20th Australasian Fluid Mechanics Conference
Perth, Australia
5-8 December 2016

On the development of a Mach 10 scramjet engine for investigation of supersonic combustion regimes

A. F. Moura[1], V. Wheatley[1], T. J. McIntyre[2] and I. Jahn[1]

[1]Centre for Hypersonics
The University of Queensland, Queensland 4072, Australia

[2]School of Mathematics and Physics
The University of Queensland, Queensland 4072, Australia

Abstract

Supersonic turbulent combustion presents many challenges and the regimes present in realistic scramjet combustors are not completely known. To experimentally investigate the combustion process in a scramjet engine, it is desirable that the flow be as representative as possible of a complete engine. In direct-connect experiments, the flow is usually too uniform, motivated by the desire to keep the flow as simple as possible. However, the non-uniformities in the internal flow path of scramjet engines have been observed in simulations to drive a large variation in turbulent combustion regimes along the combustor. These considerations set constraints on the development of a flow-path for experimental investigation of scramjet combustion: it should be simple enough to allow detailed analysis and optical flow diagnostics, but also representative of the complex flow in realistic scramjets. This paper presents the development of one such flow-path for a Mach 10 free-stream condition. With the use of CFD analysis employed at the design stage of the experiment, a simplified two-dimensional scramjet engine has been developed, complete with fore-body. Simulations were used to evaluate fuelling conditions, the development of the combustion process and effects from side-walls and corner vortices. Through iterations on the engine design, a model is created that is more reliable and whose behaviour can be better anticipated.

Introduction

Supersonic turbulent combustion is a highly complex process which is difficult to model. Time scales are usually different for combustion reactions and turbulent eddies, resulting in high computational costs [16, 17]. Although there are several models for low speed turbulent combustion that achieve accurate results [13, 17], the developments for hypersonic flow have been limited [6, 19]. Few models are available and little testing has been done to compare them. Even the physics of the supersonic combustion process are still not thoroughly understood [12, 16].

High-fidelity large-eddy simulations of an inlet-fuelled scramjet engine have shown that combustion occurs in several regimes within the flow-path [10]. This complicates modelling efforts as most models, developed for a single or few regimes, are incapable of appropriately simulating the entire problem. Detailed experimental data is needed to develop and validate models and to better understand supersonic combustion in scramjets. Direct-connect experiments are insufficient for this goal, as their flow is too uniform, without the complex flow structures generated by the scramjet inlet [6, 9]. This limits the variation in combustion regimes that can be observed in these experiments. Therefore, an experiment is needed with a model representative of a full engine and its non-uniformities, that also allows flow visualization of the complete combustion process.

Developing such an experiment adequately involves many constraints, some of them contradictory, demanding balance to achieve the desired outcome:

- The model should be simple enough to avoid complicating flow phenomena.
- The model should be representative of a full inlet-fuelled scramjet engine, including a fore-body.
  The inlet shock waves drive mixing and ignition processes, hence it is essential to include them. However, the influence other effects such as corner vortices and side-wall effects have on the combustion process can complicate analysis and are undesirable. Avoiding these influences ensures the model can be simulated with higher order methods.
- There must be optical access for flow visualization along the entire combustor.
- Size constraints due to size of test section and coreflow.
- To mimic efficient scramjet operation, inlet compression should be as low as possible as actual scramjets need to reduce losses for higher performance flight.
- Autoignition of hydrogen fuel achieved by primary shocks
- Robust combustion ($>50\%$ combustion efficiency),
  Combustion must be robust enough in order to visualize the full range of combustion regimes, including those where main heat release occurs.

CFD simulations are invaluable in the design of an experiment that meets all these requirements. CFD is used from the start of the process and its results are used in the design of the model, which is modified iteratively to optimize the design.

This paper describes the design process and final design of a scramjet model for a complex experiment with supersonic combustion in Mach 10 conditions. This process involves accounting for all the constraints resulting from the scientific goals, the facility and the techniques used for flow visualization. It also aims to demonstrate how CFD can be used as an invaluable tool throughout the design process.

Design methodology

The aim is to design an experimental model to be used in the T4 Reflected Shock Tunnel, operating at Mach 10 conditions, to investigate the regimes of supersonic turbulent combustion in a scramjet engine through the use of the OH-PLIF visualization technique. One very important constraint of the design is that it must be a simplified scramjet engine, representative of a complete one but without the complex geometry features present in engines such as the REST configuration [8]. The REST engine, with its elliptical combustor and injection scheme, has very complex flow structures inside the combustor. It is also very difficult to obtain optical access at the elliptical section. Opting for a 2D geometry for the model, the design of the engine starts by defining the experimental conditions, shown in

Enthalpy	MJ/kg	4.8
Pressure	Pa	640
Temperature	K	210
Density	kg/m^3	0.010
Velocity	m/s	3000
Mach number		10.4

Table 1: Mach 10 conditions used as basis for the model design [7].

table 1, taken from previous Mach 10 conditions used in the T4 Reflected Shock Tunnel [7].

Preliminary Design

To obtain combustor conditions of no more than $p = 50$ kPa and around $T = 1000$ K (sufficient to autoignite H_2 but without excessive shock compression losses), equilibrium air oblique shock relations [14] were used to design a 2D geometry based on the conditions in table 1. This results in the engine shown in figure 1a with a 6° fore-body and a compression ramp with an angle of 7° in relation to the flow (13° to the horizontal). The engine is also symmetric due to size restrictions in the test section.

To simplify analysis and visualization of the flow, a single injector is placed on one of the sides of the engine, on the engine centre line. This leads to a single fuel plume, with combustion restrained to the regions close to the centre plane of the engine. To avoid side-wall effects from affecting the fuel plume, the combustor was designed with a width of 80 mm (compared to its 20 mm height). Finally, to guarantee that fuel penetration is not enough for the fuel to interact with the opposite side of the engine, the mass flow of injected fuel is kept low, ranging from $\phi = 0.10$ to no more than $\phi = 0.25$. This means the engine has a mass flow rate of fuel equivalent to that of one injector in a full engine with multiple injectors. The geometry is shown in figure 1a.

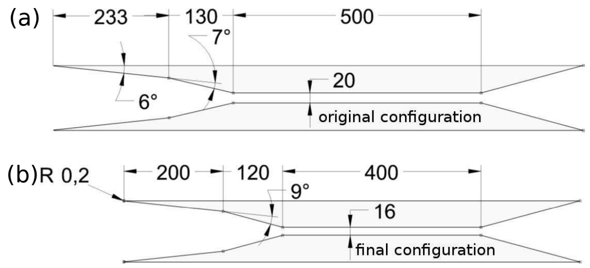

Figure 1: Two-dimensional geometry of a) the original and b) final configuration. Dimensions are in mm.

Flow visualization

Flow visualization techniques are used to allow observation of the flow inside the engine, which can give more information on flow structure and complement data gathered with sensors in the engine. Two techniques will be used in the experiments, schlieren and planar laser-induced fluorescence of OH molecules (OH-PLIF). Schlieren is used to visualize density changes in the flow, which allows the visualization of shock waves in the engine. OH-PLIF uses a planar laser sheet tuned to excite OH molecules in the flow. The excited molecules fluoresce, and this fluorescence can be captured with an Intensified Charge-Coupled Device (ICCD) camera. This gives an instantaneous image of the OH distribution on a 2D plane in the flow [4]. OH is an important intermediate radical in hydrogen combustion. By visualizing its molecules in the flow, it is possible to observe the combustion process inside the engine.

CFD solutions

All simulations were made using the US3D solver, developed at the University of Minnesota, which solves the compressible Navier-Stokes equations using a cell-centred finite volume scheme [15]. The Spalart-Allmaras RANS turbulence model was used, which is a one-equation model developed for aerospace applications [18], as implemented in US3D with corrections [1, 5]. US3D has been used extensively and successfully to simulate many hypersonic conditions and experiments within the T4 shock tunnel [2, 3, 11].

A grid analysis was made with several cell densities, settling on a grid with approximately 25 million cells for a domain without nozzle, and starting upstream from the injector. The inflow of this domain was taken from a separate simulation of the fore-body.

CFD simulations are used to provide better understanding of the flow features in the engine, which is then optimized from these results. CFD is an invaluable tool in this optimization process. To verify if the model achieved the desired simplicity, simulations were checked for the presence of any side-wall and edge effects that could interact with the fuel plume. To avoid edge effects from being ingested by the combustor, the fore-body was designed wider than the rest of the engine, with the side-walls starting only at the compression ramps. Having the side-walls shorter also lessens the impact of its corner vortices, which is also mitigated by having a wide engine. The CFD simulations confirmed the design was successful in minimising these effects, with none of them reaching the fuel plume at the centre of the combustor. This allows the fuel plume and combustion to develop unhindered from these secondary effects.

The results of the simulations of the original geometry with a 7° compression ramp configuration showed that it could not provide enough compression to achieve autoignition of the hydrogen fuel, therefore being unsuitable for the experiment. Compression in the engine had to be increased, which was done by increasing the angle of the compression ramps incrementally by 1°, first up to 8° and then to 9°, with CFD simulations run for each case. For 9º ramps, the simulated final combustion efficiency reached 54%, meeting our target. Table 2 shows the averaged results at a cross-wise plane 50 mm into the combustor for these three conditions, all fuelled with the highest equivalence ratio of $\phi = 0.25$. While temperature levels are similar, there is a considerable increase in pressure on the order of 30%. This difference is enough to achieve autoignition of the fuel. Note that, on the change from 7° to 8°, the lengths of the fore-body and compression ramps were changed. This modified the flow-field around the throat slightly by displacing the shock wave interactions in this region. The final geometry, with 9° compression ramps, is shown in figure 1b.

		7°	8°	9°
Pressure	Pa	22900	27300	33000
Temperature	K	937	950	1044
Density	kg/m^3	0.078	0.090	0.095

Table 2: Conditions at a cross-wise plane 50 mm into the combustor for each iteration of the model design.

Figure 2 shows the CFD results for two fuelling conditions, $\phi =$ 0.10 and 0.25. The simulations used a symmetry boundary condition through the injector. In this plane numerical schlieren is shown to emphasize the shock structures, as well as H_2 and

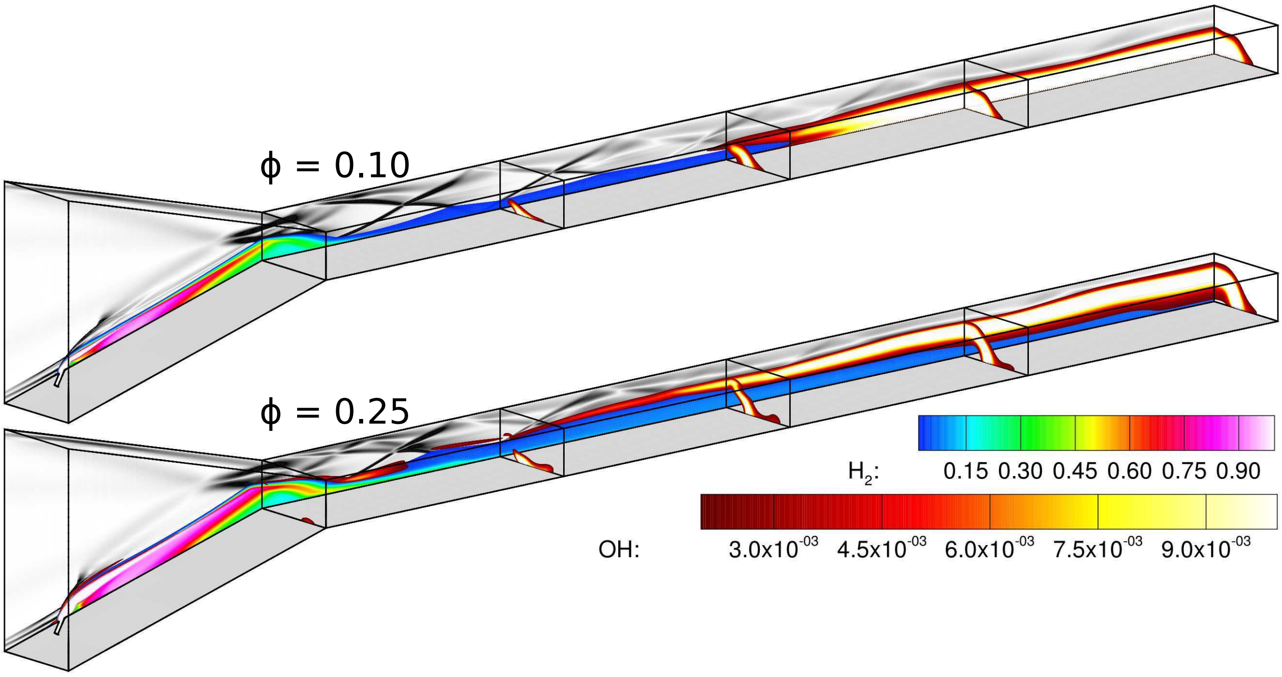

Figure 2: CFD results of the final model with $\phi = 0.10$ and 0.25.

OH concentrations. The cross-planes show OH concentration only. The complex shock structures at the throat are clearly visible and are affected differently by the penetration of the fuel in the two distinct cases. OH is visible at the borders of the fuel plume on the symmetry plane, with the higher equivalence ratio indicating OH production at the bow shock of the injector. It can also be seen that there is significant OH production away from the centre plane on both cases. These regions are good targets for OH-PLIF. The results indicate it is desirable to target off-centre to capture these structures, which confirms the need to have optical access to multiple streamwise planes in the combustor. The production of OH in the injector is also interesting, prompting the need for optical access to this region as well. The shock structures around the throat, on the other hand, are good targets for schlieren imaging. Overall, these results provide good insight into what to expect of the behaviour of the flow field in the experiment and results in a more informed planning of the experiment as a whole.

CAD design

With the CFD results providing better understanding of the flow, confirming the choice of geometry, the next step is to design the model hardware. The design constraints must also be taken into account as they lead to restrictions in the design. The need for optical access for the laser from the top, and the camera from the side, means windows must be placed at both locations. These windows serve as the top of the combustor and one of the sidewalls, and must interface with all the parts accordingly. This is achieved through a lateral support that holds both windows at 90° to each other, and can be removed for easy access to the windows. Since the windows are shorter than the combustor length, and access is needed to all regions of the combustor, they must be able to be moved to different locations. The design was made such that, with the same parts, the window support can be put at either side of the combustor, granting optical access everywhere. Since the laser position is fixed axially to the position of the top window on the test section, the model can be moved back and forth to expose different parts of the combustor to the laser. This requires flexible fuel lines into the test section, and the support is made so that the sensor cables are always shielded inside it.

The model is shown in figure 3 in an isometric view, with the closest side-wall removed to show the internal flow path of the engine. Flow is from left to right as indicated. On the top side of the engine there is a window, through where the laser sheet impinges on the flow. The fluorescence is observed through the side window, not seen in the image, which provides line of sight access for the camera. The injector is indicated, positioned on the inlet compression ramp in the bottom half of the engine, where pressure transducers and thin-film heat gauges are also present.

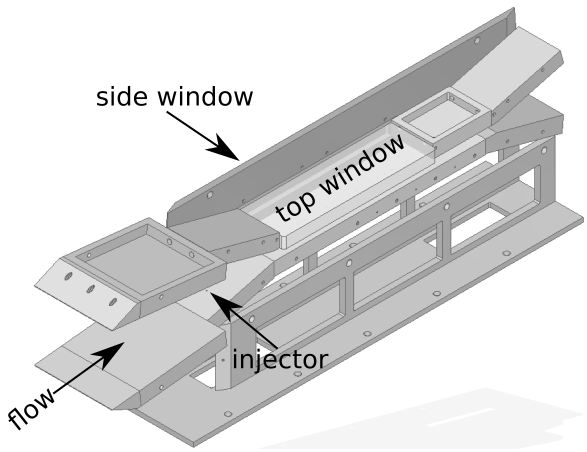

Figure 3: CAD design of the engine model.

There are also restrictions in the size of the test section and, specially, in the coreflow of the Mach 10 nozzle [8] that the model must be checked against. The flow outside the coreflow is highly non-uniform and its ingestion by the engine can compromise the experiment. It is important, then, that the captured air is inside the coreflow. This was verified by placing the CAD model of the engine inside the CAD model of the test section of the tunnel. This test section model includes the nozzle with a geometric representation of the coreflow as a surface. This enables verification of the position of the intake area with respect to the coreflow. Figure 4 exemplifies the model in the test section for one of the experimental conditions. This verification helps ensure that the model fits the coreflow at every desired location, while reducing the relative movement between model and nozzle exit. This also ensures that a successful experiment will be completed within shot-to-shot variations experienced in the tunnel. A Pitot survey is planned for the Mach 10 nozzle ahead of the experiments to verify the coreflow properties at the desired planes where the engine will be located.

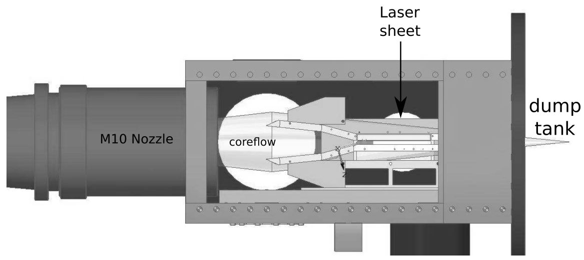

Figure 4: The model positioned on the test section of the T4 tunnel.

Conclusions

This paper demonstrated the process and final results of designing an experimental model for a complex experiment aimed at investigating supersonic turbulent combustion. Tight and often opposing constraints must be taken into account to inform the design decisions. CFD simulations, used from the start of the process, prove to be an invaluable asset in designing the flowpath and helping set the requirements.

Acknowledgements

This research was undertaken with the assistance of resources from the National Computational Infrastructure (NCI), which is supported by the Australian Government, and by resources provided by the Pawsey Supercomputing Centre with funding from the Australian Government and the Government of Western Australia. The authors would also like to acknowledge CNPq for the financial support through the Science Without Borders program of the Brazilian government under process number 200515/2014-4.

References

[1] Allmaras, S.R., Johnson, F.T and Spalart, P.R., Modifications and Clarifications for the Implementation of the Spalart-Allmaras Turbulence Model, *Seventh International Conf. on Computational Fluid Dynamics (IC-CFD7)*, 2012, 1–11.

[2] Barth, J.E., Wheatley, V. and Smart, M.K., Effects of hydrogen fuel injection in a Mach 12 scramjet inlet, *AIAA Journal*, **53**, 2015, 2907–2919.

[3] Barth, J.E., Wise, D.J., Wheatley, V. and Smart, M.K., Tailored Fuel Injection for Performance Enhancement in a Mach 12 Scramjet Engine, *20th AIAA International Space Planes and Hypersonics Systems and Technologies Conference*, 2015, 1–37.

[4] Boyce, R.R., Mudford, N.R. and McGuire, J.R., OH-PLIF visualization of radical farming supersonic combustion flows, *Shock Waves*, **22**, 2012, 9–21.

[5] Catris, S. and Aupoix, B., Density corrections for turbulence models, *Aerospace Science and Technology*, **4**, 2000, 1–11.

[6] Cocks, P.A., Dawes, W.N. and Cant, R.S., The Influence of Turbulence-Chemistry Interaction Modelling for Supersonic Combustion, *AIAA 2011-306*, 2011, 1–12.

[7] Doherty, L.J., *An Experimental Investigation of an Airfram Integrated Three-Dimensional Scramjet Engine at a Mach 10 Flight Condition*, PhD Thesis, The University of Queensland, Brisbane, Australia, 2014.

[8] Doherty, L.J., Smart, M.K., and Mee, D.J., Experimental Testing of an Airframe Integrated 3-D Scramjet at True Mach 10 Flight Conditions, *19th AIAA International Space Planes and Hypersonic Systems and Technologies Conference*, **53**, 2014, 1–18.

[9] Drummond, J.P., Diskin, G.S. and Cutler, A.D., Fuel Air Mixing and Combustion in Scramjets, *38 AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, 2002, 1–16.

[10] Gehre, R.M., Wheatley, V. and Boyce, R.R., Combustion Regimes in Inlet-fulled, Low Compression Scramjets, *20th AIAA International Space Planes and Hypersonic Systems and Technologies Conference*, 2015.

[11] Gehre, R.M., Wheatley, V., Boyce, R.R., Peterson, D.M. and Brieschenk, S.,Reynolds-Averaged Navier-Stokes and Wall-Modelled Large-Eddy Simulations of Sonic Hydrogen Injection into Hypersonic Crossflow, *18th Australian Fluid Mechanics Conference*, 2012, 7–10.

[12] Ingenito, A. and Bruno, C., Physics and Regimes of Supersonic Combustion, *AIAA Journal*, **48**, 2010, 515–525.

[13] Klimenko, A.Y., Multicomponent diffusion of various admixtures in turbulent flows, *Fluid Dynamics*, **25**, 1990, 327–334.

[14] Moura, A.F. and Rosa, M.A.P., A Computer Program for Calculating Normal and Oblique Shock Waves for Airflows in Chemical and Thermodynamic Equilibrium, *22nd International Congress of Mechanical Engineering*, 2013, 2323–2333.

[15] Nompelis, I., Drayna, T. and Candler, G., Development of a Hybrid Unstructured Implicit Solver for the Simulation of Reacting Flows Over Complex Geometries, *34th AIAA Fluid Dynamics Conference and Exhibit*, 2004.

[16] Peters, N., *Turbulent Combustion*, Cambridge Monographs on Mechanics, Cambridge University Press, 2000.

[17] Pope, S.B., Computations of turbulent combustion: Progress and challenges, *Symposium (International) on Combustion*, 1991, 591–612.

[18] Spalart, P. and Allmaras, S., A one-equation turbulence model for aerodynamic flows, *AIAA-92-G439*, 1992.

[19] Thornber, B., Bilger, R.W., Masri, A.R. and Hawkes, E.R., An algorith for LES of premixed compressible flow using the Conditional Moment Closure model, *Journal of Computational Physics*, **230**, 2011, 7687–7705.