. These models will be incorporated in an existing three-dimensional computational fluid dynamics code (CFD).

Detailed measurements in a transparent combustor will be done to get physical insights into the main vortex inside the combustor. Also numerical and experimental investigations of the flow-field and combustion performance for different atomisation systems will be performed to improve fuel injection and distribution.

Further detailed investigations are to be done to develop optimised wall-cooling methods. Using common CFD codes for analysing and predicting the complete 3D-combustor performance will lead to an optimised combustor geometry and design and result in maximum combustion stability, uniformity of wall temperature, smooth profiles of the circumferential distribution of the exhaust gas temperature and minimum pollutant emissions.

For the validation of the CFD predictions and further improvement of this various models combustor sections will be produced with optical windows for detailed local measurements of velocity vectors, temperatures and species concentrations. After all a pilot combustor will be assembled and tested for global performance measurements. This validation stage is essential for further commercial exploitation. The deduced conclusions from comparison between prediction and tests lead to the manufacture of a FLOX combustor for a specific gas turbine and a test program for endurance testing.

The proposed investigation will be distributed between the different participants while guidance, coordination and integration will be performed by the coordinator.

Test Rig Setup for Investigations at the Pressurised 360° Combustor

For the experimental investigations of the 360° pilot combustor of the FLOXCOM project a special test facility is available at the Institute of Steam and Gas Turbines at RWTH Aachen University.

The block diagram (see Fig) shows the schematic setup of this test facility. The red covered area is the test rig itself as planned to be delivered by Technion. The ignition system (igniters and transformer), an exhaust cooler with water injection as well as part of the integrated measuring system inside of the combustor (16 thermocouples and pressure sensors) will also be delivered by Technion.

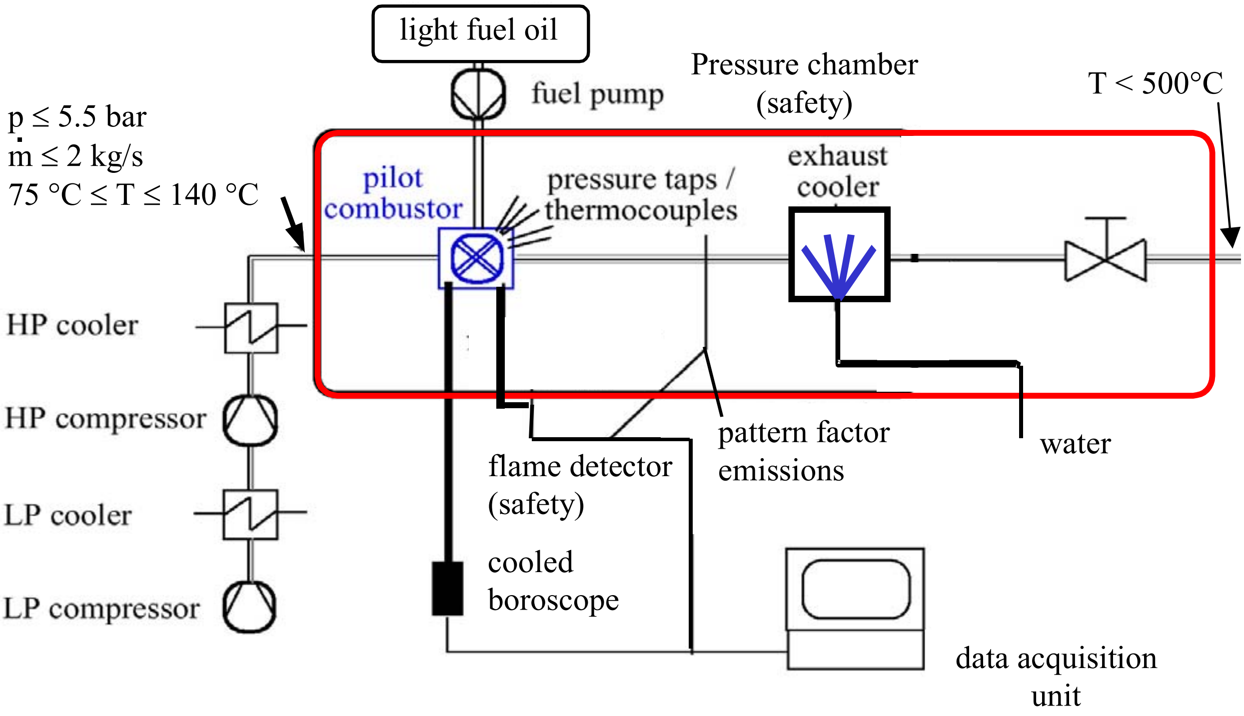


Fig 8.1 Block diagram of the test field under pressurised conditions

The test rig will be set up in a special test field, equipped with a separate control and safety cabin. A DN 300 pipe system supplies the test facility with pressurised air (up to 5.5 bar) from the compressors. Fig 8.2 shows the inlet pipe to the test rig (Fig 8.3) as it runs through the test field close to the ceiling. A Venturi nozzle is used to measure the mass flow of the inlet air as required in the German Industry Norm (DIN 1952). The connection of the exhaust gas pipe to the chimney is on the left hand side of the test field wall, near to the entrance, and has a diameter of 500 mm (DN 500).

For the liquid fuel supply of the combustor a system was prepared to provide the test rig with light fuel oil or kerosene with a mass flow of 500 g/s and a pressure of up to 20 bar. During the project the decision was made on the request of Technion to use natural gas for a better mixing in the combustor. That caused a complete new design of the test rig (see chapter 0).

Furthermore the water inlet and outlet was implemented in the test field to allow the cooling of some test rig parts, e.g. the exhaust pipe.

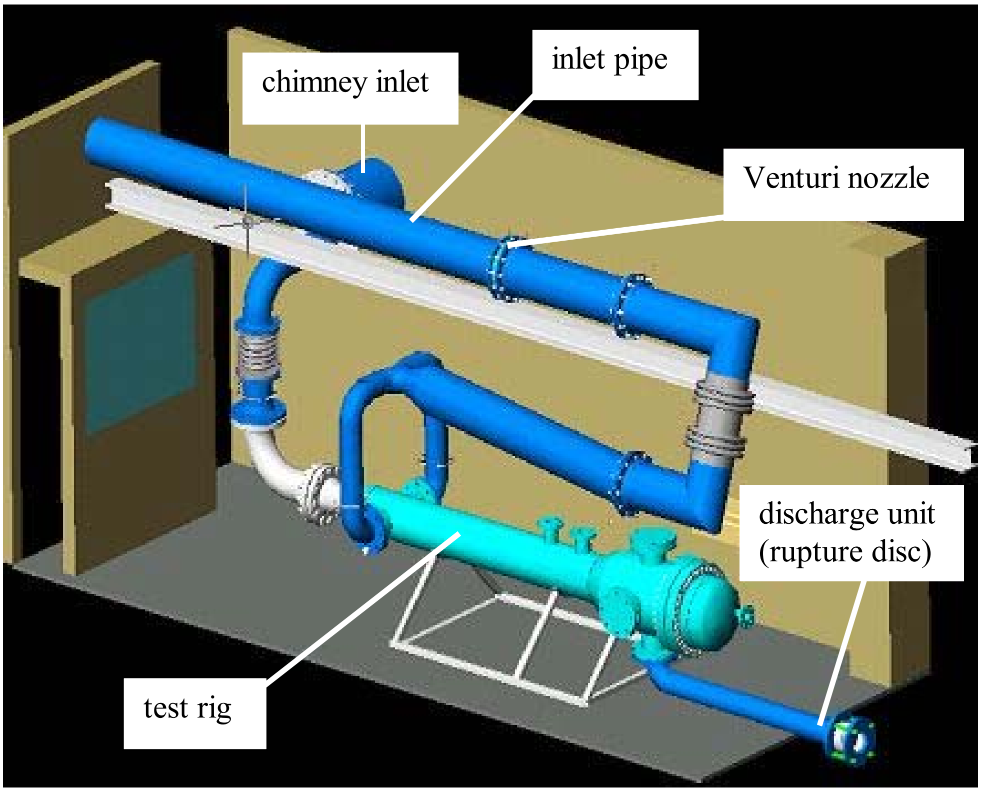


Fig 8.2 Test field

Design of the Test Rig Junctions

To connect the combustor to both, the air inlet pipe system and the exhaust gas system, special adapter fittings had to be designed and manufactured.

In the test field of the IDG, downstream from the Venturi nozzle, the air inlet pipe was new designed. An existing compensator with a connected sharp bend on each side could be integrated without any modifications. A reduction from DN 300 down to DN 250 was planned to integrate a fast closing valve to eliminate the possibility of a back running pressure wave in case of deflagration inside of the combustor.

To connect the air pipe with the two air inlet flanges of the combustor, each one arranged on one side, an integration of a y-branch was necessary. The y-branch was inserted with two 90° elbows and connected to the entrance of the combustor on both sides.

Behind the combustor all pipes were designed with high temperature resistance (up to 500°C) and for a pressure stage of PN 64. The exhaust flange of the combustor is connected with a pipe DN 200. Following is a lateral compensator that runs with a 90° elbow into the chimney. The chimney entrance has a DN 500 opening, blocked by a blind flange with an opening of DN 200 for the exhaust gas pipe.

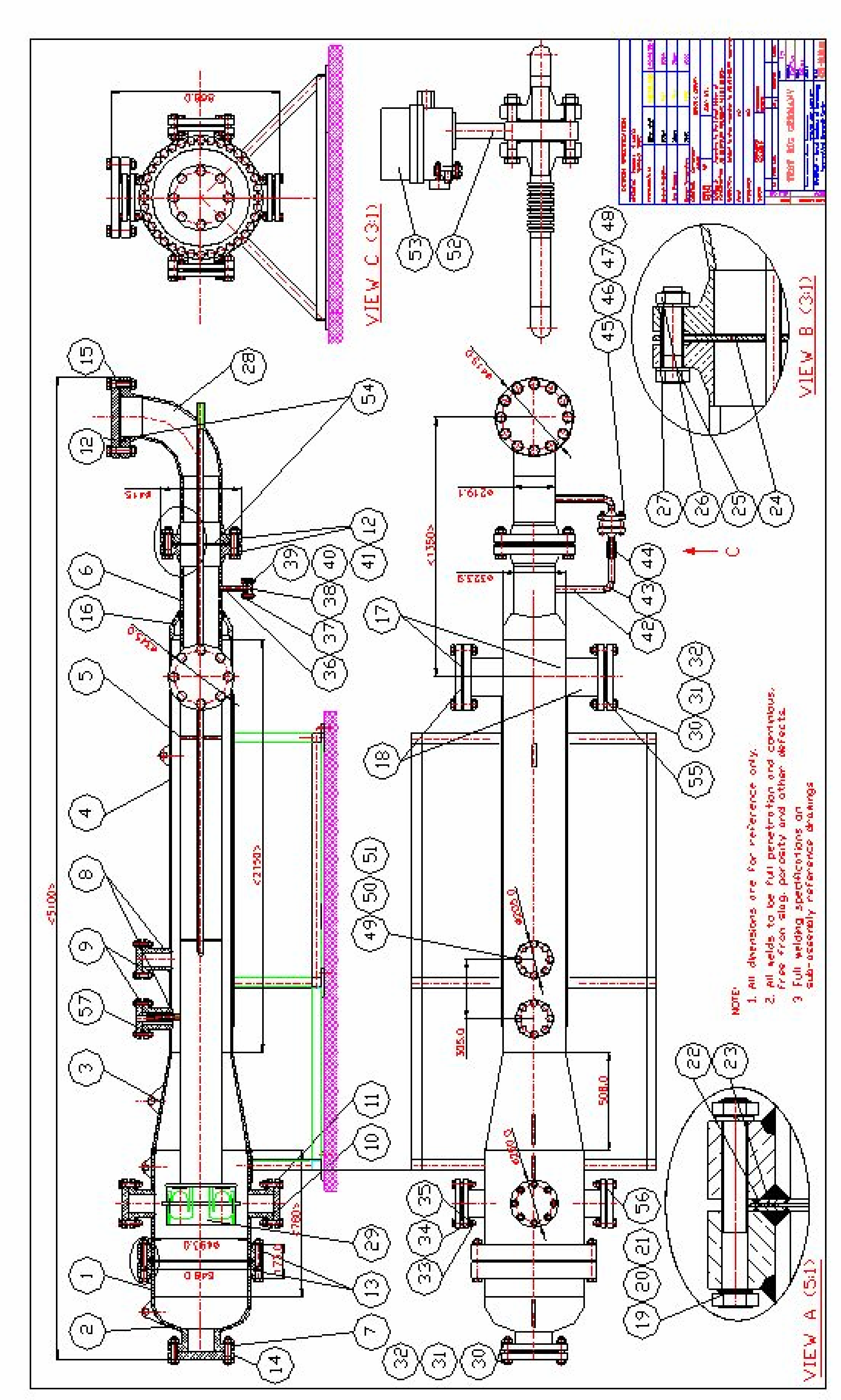


Fig 8.3 Draft of the pressurized Combustor by Technion

Air Supply

Two radial compressors operated by IDG were provided for air supply. These radial compressors are capable to deliver the operating mass flow of 2 kg/s at a pressure of 5.5 bar if they are operating in series. The air of the low pressure compressor is cooled down before entering the high pressure compressor. A list of compressor configuration can be seen in Table 2.

The required inlet temperature of 140°C in the combustor is reached only by compression of the inlet air.

Table 2: Configuration of two row-connected compressor

	Serial connection in an open cycle	
	LP	HP
Inlet volume flow [m^3/s]	4,8	2,08
Inlet pressure [bar]	1,0	2,5
Inlet temperature [K]	293	313
Pressure ratio π	2,6	2,38
Outlet pressure [bar]	2,6	5,95
Outlet temperature [K]	418	438
Mass flow [kg/s]	5,71	5,71

Fuel Supply

As the decision was made to use gas as fuel instead of oil or kerosene (see chapter 3.1), a gas supply system had to be built up delivering a maximum fuel mass flow of up to 50 g/s at at least 8 bar.

It was not possible to provide such a high mass flow with the internal natural gas pipe system of the institute. The alternative was the installation of a 6000-litre tank of natural gas. However, due to the buildings surrounding the institute, it was not possible to fulfill the TÜV requirements to keep space for safety corridors.

The solution of the problem was to place natural gas close to the institute in a mobile tube trailer. The trailer could remain in the parking place on the opposite side of the institute. The natural gas can be throttled down to 12 bar and afterwards heated in a (from the gas contractor provided) control station (see Fig) before entering the pipe system to the test facility. In the test facility itself there is a second regulation to reduce the pressure of the natural gas at the inlet of the combustor down to approximately 8 bar and also to regulate the mass flow. The trailer has a standard volume of approx. 3500 m³. With the planned one hour measuring a day under full load the trailer delivers gas over a period of 10 - 15 days.

Fig 8.4 Mass flow control station (2x2m)

Safety Engineering

Due to the safety risks during a pressurised combustion, a safety concept was created to prevent human injury and equipment damage and all potential incidents by using safety procedures.

The most dangerous incident, which could be expected, is a deflagration, which has a pressure rise of up to ten times the working pressure /2/. This was the reason for using a PN 64 pressure stage design for the whole test rig. Furthermore there was a control mechanism equipped, which shuts down the entire system in cause of a forbidden pressure rise.

The following parameters are not allowed to be exceeded in the test rig:

- Inlet pressure p_{in}: 6.8 bar
- Exhaust gas temperature of the test rig $T_{Exhaust}$: 500 °C

(at the entrance to the chimney)

- Wall temperature of the combustor T_{Wall}: 350 °C
- Explosive concentration of methane in the test field

(explosive boundary for air: 4,4 Vol.%) /3/

Fig shows two important components of the safety equipment:

- a flame detector and
- a rupture disc.

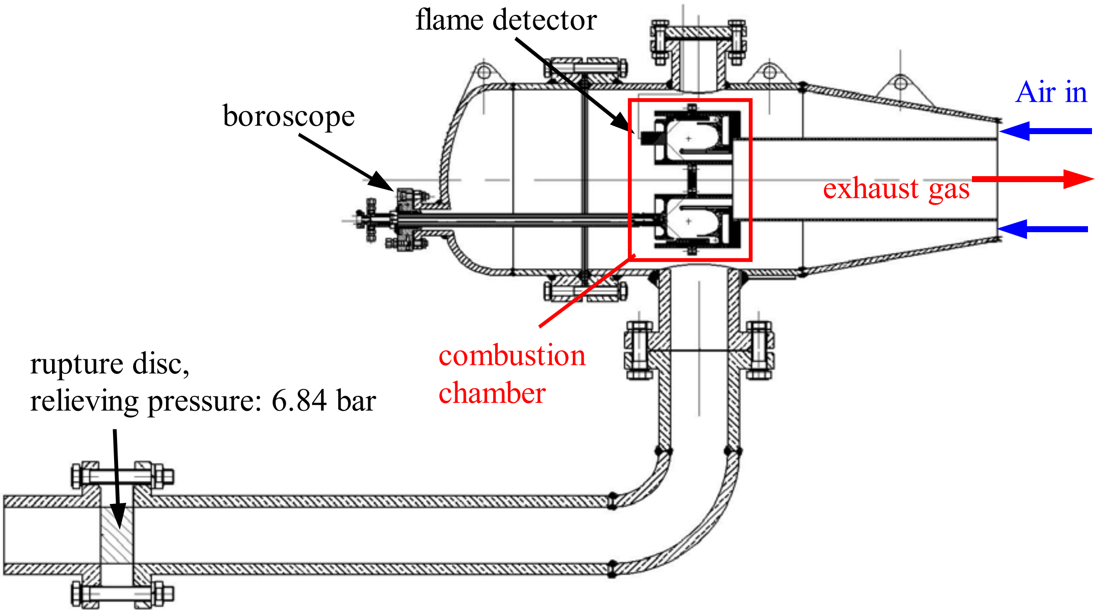


Fig 8.5 Safety concept

Further safety components were installed like:

- Fast closing valve, integrated in the inlet air pipe system
- Fast closing valve, integrated in the fuel service pipe
- Flashback protective valve for the gas pipe
- Independent control unit

Rupture Disc

To guarantee a riskless operation and to prevent damage of components, such as pipes, compressors and chimney, by an inadmissible pressure rise, a rupture disc was designed.

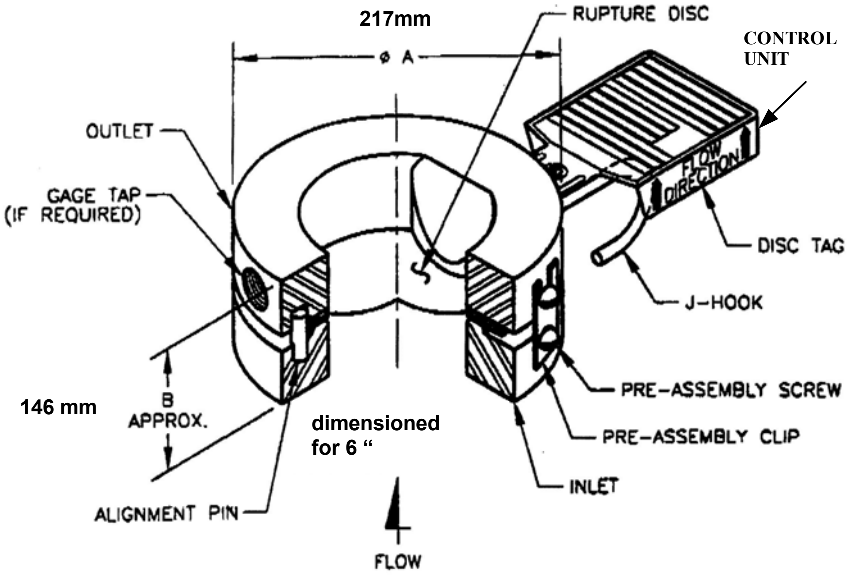


Fig 8.6 Rupture Disc

The opening gauge pressure for the rupture disc is 6.84 bar. In case of activating the rupture disc the pressurised gas is guided outside the test field and the pressure in the system will decrease.

Quick-action stop valve

It was planned to insert a quick-action stop valve into the air inlet pipe of the combustor to prevent - in case of a deflagration - the upstream propagation of a pressure wave and to protect consequently the compressor system. To guarantee the protection of the upstream pipes and the compressor system a closing time faster than 50 ms had to be realised. Regarding to the short closing time the pipe diameter of 300 mm was reduced to at least 250 mm. Additionally this caused a reduction of the costs.

A company was found being able to manufacture such a valve, which can be used several times without any destruction /4/. In case of a deflagration an explosion detector with pressure sensor near the combustor detects the pressure wave and activates the fast closing magnet system of the quick-action stop valve, which shuts it within a few milliseconds in order to protect the pipe system and upstream components against damage.

Fig 8.7 Quick-action valve

Gas Detector

Three gas detectors were integrated into the test field to prevent deflagrations or explosions caused by a leakage in the fuel system. In case of danger an optical and acoustical signal is alarming the test field staff. Simultaneously the fuel valve closes and the system shuts of.

Flame Detector

Another safety equipment in the combustor is the flame detector. The spectral range of the light emissions from the flame in the FLOX-mode is in the IR-light. So the light detection of the flame detector works in the IR-light spectral range between 550 nm-1100 nm /5/.

If the flame in the combustor extinguishes, the flame detector sends a signal to the SPS control unit, which shuts off the fuel valve of the fuel feeding immediately. So it is prevented, that unburned natural gas enters the combustor, and consequently a deflagration could not occur, due to the fact, that the still running continuous airflow lowers the methane concentration under the explosive boundary.

An important property of this flame detector is, that the flame reflections and glowing walls are not detected as flames. A glint frequency detector with a frequency of 20 Hz is guaranteeing that a wrong detection will not occur. After losing the operation signal caused by flame or in case of main failure (power loss), an immediate fault shutoff within 1 second is guaranteed. A risk of a flame detector is therefore minimised and no additional redundant flame detector is needed.

Fig 8.8 IR-flame detector

Measurement Setup

To detect the operating status of the combustor and to evaluate the quality of the combustion process the following parameters had to be measured and calculated respectively:

- Air mass flow
- Combustor inlet pressure

- Combustor inlet temperature
- Exhaust gas temperature
- Pressure in the exhaust pipe
- Fuel mass flow
- Fuel injection pressure
- Fuel/Air ratio
- Exhaust gas quality

In addition the combustion chamber will be observed with an optical system, the boroscope, to detect the FLOX-mode. The combustor itself will be equipped with temperature and pressure measurement technique from the project partner Technion.

For recording all data and operating the combustor a special acquisition and control unit will be used.

Fig 8.9 shows the positions for temperature, pressure, mass flow measurements and exhaust gas analysis.

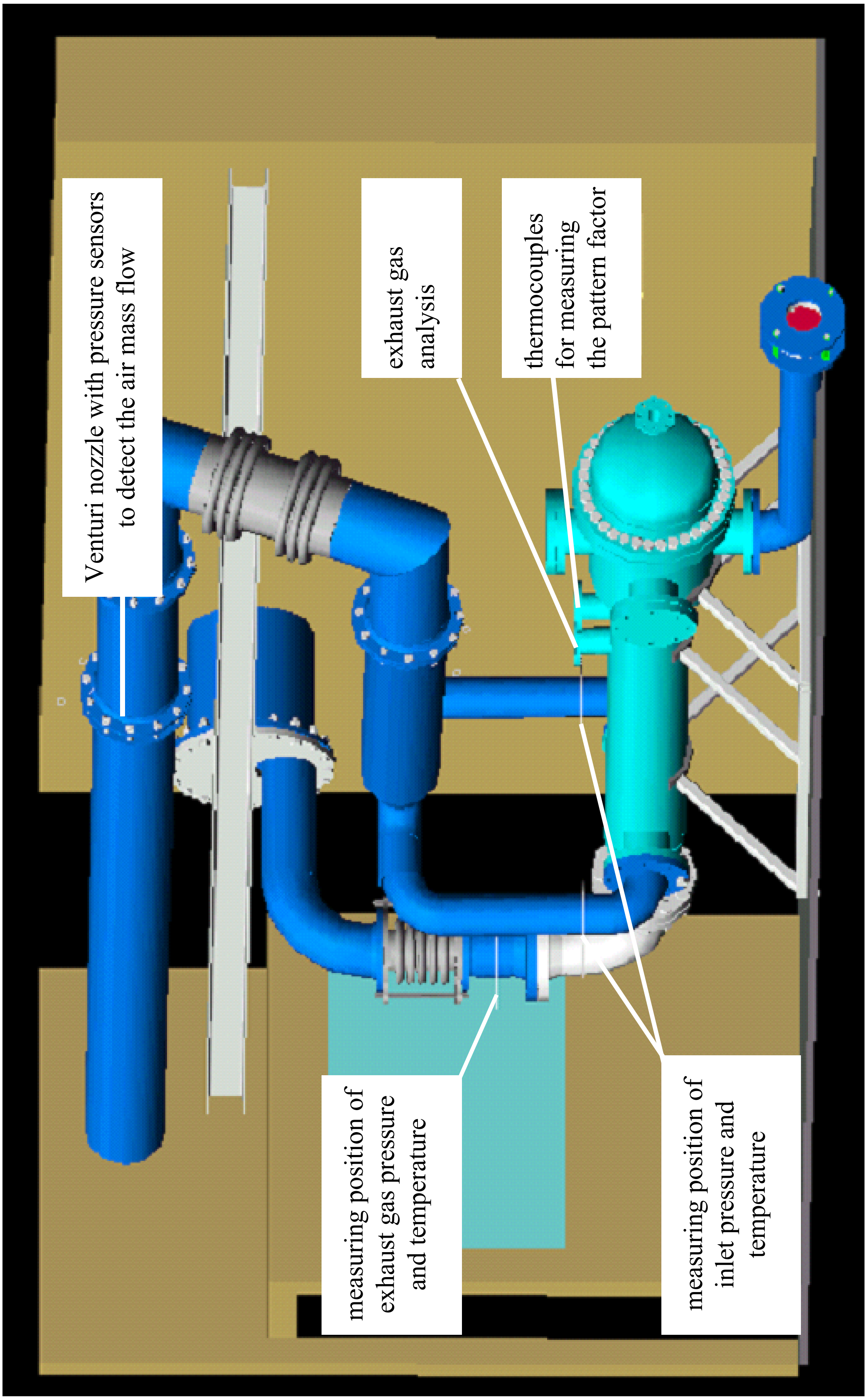


Fig 8.9 Combustor Test Rig

Air Mass Flow

To measure the air mass flow a calibrated Venturi nozzle was installed. The pressure differences of the static pressures at the entrance and in the throat of the nozzle are registered. The mass flow is calculated by DIN 1952 /1/ as follows:

$$\dot{m} = \alpha \cdot \varepsilon \cdot \frac{\pi}{4} \cdot d^2 \cdot \sqrt{2 \cdot \Delta p \cdot \rho} \qquad \text{Equation 1}$$

To determine the density ρ additional a thermocouple measured the temperature at the entrance of the nozzle.

Pressure Detection

The static pressure is an important parameter for the FLOXCOM process.

It will be measured by means of ring pipes, which are installed at the air inlet and the exhaust gas pipe. Each ring pipe has three wall pressure holes, circumferentially by 120°, in order to measure an average static pressure. The ring pipes are connected with rubber hoses to absolute pressure transducers.

A pressure box (see 8.10) with calibrated pressure transducers was set up to convert the pressures into voltage signals.

Fig 8.10 Pressure sensor box

Temperature Detection

NiCr-Ni thermocouples are positioned circumferentially (shifted by 120°) at the inlet- and the outlet of the test rig to measure the air and exhaust gas temperatures. This circumferential alignment allows to detect and to balance a temperature streak and in case of a thermocouple failure a malfunction can be recognised.

Directly behind the combustor a movable thermocouple is positioned to measure the distribution of the outlet temperature and therefore the pattern factor.

The maximum working temperature of the chimney is 500°C. Therefore three thermocouples control the temperature of the exhaust gas to protect the chimney against damage. The SPS safety control unit shuts off the fuel valve in case of exceeding the maximum outlet temperature caused by a temperature rise of the combustor or a failure in the exhaust pipe cooling.

Fuel Mass Flow

To determine the natural gas flow an oscillating flow meter was chosen (see Fig 8.11). It is a non-contact flow meter /6/ with an explosion-proof design integrated in the fuel inlet pipe.

Fig 8.11 Fuel flow meter

Exhaust Gas Analysis

Fig 8.12 Exhaust gas analyze system CO, CO_2, NOx, HC

An exhaust gas analyse system was set up for measuring CO, CO_2, NO_x and HC. All main exhaust gas analysers where installed to investigate the whole spectrum of exhaust gases to have a good opportunity to compare the results with other combustor systems. Although the most important is the detection of the NO_x. In Fig 8.12 one can see, one of the two exhaust gas measuring cabinets is shown to analyse the exhaust gas in the area behind the combustor and in front of the water-cooling system.

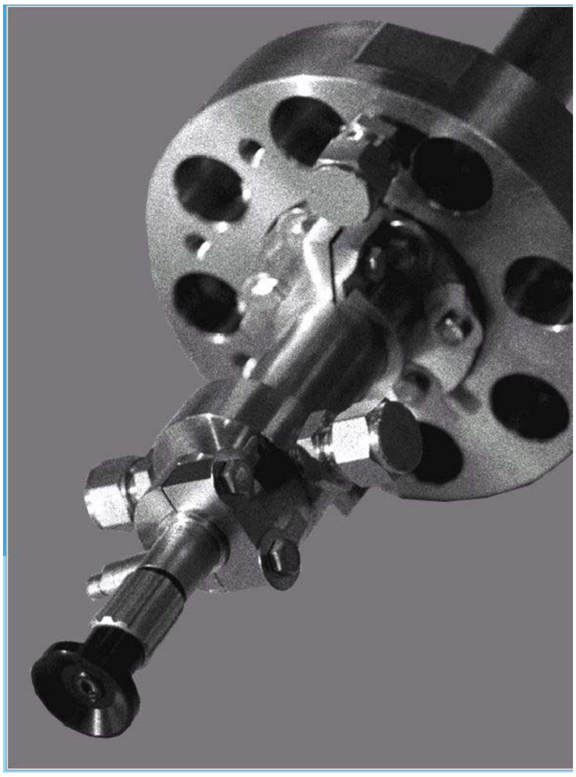


Fig 8.13 Boroscope

Boroscope (Fig 8.13)

For the optical monitoring of the combustion inside the combustion chamber an endoscope will be used. This type is a rigid 4 mm diameter endoscope with a view angle of 0° straight ahead and an aperture angle of 90°.

The working temperature is limited by the manufacturer up to 120°C. A cooling system, developed by IDG, was applied to keep the endoscope temperature below 120°C using cooling air. The boroscope itself is mounted in a cooling pipe. Air channels inside the boroscope transfer heat out of high temperature sensitive areas. In the front wall of the cooling pipe a sapphire glass is positioned. The cooling air is reducing the temperature at the hot spots and prevents the endoscope lenses from melting. This system, endoscope with cooling system is called boroscope.

It was specially designed and built for the FLOXCOM-project.

To record the pictures from the combustor a CCD-colour camera with 1/3“ CCD-picture sensor was bought. The camera can be connected with the endoscope by using a quick release coupling.

The CCD-color camera has a PAL output signal. A tv-pci card in the computer is triggering the output signal and recording the pictures.

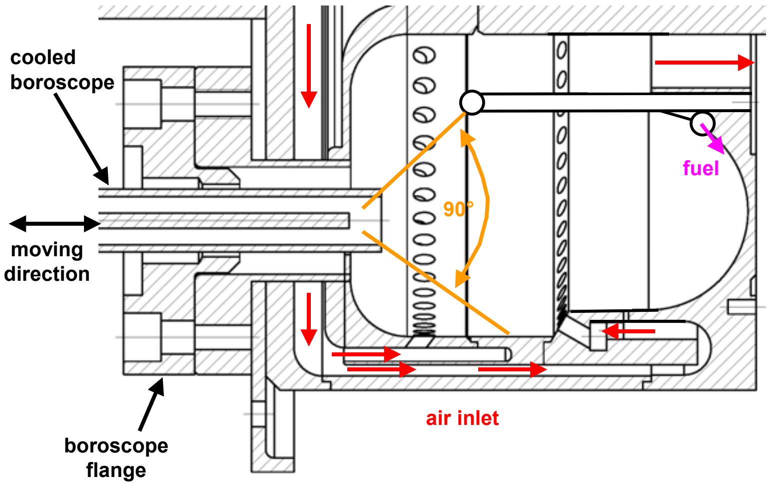


Fig 8.14 Cross cut of combustor with integrated boroscope

Data Acquisition Unit

The data acquisition for temperature and pressure data is done by a real-time data acquisition unit from National Instruments with 64 analog inputs at high accuracy.

This system also records simultaneously additional information of the exhaust gas analysis system.

A personal computer is processing and monitoring the incoming data in a LABVIEW-program.

Fig 8.15 Data acquisition unit

The Change of the Technical Status

At the beginning the project partner TECHNION designed a first draft version of a pressurised pilot combustor. This version was reviewed and the gross dimensions and the procedure to install it in the IDG combustor test field were agreed.

For this first draft IDG developed and constructed the complete installation for this combustor. Furthermore a safety concept with flame detection, rupture disc, gas detector, fast closing valve and gas warning system was developed.

Deviating of the project proposal, where atomisation and vaporizations of liquid fuels was planned, project leader TECHNION proposed to use methane as fuel. IDG investigated the feasibility of the use of gaseous fuels and according to the boundary conditions of the institute's test facility a solution for this change was worked out.

Another program change was made by Technion due to the limit of financial resources: The pressurised combustor must be manufactured and operate according to German regulations. This means the use of high quality materials and high safety standards. Furthermore the manufacturing process has to be controlled continuously by an authorised expert of the German TÜV. These facts lead to a rise of costs. To avoid this Technion decided to reduce the pressurised combustion to an atmospheric combustion. This modification caused also a change of operating parameters and design.

The pressure drop from 5.5 bar down to 1 bar will reduce the mass flow from 2 kg/s to 0.4 kg/s and the fuel mass flow from 50 g/s down to 10 g/s. The inlet temperature will still kept to a range between 75.0°C and 140.0°C.

Test Rig Setup for Investigations at the atmospheric 360° Combustor

A pressurised combustor has to be handled in Germany according to the German guidelines for pressurised apparatuses ("Druckgeräterichtlinie"). These are fixed standards. That means the manufacturing process is continuously controlled by an authorised expert and all required materials must be chosen according to these regulations/guidelines.
Due to these standards, high cost are caused by:

- Material choice X5CrNiMo 17 12 2 (1.4401), X2CrNiMo 17 13 2 (1.4404)
- Use of safety technique
- Authorisation by an expert and TÜV-acceptance

The finance resources of Technion for the design and manufacture of the combustor were limited in the project. Therefore, the project coordinator decided to design the combustor for the operation under atmospheric conditions. This change effected, that the rules for pressurised bodies were no longer of any importance for the combustor design and thus the costs could be kept in a sustainable limit.
Technion set the new parameters as follows:

Operation pressure: 1 bar (absolute)
Design inlet pressure: 10 bar (absolute)
Inlet temperature: 75°C $\leq$ T $\leq$ 140°C
Operation outlet pressure: 1.0 bar (absolute)
Design outlet pressure: 10 bar (absolute)
Operation outlet temperature: 400°C
Design outlet temperature: 500°C
Air mass flow reduced to 400 g/s

The modifications at the test rig and additional work for the IDG caused by this project change is described in the following paragraphs.

Design of Test Rig Junctions

The new atmospheric combustor concept caused a complete redesign of all pipe-connecting systems in the test field. The design pressure level for the pipe system was reduced to a pressure level of 10 bar because of the atmospheric combustion. Technion reduced the diameter of the air inlet pipe from DN 150 down to DN 80 at the inlet of the combustor. In case of deflagration during the start up, running or shut off phase, a sudden pressure rise -up to 10 times of the operation pressure- can happen.

No parameter change was made at the exhaust pipe system.

Air Supply

Because of the reduced pressure ratio the compressors provide air with an inlet temperature at surrounding conditions. Therefore an air preheater was integrated into the air inlet pipe system (see Figure) to raise the temperature up to 140°C as foreseen in the pressurised combustor. The preheater has 45 heating elements installed with a maximum electrical power of 120 kW.

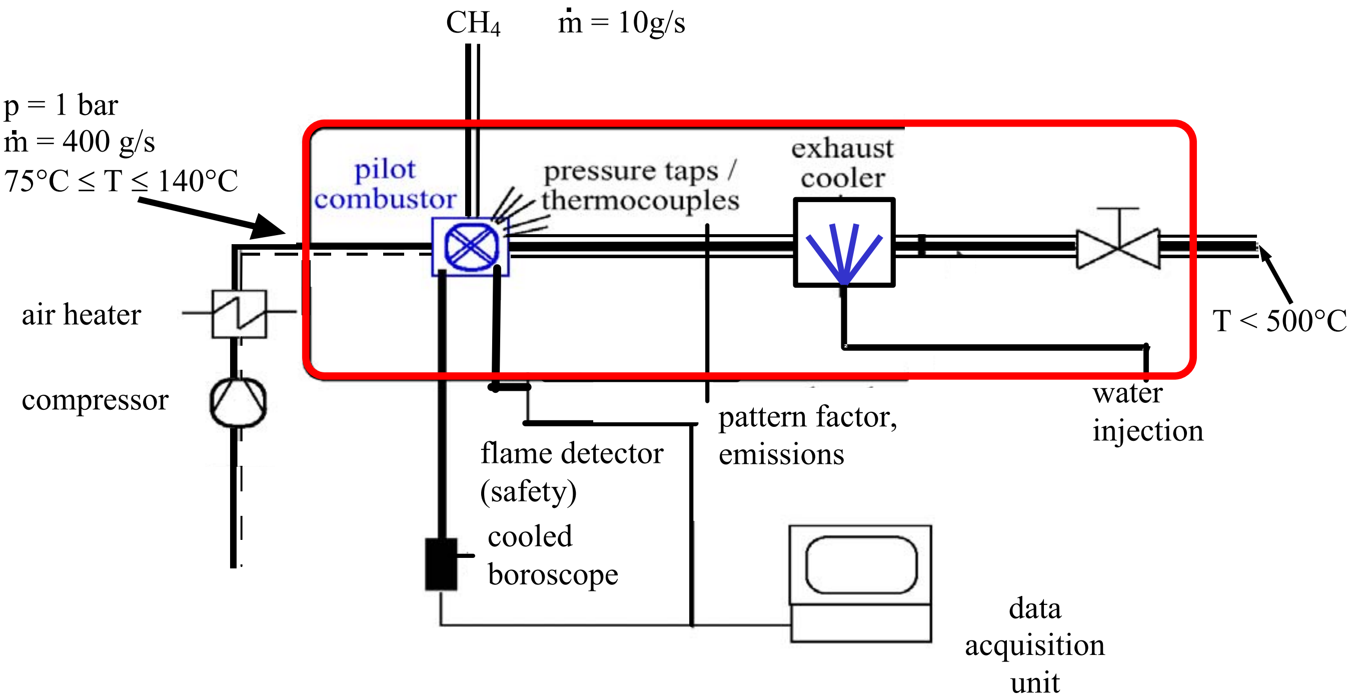


Figure 8.16: Block diagram of the test field

Fuel Supply

With atmospheric combustion the fuel mass flow was reduced to a fifth (approx. the same factor as the pressure) of the mass flow rate of the pressurised combustor to keep the air/fuel ratio constant. Therefore the tube trailer which had been designated to serve as a large gas reservoir was not needed any more. Instead of the tube trailer natural gas bottle packages will be used. One package consists of twelve bottles, each containing 50 liters and delivering 10m^3 gas, connected to each other by gas pipes. For a volume flow of 54 m^3/h one package provides gas for approx. 2 hours.

The installation of a gas compressor would be the alternative to rise the pressure level in the gas pipe system at the institute from 1.5 bar up to the required pressure. This option was dropped with regard to the high costs.

Water Cooling

The exhaust gas has to be cooled down to 500°C to prevent the chimney from superheating. To cool the exhaust gas in the DN 200 pipe straight behind the combustor a water injector is integrated into the pipe (see Fig). Nozzles are injecting water into the exhaust pipe. The amount of non-evaporated and condensed water is unknown so far, so that a drainage with a water pump will be installed to prevent a water level rise in the exhaust pipe.

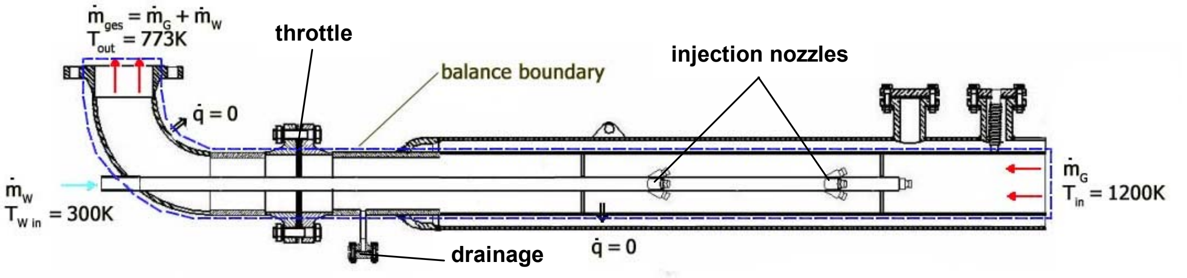


Fig 8.17 Energy balance for the exhaust pipe

For determining the water mass flow which will be needed for cooling down the exhaust gas from the combustor to the permissible value at the chimney entrance, a calculation was performed on the basis of the following operation conditions:

Estimated combustion exit temperature:	1200 K
Gas mass flow rate:	10 g/s

Gas outlet temperature:	773 K
Water inlet temperature:	300 K
Exhaust gas pressure	1 bar

Energy balance

All parameters used for the atmospheric energy balance are taken from the VDI Wärmeatlas /7/

$$\dot{H}_{\mathrm{Air,\,in}} + \dot{H}_{\mathrm{Water,\,in}} = \dot{H}_{\mathrm{Air,\,out}} + \dot{H}_{\mathrm{Water,\,out}} + \dot{H}_{\mathrm{eoe}}$$

Equation 2

$$\dot{m}_{G}\bar{c}_{p,\,G}\left(T_{in} - T_{out}\right) - \dot{m}_{W}\bar{c}_{p,\,W}\left(T_{b} - T_{W,in}\right) - \dot{m}_{W}\bar{c}_{p,\,v}\left(T_{out} - T_{b}\right) - \dot{m}_{W}h_{eoe} = 0$$

Equation 3

$$\dot{m}_{W} = \frac{\dot{m}_{G}\bar{c}_{p,\,G}\left(T_{in} - T_{out}\right)}{\bar{c}_{p,\,W}\left(T_{b} - T_{W,in}\right) + \bar{c}_{p,\,v}\left(T_{out} - T_{b}\right) + h_{eoe}}$$

Equation 4

$$\dot{m}_{W} = \frac{0.41\mathrm{kg/s} \cdot 1185\mathrm{J/kgK} \cdot \left(1200\mathrm{K} - 773\mathrm{K}\right)}{2256.5 \cdot 10^{3}\,\mathrm{J/kg} + 4199\mathrm{J/kgK} \cdot \left(373\mathrm{K} - 293\mathrm{K}\right) + 2082\mathrm{J/kgK} \cdot \left(773\mathrm{K} - 373\mathrm{K}\right)} = 0.06\ {}^{\mathrm{kg}}\!/_{\mathrm{s}}$$

Equation 5

So a cooling water mass flow of 0.06 kg/s will be required for the atmospheric combustion.

Safety Engineering

For atmospheric combustion, the German technical regulations of the high safety standards do not instruct the use of the following safety equipment: High pressure pipes, thick combustor walls, rupture disc and fast closing valve.

The new construction of the combustion chamber with a sandwich design (the combustor is constructed out of many overlapping combustor parts, hold together with tie-rods, see Fig 8.) was chosen by Technion to reduce the risk due to deflagration. This sandwich construction acts like a rupture disc and should prevent the pressure rise of deflagration by opening itself through each layer. This means that in case of a combustor burst, hot gas and flames enter the test field.

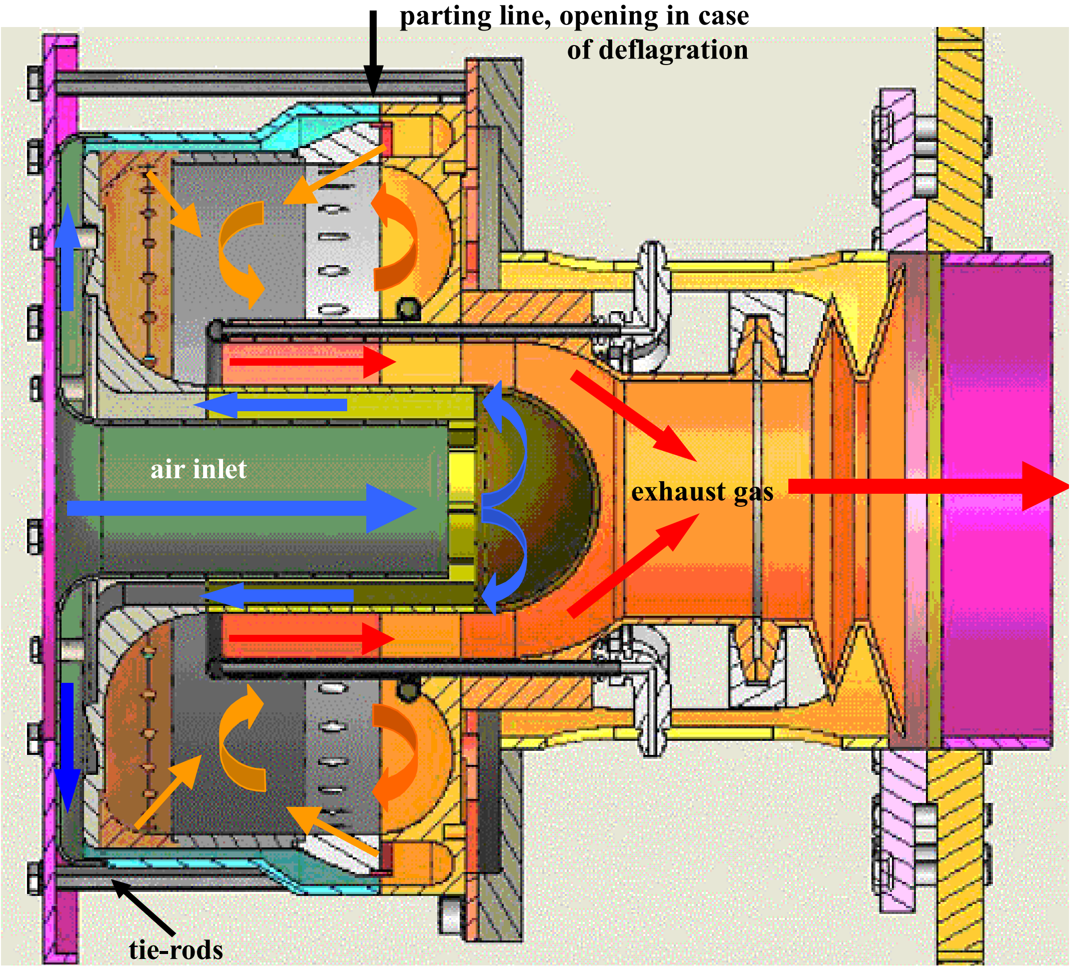


Fig 8.18 View cross section FLOX-Combustor operation mode (source: Technion)

The flame inside the combustor during the measurements is still detected by an IR-Sensor (see chapter 0). The assembly for the junction of the IR-Sensor was redesigned.

The safety regulations for the fuel supply are still the same. A flame protection and a fast closing valve in a double execution prevent the flame from reaching the fuel supply.

Never the less IDG decided to keep hold on safety equipment for the air pipe system such as rupture disc, fast closing valve and high pressure pipes, to minimise the risks for the test field, equipment and the staff.

Measurement Setup

The parameters measured at atmospheric conditions will be the same as planned for the pressurised combustion. So the measured data will be:

- Air mass flow
- Combustor inlet temperature
- Combustor inlet pressure
- Fuel mass flow
- Fuel injection pressure
- Fuel/Air ratio
- Pressure in the exhaust pipe
- Exhaust gas temperature level
- Exhaust gas temperature distribution (pattern factor)
- Emissions

Furthermore the combustion zone will be observed by optical equipment.

The used measurement techniques are the same as described in detail in chapter 0.

Experiments

The experimental investigations are divided into two steps:

1) Initial operation check of the test rig, especially the combustion chamber, the safety equipment and measuring technique.
2) Experimental investigations of the FLOXCOM combustion.

The experimental investigations should verify the excellent performance of this combustor concept with regard to:

a) Complete and efficient energy conversion
b) Stable combustion
c) Temperature distribution at the combustor outlet
d) Low NO_x-Emissions

Criteria for the assessment of these properties are:

- Homogeneous temperature distribution at combustor outlet
- Species concentrations in the exhaust gas
- Flame behavior

To analyse the operational behavior of the combustor, to find out the stable operation mode for the optimised combustion and to work out measures to improve the existing layout configuration, the above mentioned properties are analysed in dependence of the following parameters:

- Air inlet temperature
- Air mass flow
- Fuel mass flow resp. equivalence ratio

During the investigations the following parameters will be measured continuously.

- Combustor wall temperature in-/ outside
- Combustion glimmer
- Exhaust gas temperature
- Exhaust gas emissions

At the beginning the measurements will be compared with the measurements of the 60° combustor segment from Instituto Superior Técnico (IST) Portugal (see Fig).

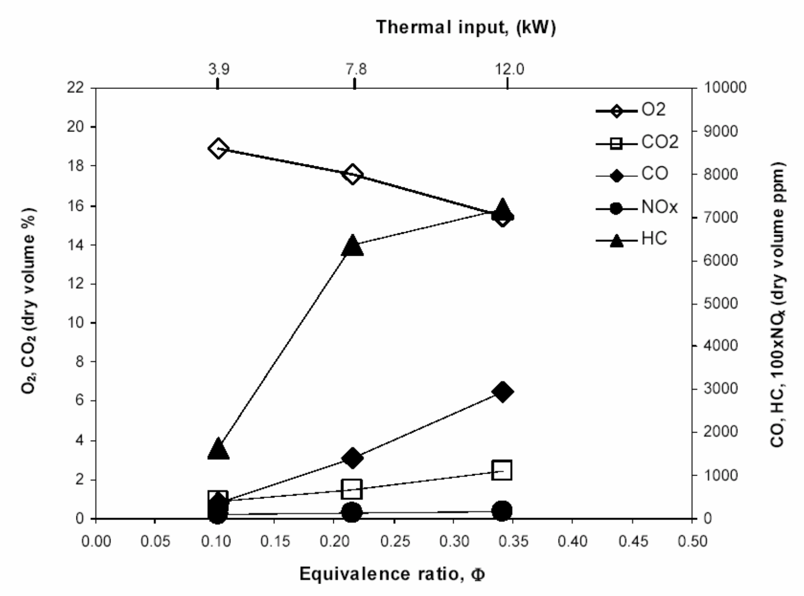


Fig 8.19 Flue-gas data for the combustor model of 60° atmospheric combustion at IST (Portugal)

It is planned to proof the transferability of the results of the 360° atmospheric pilot combustor with the measurements of the 60° atmospheric sector combustor at the IST in Portugal.

The FLOX phenomenon mainly depends on the internal recirculation (see Fig 8.). Therefore in addition the circulation and thus the dwell time of the air/fuel mixture within the combustion zone will be varied well-aimed by closing every second “air 2” hole (see Fig 8.20).

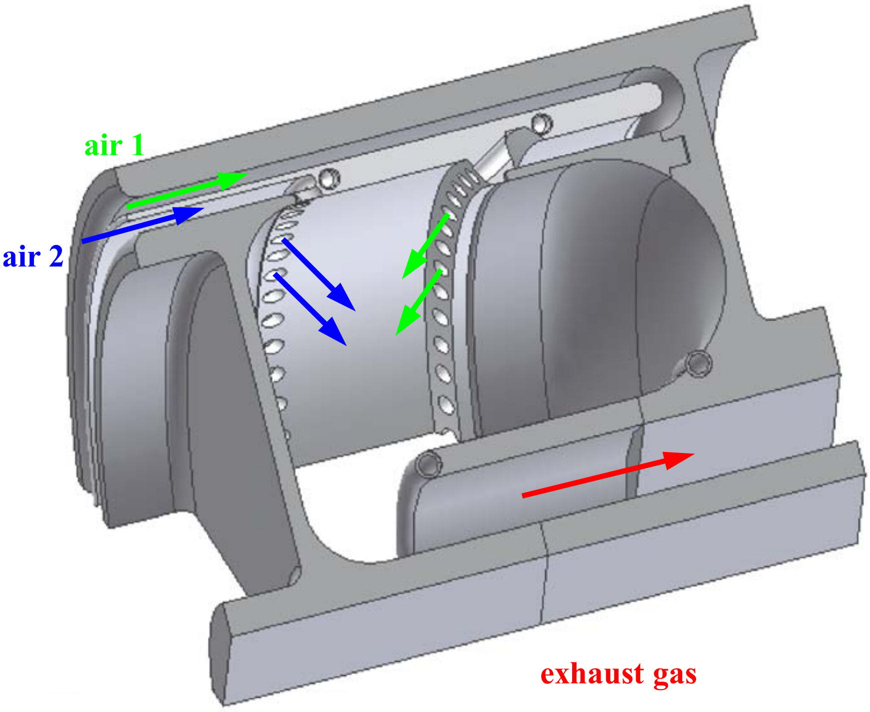


Fig 8.20 Combustor configuration

The measuring schedule is shown in Table 3.

Table 3: Planned measurements for 360° atmospheric combustor

Geometry	T_{in} $[K]$	$\dot{m}_{Air}$ $[kg/s]$	$\dot{m}_{Fuel}$ $[kg/s]$	Φ
all air holes open	varying	const	const	const
all air holes open	const	varying	const	varying
all air holes open	const	const	varying	varying
every second “air 2” hole closed	varying	const	const	const
every second “air 2” hole closed	const	varying	const	varying
every second “air 2” hole closed	const	const	varying	varying

As soon as the combustor arrives, the installation of the combustor will start at once. The planned test program will be accomplished and the results will be analysed and summarised in a supplementary report.

Conclusions

Among participating at WP 7 under the leadership of B&B AGEMA IDG developed and designed the complete test rig for the pressurised 360° combustor. A safety concept with fast closing valve, rupture disc, flame detection, gas detector and gas warning system was developed, too.

IDG investigated the possibility to use gaseous fuel according to the boundary conditions of the institute’s test field. A data acquisition unit for temperature and pressure measurements as well as an exhaust gas analysis system were set up and implemented into the test facility and combined with the safety control unit.

The test facility was completely prepared for the installation of the pressurised combustor.

In the proposal a well considered concept of complementary tests of the partners was developed and submitted to the EU, starting with experiments at a 60° sector combustor at atmospheric pressure (IST), continuing with investigations at a 60° sector combustor under pressurised conditions (Ansaldo) and coming up to the final stage by a full scale testing under pressurised conditions (IDG) (see Table 5). Finally it was planned to demonstrate the function of the combustor under practical conditions as close to the engine as possible.

Table 5: Experimental Overview

Combustor Angle [°]	Pressure	Temperature	Institute
60	p_a	hot	IST (P)
60	$p > p_a$	hot	Ansaldo (I)
360	$p > p_a$	hot	IDG (D)

During the design process of the combustor, done by Technion, it comes out that the costs for the manufacturing of the combustor exceeded the estimated amount significant, due to the German regulations for pressurised combustion. Thus Technion - as coordinator - decided to change the experimental concept and to run the test rig under atmospheric pressure conditions, although these new operating parameters changed the design and the geometry of the combustor as well as the test rig substantially. These changes caused a delay in the project, thus the full-scaled pilot combustor is not delivered at the Institute of Steam and Gas Turbines at Aachen University until now.

In consequence of this change a large investment had to be done by IDG to compensate the design changes. Caused by the new parameters IDG had to provide an air heater to keep the inlet temperature at a constant level. Resulting work such as heater control programming, redesigning of the connecting pipes for the new combustor geometry and revision of the safety concept and the fuel supply caused a additional personal and equipment expenses.

As soon as the atmospheric combustor arrives the whole planned test program will be accomplished and the results will be analysed as described in chapter 0. An addendum report will be submitted to the EU as soon as the measurements will be finished.

The additional personal and equipment expenses after the end of duration of the project for finishing the project will be financed from IDG's own resources.

Literature

/1/	DIN 1952	„Durchflussmessung mit Blenden, Düsen und Venturirohren in voll durchströmten Rohren mit Kreisquerschnitt“ Beuth-Verlag GmbH Berlin
/2/	Steinbach, Jörg	Chemische Sicherheitstechnik VCH Verlag ISBN 3-527-28710-8
/3/	Sicherheitsblatt Erdgas	Firma Transgas AG
/4/	Kammerer	„Explosionsschutzschieber ab DN 250“, Stand August 2002
/5/	Lamtec Mess- und Regelungstechnik	„Anleitung für die Montage und Inbetriebnahme Lambda Transmitter und Lambda-Sonde LS 2Ex“
/6/	Kobold Messring GmbH	„Oszillations-Durchflussmesser für Gase“
/7/	VDI Wärmeatlas	Berechnungsblätter für den Wärmeübergang VDI Verlag Karlsruhe 1988

2.2.9 WP9 PILOT COMBUSTOR MANUFACTURE

WP Leader: Technion

Objectives: 1. Design specification and details of the pilot combustor.

Deliverables :

D9.1 Design specification of the pilot combustor - report (M20)

D9.2 An operating pilot combustor - hardware (M24).

The design of the pilot combustor was complete. Two models were allowed and they differ mainly in the way is introduced into the combustion zone, The difference between two types can be seen through the different combustor sector (see figures 4.8.c and figure 8.20).

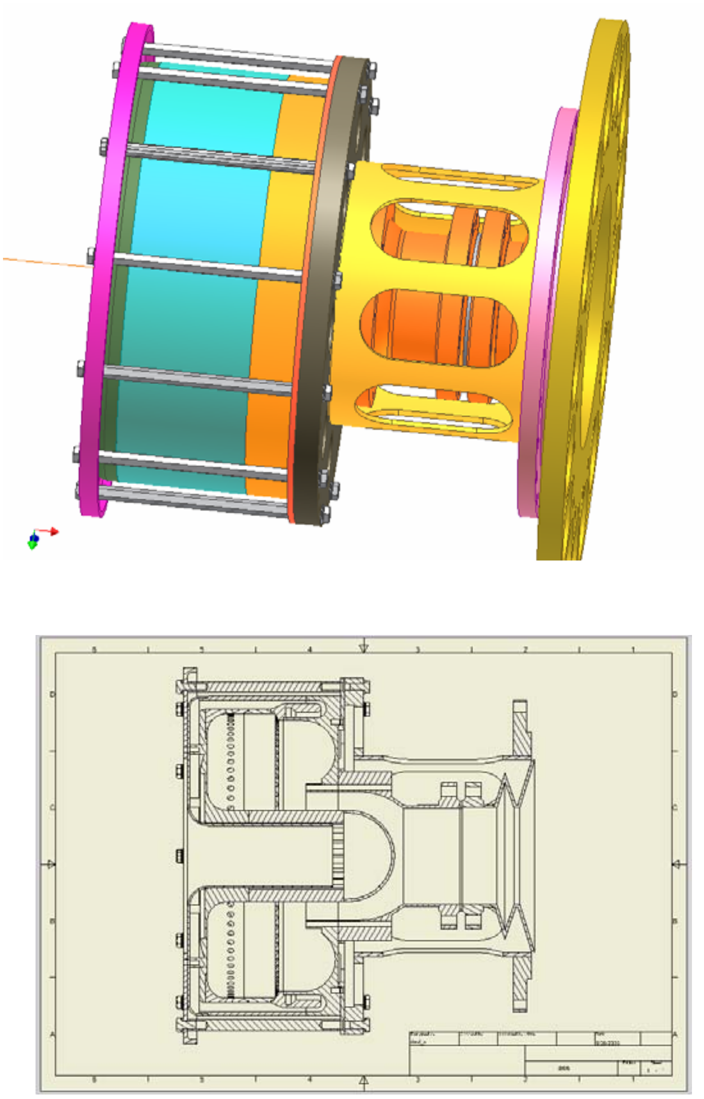
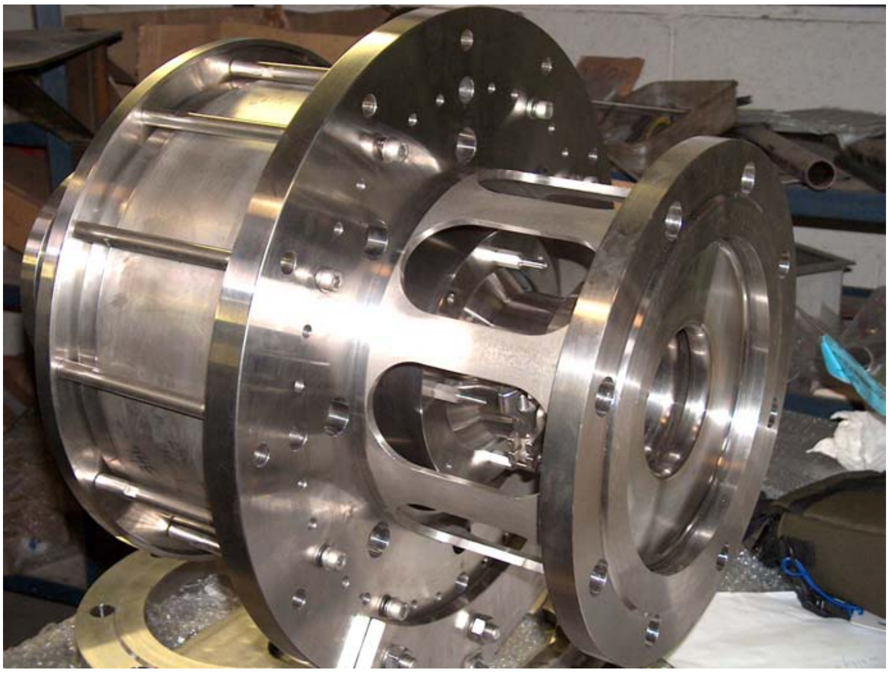


Fig 9.1 Engineering drawing of the pilot combustor (a) and its 3D view (b)

The drawings of the pilot combustor are given in the following; however the detailed drawings could not be attached due to the numerous numbers of components, hence only the assembly in shown here. The photographs of the combustor itself are shown in figures 9.2 and 9.3. The test rig for atmospheric pressure including the cooling chamber is show in figures 9.6 – 9.8.



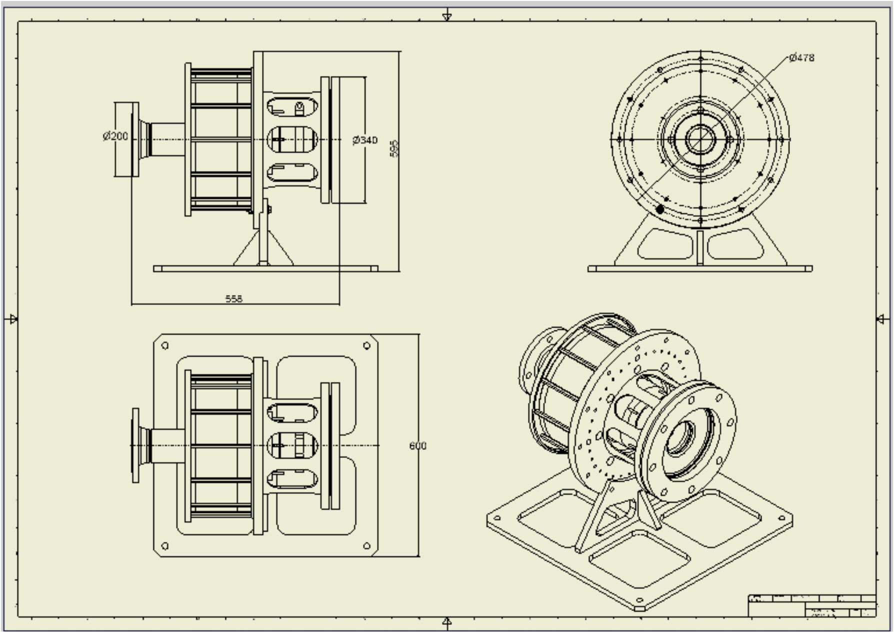


Fig 9.2 The pilot combustor – back view

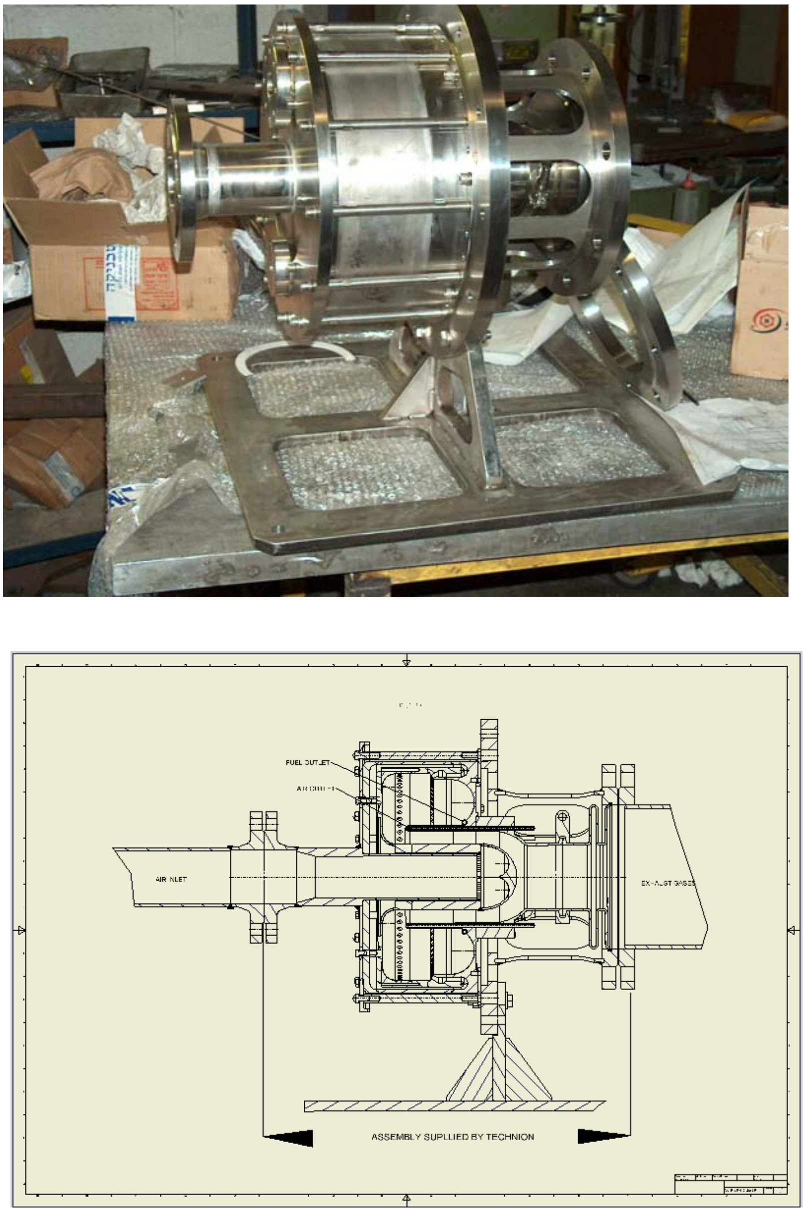


Fig 9.3 The pilot combustor - inlet section

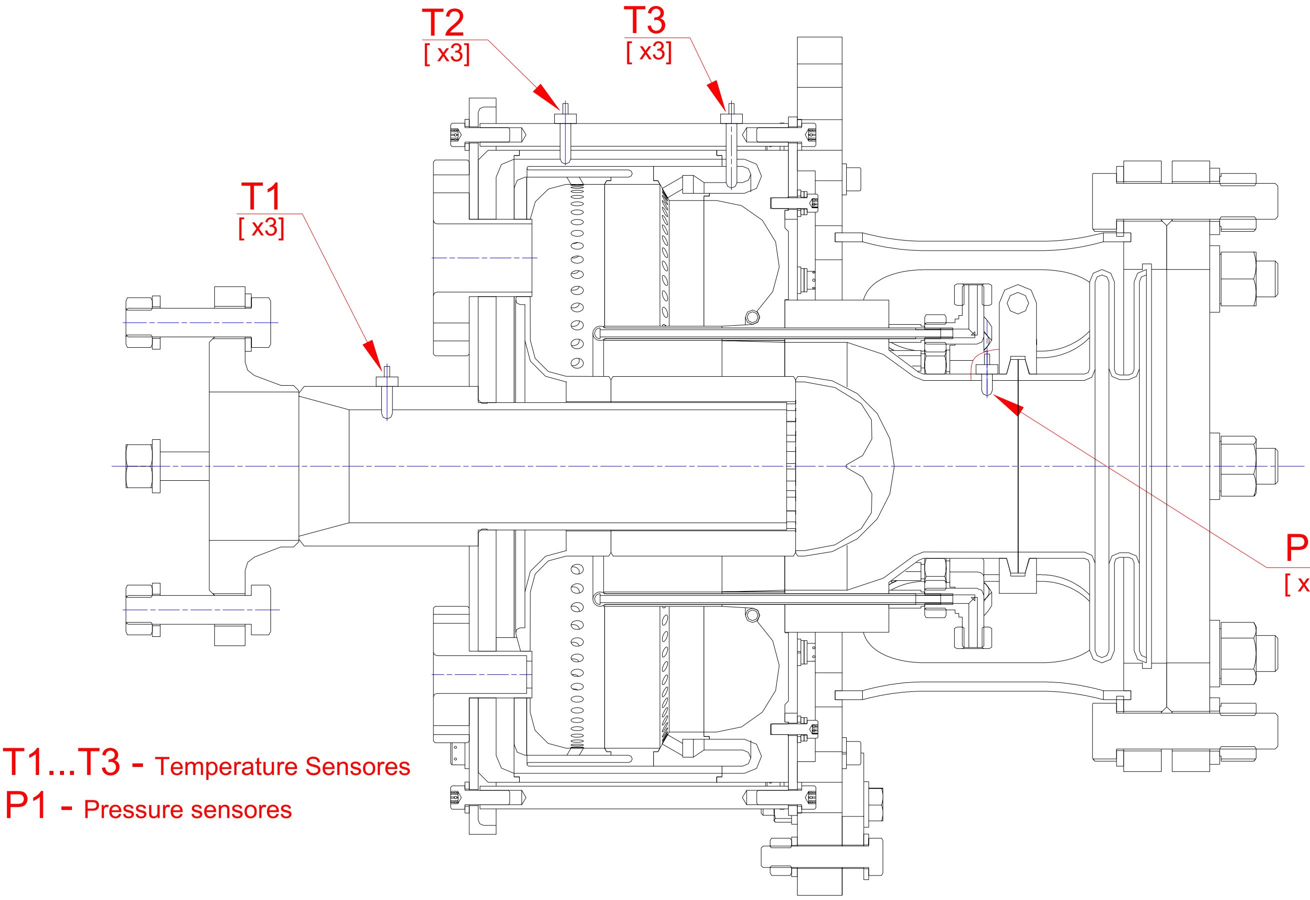


Fig 9.4 Schematic of measurement points inside the pilot combustor

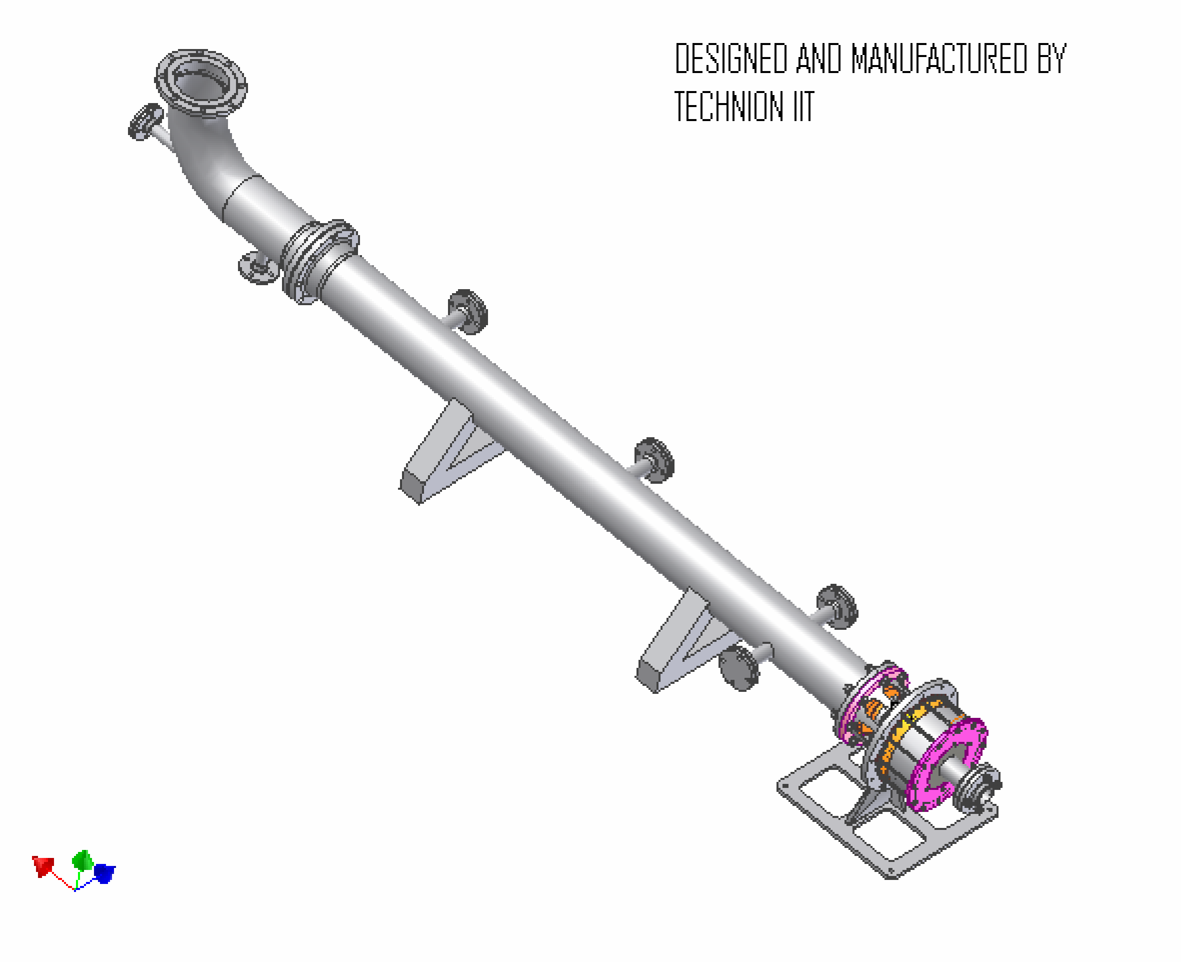


Fig 9.5 Schematic exhaust system (attached to the pilot combustor)

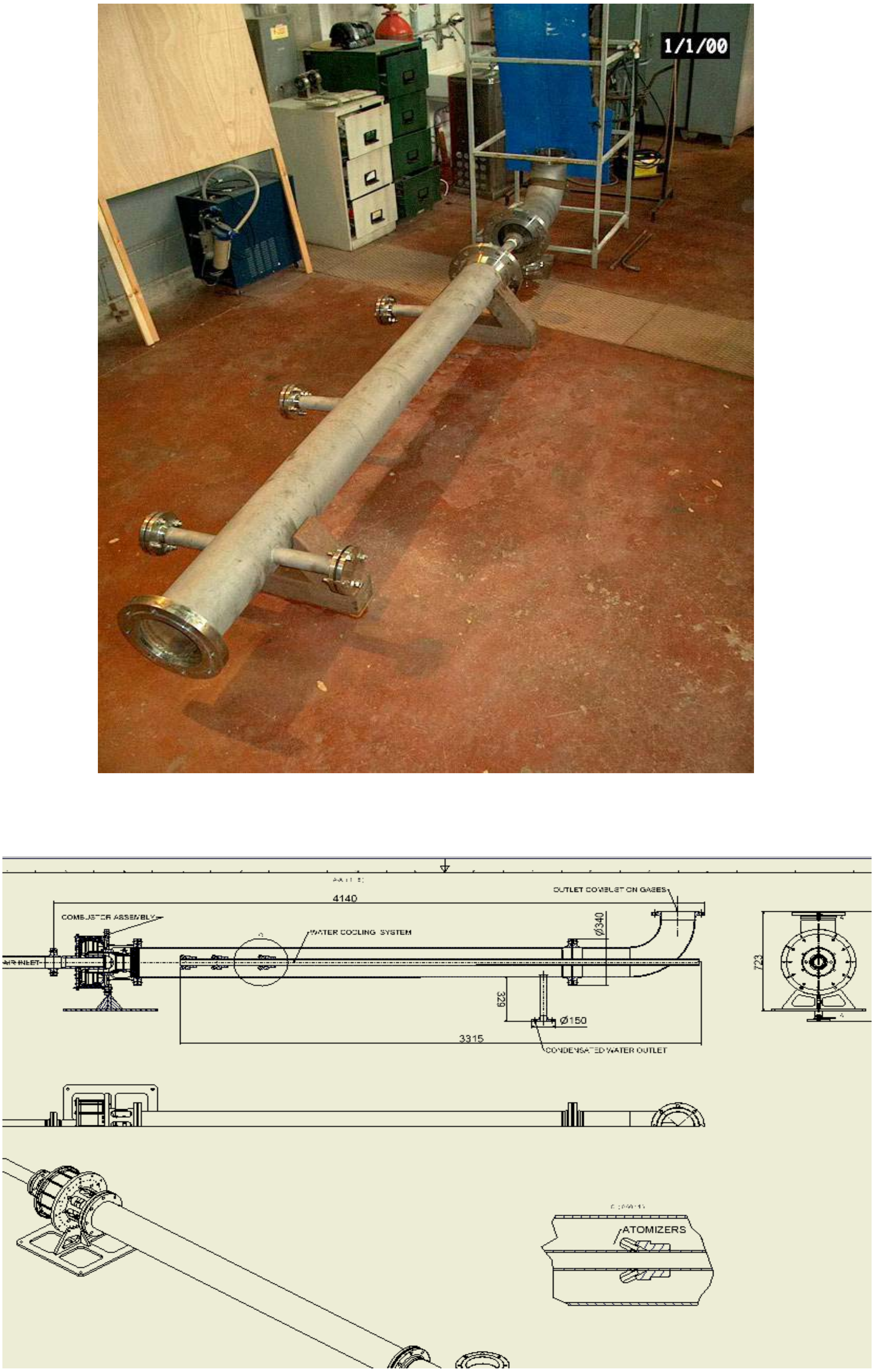


Fig 9.6 Exhaust system for the pilot combustor

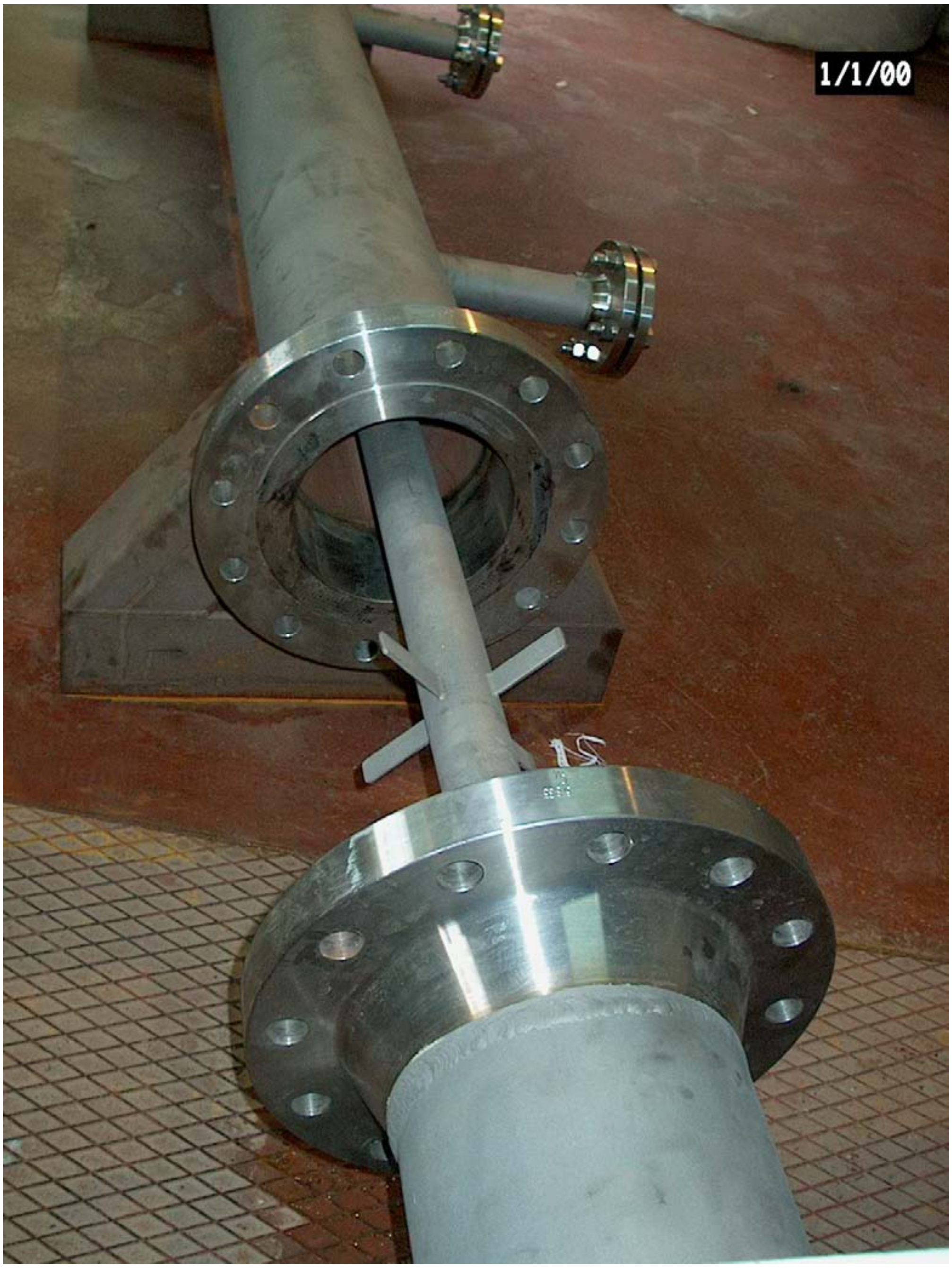


Fig 9.7 Internal water pipe for water injection

Fig 9.8 Water injection nozzle for gas exhaust cooling

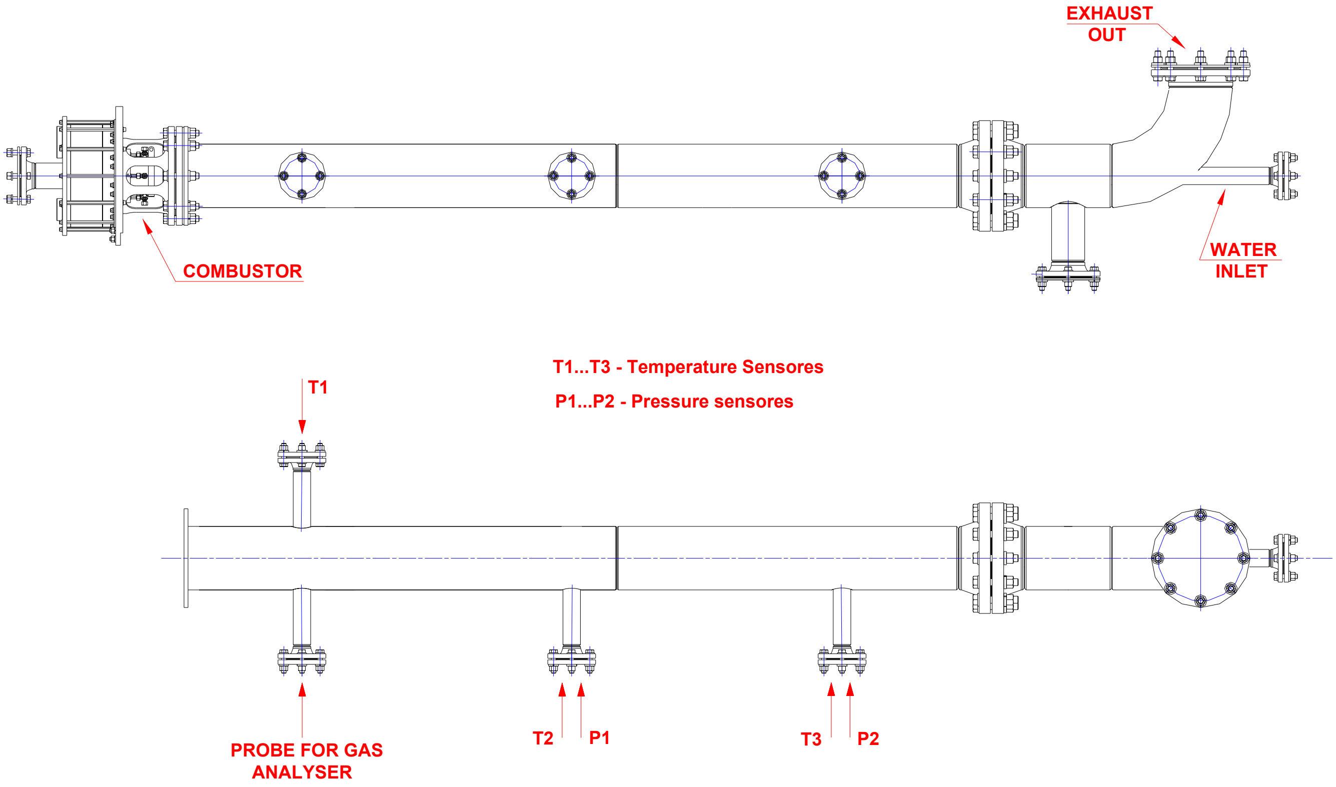


Fig 9.9 Measurement points for the Exhaust system

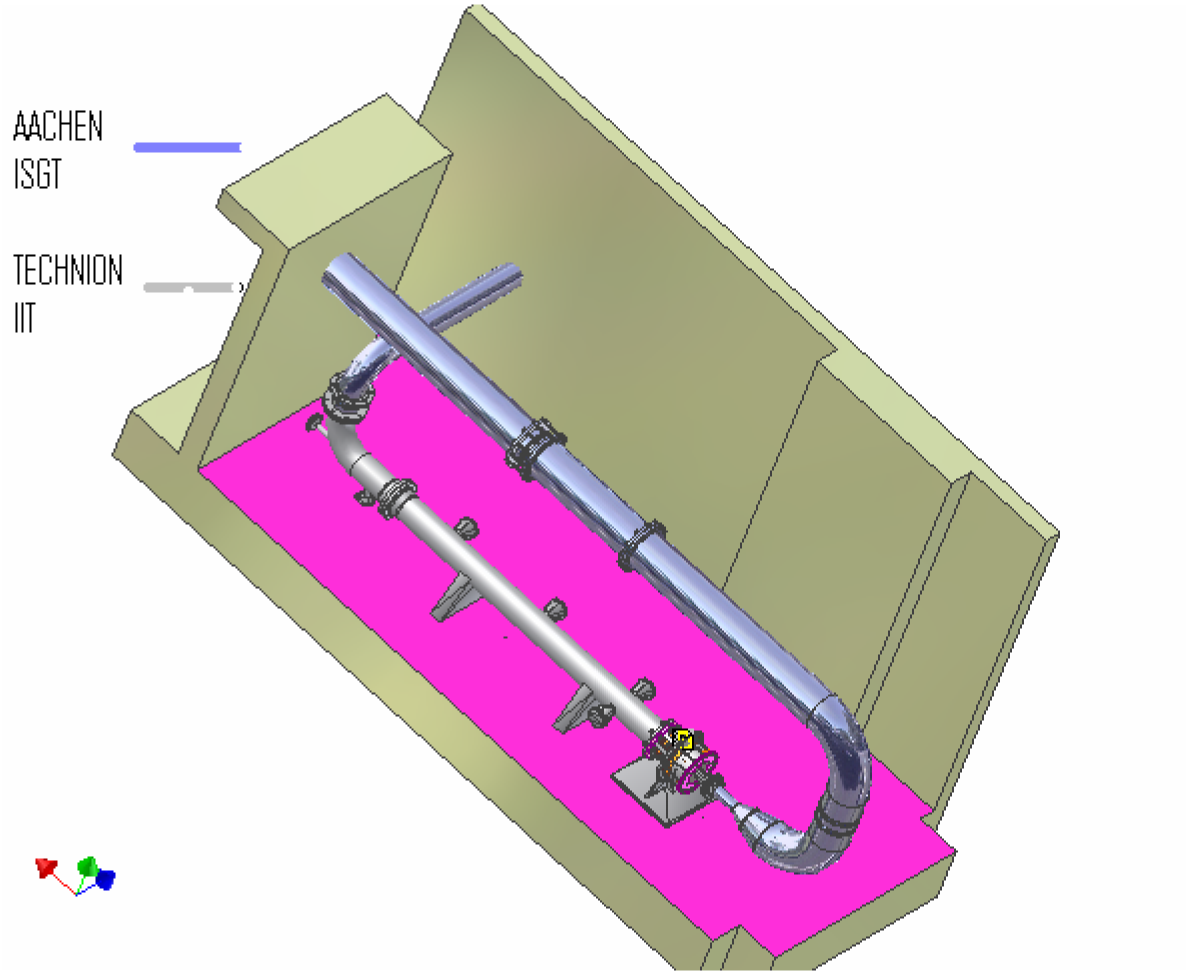


Fig 9.10 Schematic of the test stand at the RWTH including the intake pipe, the pilot and the exhaust inside the test room

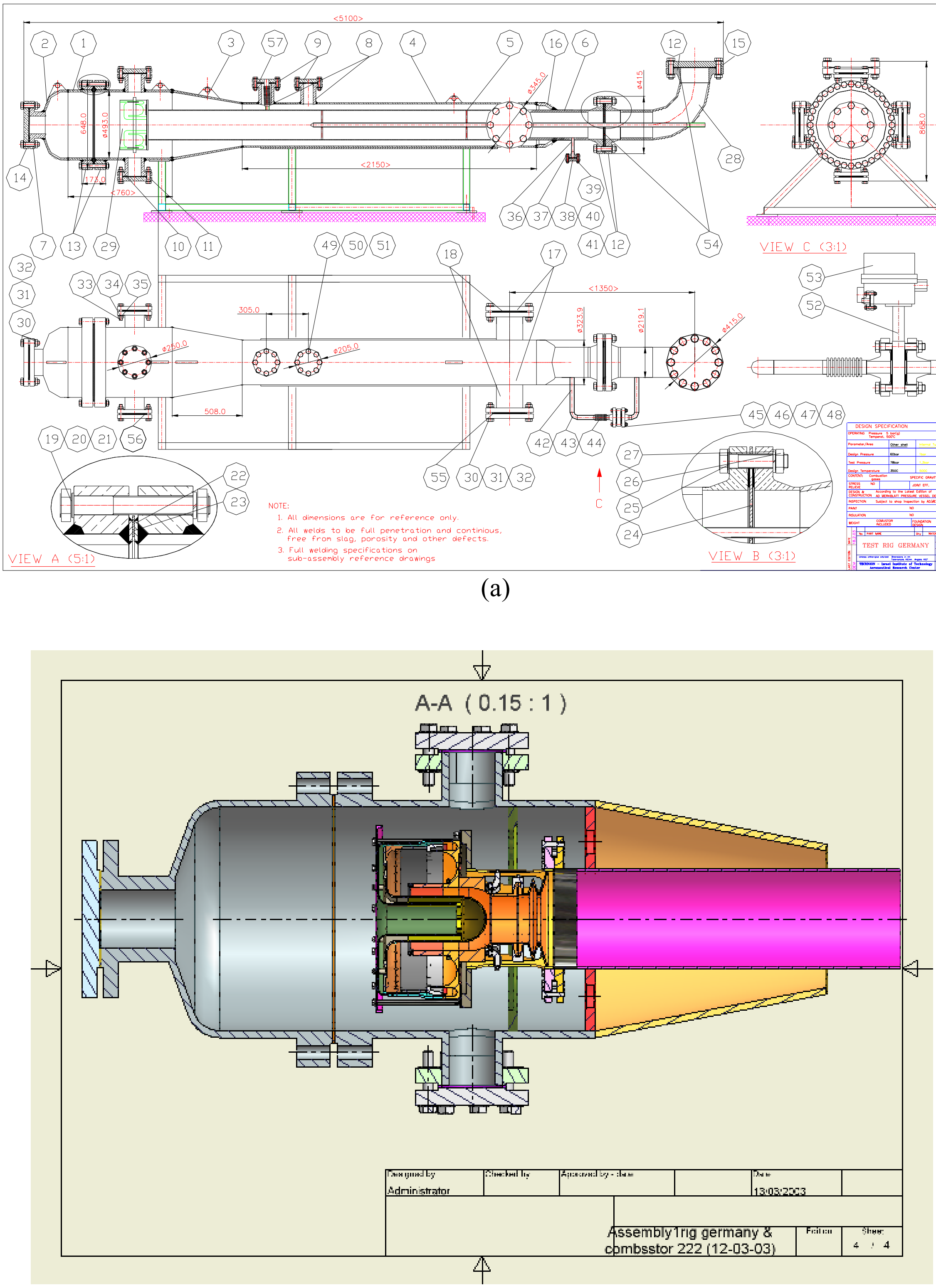
PRESSURIZED TEST RIG

Initially the requirements for the test rig were as following:

Design inlet pressure: 6.0 bar (absolute)
Operation inlet pressure: 4.5 bar (absolute)
Inlet temperature: 75°C <= T <= 140°C

Design outlet pressure: 6.0 bar (absolute)
Operation outlet pressure: 4.5 bar (absolute)
Design outlet temperature: 500°C
Operation outlet temperature: 400°C

It was required by the RWTH that every part in the pressurized section before the outlet valve or outlet orifice (e.g. the cooler and the combustor itself) had to be designed for 60 bar. Accordingly a test rig was designed with all details, see figures 9.11a and 9.11b. However it was then realized that it was not practical (in terms of cost and delivery date) to construct and deliver and install it in Aachen and a simplifier was designed and built (for atmospheric testing).



NOTE:
1. All dimensions are for reference only.
2. All welds to be full penetration and continious, free from slag, porosity and other defects.
3. Full welding specifications on sub-assembly reference drawings

DESIGN SPECIFICATION

OPERATING Pressure 5 bar(g)
Temperat. 500°C

Parameter/Area	Other shell	Internal Tube	Exhaust Sect
Design Pressure	60bar	1bar	60bar
Test Pressure	78bar	1.5bar	78bar
Design Temperature	350C	500C	700C

CONTENT: Combustion gases
SPECIFIC GRAVITY
STRESS RELIEVE NO
JOINT EFF.
DESIGN & CONSTRUCTION According to the Latest Edition of AD MERKBLATT PRESSURE VESSEL DESIGN
INSPECTION Subject to shop inspection by AD.MERKBLATT supervisor
PAINT NO
INSULATION NO
WEIGHT COMUSTOR INCLUDED
FOUNDATION DESIGN

No.	PART NAME	Qty	MATERIAL	NOTES

TEST RIG GERMANY
SCALE 1:1
DESIGNED DEVI
DRAWN DEVI
CHECKED
DATE 18.2.03
LAST EDITION DEVI
TECHNION - Israel Institute of Technology
Aeronautical Research Center
Pt. No. GER-00.00.00

(a)

Designed by Administrator
Checked by
Approved by - date
Date 13/03/2003
Assembly1rig germany & combsstor 222 (12-03-03)
Edition
Sheet 4 / 4

(b)

Fig 9.11 A general view of the pressurized testing for the pilot combustor (a) and its cross section (b)

Appendix

SWIRL ATOMIZER FOR COOLING SYSTEM

In the framework of the FLOXCOM project, a special test-rig for investigation of a combustor model with low NO_x emission was developed. One of the requirements to this test-rig is cooling the hot exhaust gases from 1200 K to 770 K. Preliminary calculations showed that for coo1ing exhaust gases of 1/6 combustor model, the water flow rate should be about 70-100g/s. Detailed calculations which takes into account droplet evaporation time [1] disclosed, that a maximal droplet diameter should not exceed100-120 microns. As available atomizers can not provide such flow rate and spray quality, a new swirl atomizer was developed and investigated by the Technion Turbo and Jet Laboratory.

In order to provide macro and microstructure of spray the following tests were carried out:

1. Continuous operation atomizer characteristics [2], i.e. dependence mass flow rate on pressure drop and the atomizer "flow number".
2. Spray cone angle.
3. Dependence droplets size and droplets size distribution on the atomizer pressure drop.

Principal results

1. Dependence mass flow rate on pressure drop is shown in Fig. 1.

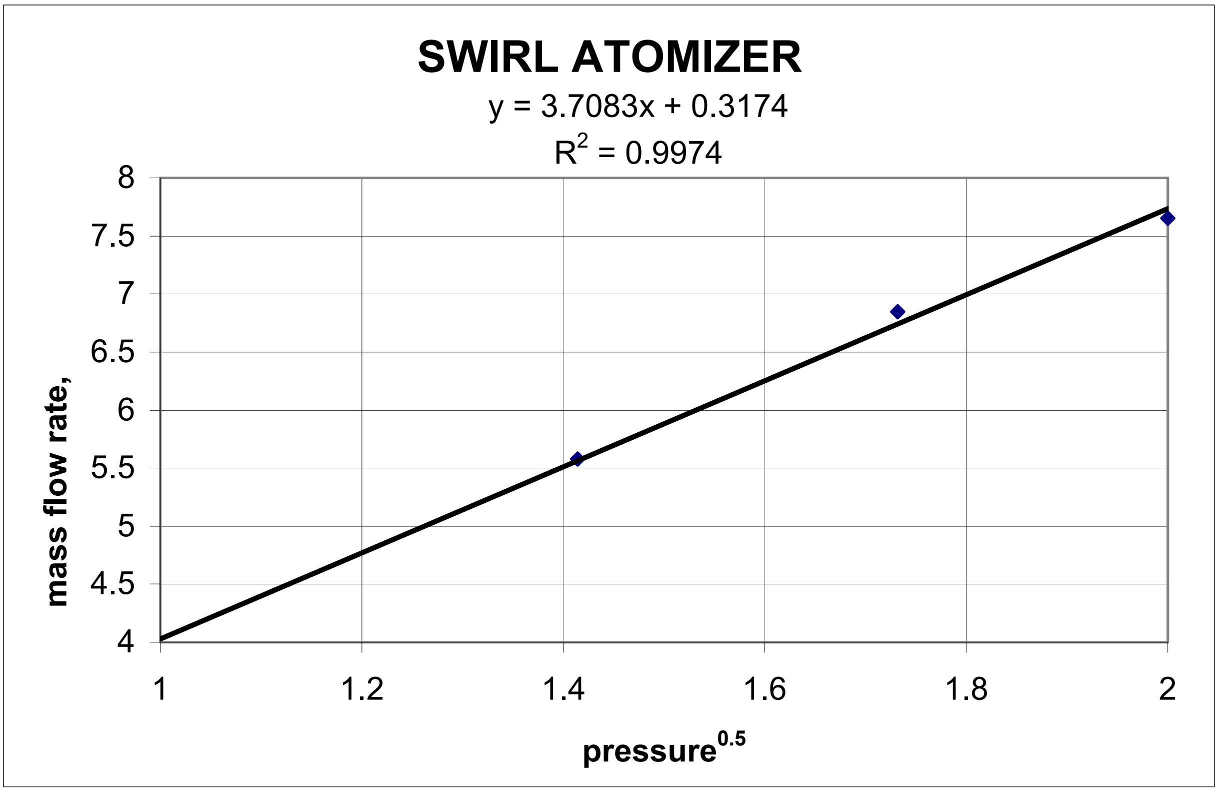


Fig A.1 Dependence mass flow rate on pressure drop

According to this data atomizer *flow number* is equal to **$3.85{\cdot}10^{-7}$**. Fig. A.1 shows, that the atomizer satisfies flow rate requirements (8 atomizers for test-rig are provided).

2. The atomizer spray cone angle was found from the spray photos (Figs. A.2, A.3).

Fig A.2 Spray cone, pressure drop 2 bar

Fig A.3 Spray cone, pressure drop 7 bar.

The spray cone is equal to 67 degrees for pressure drop 2 bar and 77 degrees for 7 bar.

4. Dependence droplets size and droplets size distribution on the atomizer pressure drop is shown in Fig. A.4.

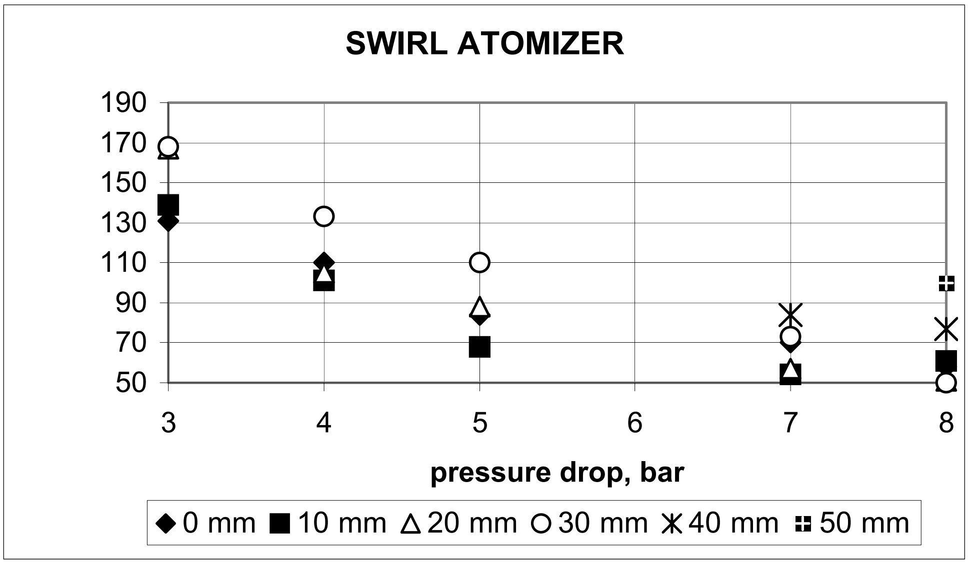


Fig A.4 Dependence droplets size and droplets size distribution on the atomizer pressure drop.

One can see that droplets size decreases with the pressure drop increasing.

Detailed droplet performances are shown in Appendix.

Conclusion

The developed swirl atomizer provides required mass flow rate and droplets size and it is suitable for cooling exit gases of the experimental test-rig.

References

1. A. H. Lefevre. *Gas Turbine Combustion*, Taylor & Francis, 1999.

2. Abramzon B., and Sirignano W. A.*, "Droplet Vaporization Model for Spray Combustion Calculations."* Int. Journal of Heat and Mass transfer, Vol. 32, Sept. 1989, pp.1605-1618.

Recommendations for further development based on test results

The project was conducted by 8 partners each one contributing in his own field for the assembly of a novel gas turbine combustor for low NOx pollution. The project has reached a critical stage where the theory of operation is now understood and validation tests were performed at atmospheric and elevated pressure which confirmed the general principle of the new combustion method. These tests were performed under nearly realistic industrial conditions and hence represent the validity of the method and its feasibility as a potential attractive combustion method that will present an alternative to the existing low NOx combustion methods which were found to be unstable and problematic.

The current level of knowledge is sufficient to understand the principle of operation and its basic features which are sufficient to gain the necessary confidence in its potential. However, there is a necessity to further develop the associated technology in-order to bring forward the actual engineering design. Such work requires further optimization of the combustion method and the associated combustion design. Hence further research budget is needed in order to elevate the technology to an engineering standard that would allow exploiting and benefiting from this new innovative combustion method.

Part 3: Management Final Report (CONFIDENTIAL)

3.1 List of deliverables

Part 3: Management Final Report (CONFIDENTIAL)

3.1 List of deliverables

Deliv. No.	**Deliverable title**	**Deliv. Date** (1)	**Dissemination Level** (2)	**Type** (3)	**Deliv. Status**
D1.1	Design of the combustor sectors	11	RE	R	☺
D1.2	Experimental Model of the combustor sector suitable for hot and atmospheric tests (for IST).	12	RE	H	☺
D1.3	Experimental Model of the combustor sector suitable for hot and pressurised conditions (for Ansaldo Ricerche S.r.l.).	16	RE	H	☺
D2.1	A mathematical prediction procedure of the combustion aerodynamics applicable to the general case of gas turbine combustion and validated for FLAMELESS OXIDATION conditions.	10	PU	S	☺
D2.2	Experimental data and numerical analysis of FLAMELESS OXIDATION fundamentals	19	PU	R	☺
D2.3	Prediction procedure of pollutant emission model	16	PU	S	☺
D2.4	Extracted conclusions that give enhanced insight to the physics of gas turbine combustion chambers.	26	RE	R	☺
D3.1	Integration of the new combustion, atomisation and evaporation models into an existing computational procedure	22	RE	R	☺
D3.2	Validation predictions of selected gas turbine combustor data.	30	RE	R	☺
D3.3	Recommendations for improving the pattern factor, NOx and unburned emissions, combustion efficiency and flame retention performance of the FLAMELESS OXIDATION design concept.	36	RE	R	☺
D4.1	Combustor design, operating conditions and selection of fuel supply systems..	5	RE	R	☺
D4.2	Numerical simulation of the combustor performance for different fuel supply systems.	21	RE	R	-
D4.3	Experimental investigation of the fuel supply system	23	PU	R	☺
D4.4	Experimental investigation of the combustor model	32	RE	R	☺

D5.1	Transparent combustor model.	12	PU	H	☺
D5.2	Interaction of the fuel injection aerodynamics and the main vortex.	22	RE	R	☺

Deliv. No.	Deliverable title	Deliv. Date (1)	Dissemination Level (2)	Type(3)	Deliv. Status
D5.3	Interaction of the cooling jets with the main vortex flow.	30	RE	R	☺
D6.1	Experimental data (flow field measurements and overall performance);	30	RE	R	☺
D6.2	CFD results and experimental data assessment.	30	RE	R	☺
D7.1	Design specifications for the pilot combustor from thermal load point of view.	15	RE	R	☺
D7.2	Evaluation of the results from operation of the small scale model	19	RE	R	☺
D7.3	Numerical simulations of wall cooling including combustion aerodynamics.	32	RE	R	☺
D8.1	Design specification and assembly of the interface to the pilot combustor.	22	RE	R	☺
D8.2	An assembled pilot combustor, operating within the test facility.	24	RE	H	partially
D8.3	Testing performance of the pilot combustor.	34	RE	R	-
D8.4	Recommendations on exploitation of results for further development	36	RE	R	☺
D9.1	Design specification of the pilot combustor.	20	RE	R	☺
D9.2	An operating pilot combustor.	24	RE	H	☺
D9.3	Recommendations for further development based on the results.	36	RE	R	☺

[1] Month in which the deliverables will be available. Month 0 marking the start of the project, and all delivery dates being relative to this starting date

[2].PU =Public
RE = restricted to a group specified by the Consortium
CO= Confidential, only for members of the Consortium

[3]R = Report
H = Hardware
S = Software

3.2 Comparison of initially planned activities and work actually accomplished

The FLOXCOM project is a challenging one. It aimed at developing a new combustion method that would present an alternative to current dry low NOx combustion technologies appropriate to be used in gas turbine.
This is a project of large scale that relates to many technological aspects. The FLOXCOM development program aimed to refer to most practical aspects of the development work. It was planned more than a year before it actually started and lasted for three years. It is obvious that over such a long period, while handling topics related to novel combustion method, the actual work would somehow differ from the one envisioned. However, due to dedication and the invested effort by all parties of the consortium, significant progress was achieved while fulfilling most of the tasks and deliverables.
This can be seen from the above table where the fact that most of the deliverables were delivered is clearly marked. The main achievement of the project can be summarized, to my opinion, in two topics; 1st) the collaboration and gain of knowledge by the members of the consortium and, 2nd) the successful operation of the FLOXCOM combustor at elevated and atmospheric pressures. It is difficult to judge which is more significant or of better value.
The smooth operation of the program, the responsibility of each partner and the friendship that was developed throughout the program, has lead to fruitful collaborative work that enabled to bring the work to an advanced stage where we could state that FLOXCOM IS a viable low NOx combustion model, potentially suitable for gas turbine application

3.3 Management and co-ordination aspects

- performance of the consortium and individual partners in terms of dedication, motivation and contribution, including supply of deliverables, organization, management and communication aspects.

I t is not easy for me to write about the consortium because I can only repeat what said before and by that repeat the compliment I would like to give my colleagues within the consortium. To my opinion, the consortium, in general, was a successful one. By that I refer to the cooperation and dedication of all the partners to fulfill their share in the joint project .
The work that was assigned to each partner in the beginning of the project was carried out by all partners. There were 6 consortium meetings during the project. All meetings were carried out while all partners were present. All meetings included detailed reports of the work progress carried out and most meetings were performed at the different partner's sites, hence the meetings soon became a professional and social event.
The website, controlled by Prof. Kowalewski of IPPT-PAN, is an active and very clear site that includes massive information on the project and is quite useful for the dissemination of the project.
All partners have expressed their will to participate in the next stage of the project and consequently additional research funds are being seeked for.
It is our intention to continue the work in this field till an engineering version of the combustor is developed and by that contribute our share for a better tomorrow.

Name and contact details for the follow-up of the project.:

Associate Professor Yeshayahou Levy

Head, Turbo & Jet Engine Laboratory,
Coordinator to FLOXCOM

Faculty of Aerospace Engineering,
Technion, Israel Institute of Technology,
Haifa, 32000,
Israel

e-mail: levyy@aerodyne.technion.ac.il
http: //jet-engine-lab.technion.ac.il

Tel (Work): ++ 972 - 4-829 3807
(Home): ++ 972 - 4-825 4079
(Mobile): ++ 972-507-355 888
Fax (Work): ++ 972-4-812 1604
(Work): ++ 972-4-829 2030