COMPUTATIONAL AND EXPERIMENTAL STUDY ON SUPERSONIC FILM COOLING FOR LIQUID ROCKET NOZZLE APPLICATIONS

by

Vishnu VIJAYAKUMAR [a*], Jagadish Chandra PISHARADY [b], and Ponniah BALACHANDRAN [b]

[a] National Institute of Technology (Calicut), Kozhikode, Kerala, India
[b] Liquid Propulsion Systems Centre, Valiamala, Thiruvananthapuram, Kerala, India

Original scientific paper
DOI: 10.2298/TSCI120908077P

An experimental and computational investigation of supersonic film cooling was conducted on a subscale model of a rocket engine nozzle. A computational model of a convergent-divergent nozzle was generated, incorporating a secondary injection module for film cooling in the divergent section. Computational fluid dynamics simulations were run on the model and different injection configurations were analyzed. The computational fluid dynamics simulations also analyzed the parameters that influence film cooling effectiveness. Subsequent to the computational fluid dynamics analysis and literature survey an angled injection configuration was found to be more effective, therefore the hardware was fabricated for the same. The fabricated nozzle was later fixed to an Air-Kerosene combustor and numerous sets of experiments were conducted in order to ascertain the effect on film cooling on the nozzle wall. The film coolant employed was gaseous nitrogen. The results showed substantial cooling along the walls and a considerable reduction in heat transfer from the combustion gas to the wall of the nozzle. Finally the computational model was validated using the experimental results. There was fairly good agreement between the predicted nozzle wall temperature and the value obtained through experiments.

Key words: *thrust chamber, nozzle, liquid rocket engine, supersonic film cooling*

Introduction

Film cooling is defined by Goldstein [1] as "The employment of a secondary fluid injected through discrete slots to insulate thermally a solid surface from a gas stream flowing over it is called film cooling". From this definition, it is clear that film cooling introduces a secondary fluid into the primary flow stream in order to decrease the heat transfer rate from the primary flow stream to the wall. With the wall surface temperature at a lower level, less expensive materials can be used in structural fabrication. Film cooling can be used on blades of gas turbines, scramjet intake surfaces, combustor walls of high-speed vehicles, rocket nozzles and the extension surfaces of rockets all of which usually work under high heating loads. The configuration and method of injection of secondary fluid is the most important parameter to achieve effective surface cooling. The angle of injection of the coolant gas to the mainstream can have two direc-

* Corresponding author; e-mail: vfoorvishnu@gmail.com

tions; an angle with respect to the axis and an angle in the azimuthal direction. Performance is usually quantified by referring to film-cooling "effectiveness" which can be defined as:

$$\eta = \frac{T_{\mathrm{aw}}(x) - T_{\mathrm{c}}}{T_{\mathrm{cool}} - T_{\mathrm{c}}} \tag{1}$$

where η is the adiabatic film cooling effectiveness, T_{aw} – the adiabatic wall temperature, T_{c} – the hot gas temperature, T_{cool} – the coolant temperature, and x – the distance downstream of injection.

For rocket applications film cooling is an effective cooling method, where one of the propellants either in liquid or gaseous form is injected through discrete holes. The films formed along the walls protect the surface from the hot combustion products. The effectiveness of film cooling depends on various factors like the injection angle, uniformity of the film formed, injection velocity and there by the boundary layer formation. In a typical liquid rocket engine the combustion products is the primary flow and the film coolant flow which interacts with the primary is the secondary flow. Primary flow is subsonic in the combustion chamber and supersonic in the divergent section of the nozzle. For nozzle divergent cooling, film coolant must also be in supersonic condition in order to avoid any flow disturbances. This concept is termed as Supersonic Film cooling (SFC). In supersonic flows, unlike subsonic flows, there are greater chances of shock formation which can result in the breaking of the coolant boundary layer. Therefore SFC should account for the effect of shock waves, thereby making the design of an effective film cooling configuration all the more challenging. Here, film coolant is introduced downstream of the nozzle throat *i. e.*, in the nozzle divergent and the injection is through a separate module. Moreover, in this case of injection at the point of impingement static pressure of both the streams needs to be matched in order to have minimum disturbance of the primary flow due to secondary injection [2]. Secondary injection is through the periphery of the nozzle. The inlet condition of inject ant, location of injection, direction of injection, *etc*. are all variables in the process. Mixing process of the two flow streams in the vicinity of the injection point, the mode of heat exchange, disturbance of core flow, *etc*. needed to be studied in detail before going in for the experiments. Earlier studies of Schuchkin *et al*. [3] and Hatchett [4] provide an insight into the development of the computational model and the subsequent experimentation procedure. The overall objective of the present effort was to create an experimentally validated computational tool for simulating SFC, which can be used to shorten the actual engine design cycle.

Computational study

The computational model was adopted from work previously done at LPSC [5]. The changes incorporated include the combustion pressure, the area ratio at which secondary injection was initiated and the boundary conditions. In the present work experiments were planned using an air-kerosene combustor which has a maximum working pressure of 10 bar and therefore the secondary injection module was suitably altered in order to by-pass the effects of flow separation. The secondary injection should be initiated well before flow separation occurs. Figures 1 and 3 show the geometric model of the hardware generated using AutoCAD.

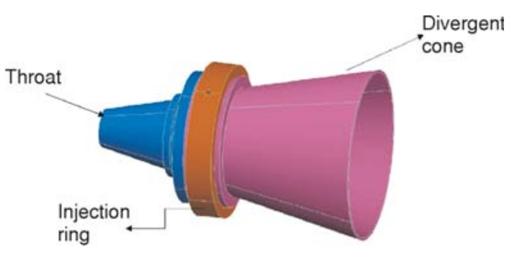

Figure 1. Nozzle divergent

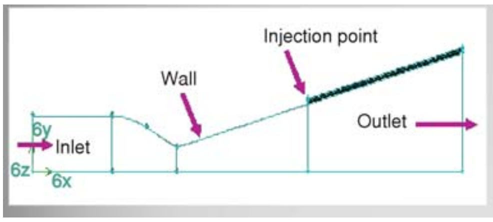

Figure 2. Computational model for analysis

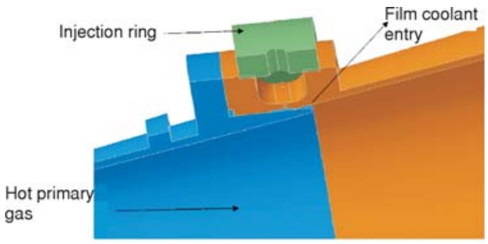

Figure 3. Sectional view of injection module

The 2-D computational model is shown in fig. 2. During the preliminary design phase different types of film laying techniques from open literature have been investigated. Mainly two types of injection techniques were analysed: parallel and angled injection. Computational fluid dinamics (CFD) simulations showed better wall cooling for the angled configuration. The design chosen, which has undergone several design reviews, is characterized by its simple and flexible design. Figure 3 shows the mode of secondary injection along the wall of the nozzle divergent (angled). The present CFD analysis using FLUENT software predicted the mixing pattern of two supersonic flow streams (primary flow and secondary flow),distribution of temperature, velocity, pressure, and Mach number in the vicinity of injection region and downstream of the injection point. Gambit preprocessor was used for meshing the 2-D geometry of the nozzle. The total configuration was divided in to various segments like cylindrical portion, convergent, throat region, *etc*., for easy meshing. Quadrilateral element was used for meshing. The spacing for the nozzle section was selected as 0.25 mm and that of injection point was finer with 0.01 mm spacing, with total numbers of nodes as 361938. An axisymmetric flow model was considered for the analysis. Spalart Allmaras turbulence model was employed because the time taken for convergence was lesser and the results did not show a significant variation from that of the k-ε and k-ω models. The boundary condition for the analysis is shown in tab. 1.The same computational model was later employed to simulate the experiment conditions, and the predicted results were compared with the experiment results.

Table 1. Boundary conditions for initial analysis

Inlet	Injection point	Outlet	Wall
P = 10 bar M = 0.7 kg/s T = 1258 K	P = 1.03 bar M = 0.04 kg/s T = 303 K	P = vacuum T =300 K	No slip, Adiabatic

Experimental study

The experimentation process involved design of related hardwares, setting up of test facility, conducting qualification test and finally the hot fire test.

Test facility

The software validation laboratory of LPSC at Trivandrum, India, has been chosen as the test bed for conducting the supersonic film cooling experiments .The test facility features propellant tanks, feed lines, combustor, nozzle assembly, cooling system for combustor/nozzle, igniter system and gas pressurization systems. A detailed layout of the test facility is shown in fig. 4. The combustor was designed to withstand a maximum pressure of 10 bar and temperature of 1500 K for a maximum flow rate of 0.7 kg/s. It contains a port each for air, kerosene, and LPG

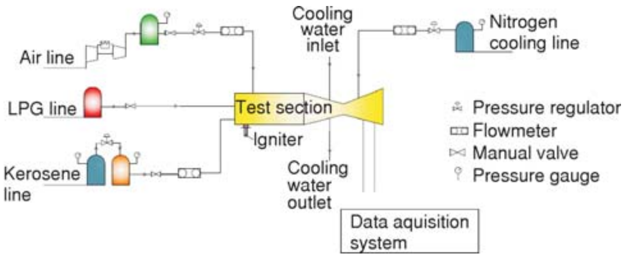

Figure 4. Schematic of test facility

injection along with an integrated igniter system. A disassembled view of the combustor is shown in fig. 5.

The nozzle designed has an inlet diameter of 70 mm, throat diameter of 28 mm and exit diameter of 150 mm.The point of secondary injection is at an area raio of 4, based on the design criterion for no flow separation. The convergent, throat, and divergent sections were fabricated separately using SS304 after which they were tungsten electrode inert gas welded into a single structure incorporating water cooling at selected portions. The nozzle components and assembly is shown in figs. 6 and 7. In order to ascertain the complete effect of film cooling the nozzle divergent was made devoid of any other cooling systems, whereas part of the combustion chamber and the nozzle convergent are water cooled.

Figure 5. Combustor

The propellant feed lines consisted of an air and a kerosene line. Compressed air was stored in 1000 litre capacity tanks and the required flow was attained by means of a regulator. Kerosene flow was regulated by means of a pressurant gas (nitrogen in the present case), together with a regulator. The film coolant was introduced in the nozzle divergent from a cylinder of 60 litre capacity and tank pressure of 120 bar. The coolant flow was adjusted by means of a pressure regulator. The ignition system consisted of a transformer having an output voltage of

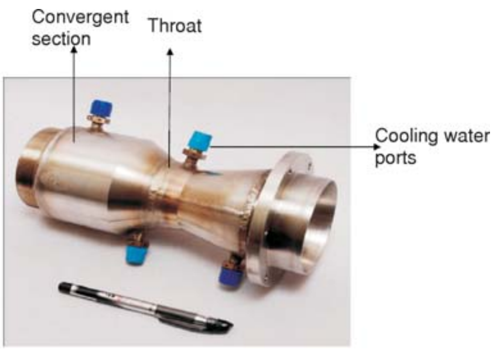

Figure 6. Convergent cone

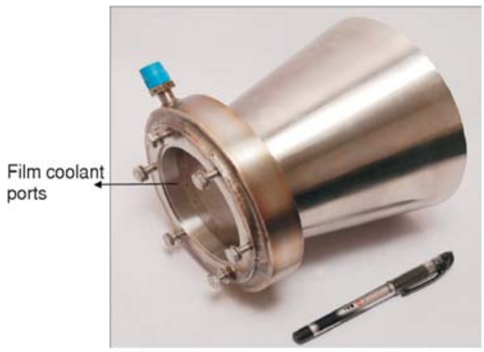

Figure 7. Divergent cone with injection module

30000 volts. Cooling water was supplied to the combustor and selected portions of the nozzle from an overhead tank. The data acquisition system was a Yokogawa make data logger which periodically sampled D. C. voltages or analog input such as thermocouples and RTD. Since the ignition temperature of kerosene -air mixture is around 210 °C; LPG was employed initially to start the combustion process.

Measurement scheme

The experimental set-up was equipped with standard instrumentation required to characterize the operation of the chamber and nozzle. Orifice meter was employed to measure air flow. Kerosene and film coolant flows were measured using rota meters. A pressure transducer (Macurex make) was mounted at the nozzle entry in order to characterize combustion chamber pressure oscillations and also to ascertain the pressure of hot gases entering the nozzle. In order to characterize the effectiveness of the film applicator downstream of injection, K-type thermocouples were installed at specified axial distances in the nozzle divergent. Pressure gauges were installed at specified locations in the propellant lines in order to assess the test condition. The measurement plans for the tests are as shown in tab. 2.

Table 2. Details of measurements

Pressure	Air tank
	Kerosene tank
	Gaseous nitrogen tank
	Chamber pressure (one channel)
Temperature	Nozzle body temperature (14 points)
	Exhaust gas temperature (during trial)
Mass flow rate	Air mass flow rate
	Kerosene mass flow rate
	Gaseous nitrogen mass flow rate

Test plan and first results

Experimental approach

The operation of a film cooling system is affected by a number of independent parameters. While the combustion chamber pressure takes direct effect on the location of injection, the mixture ratio influences the film indirectly by hot gas temperature and heat transfer. The cooling mass flow directly affects the film temperature. The film parameters define the initial conditions of the film at entry into the nozzle divergent. Mass flow and the film inlet geometry take direct influence on the entry state of the film. The operation of the film applicator hardware and the influence of the relevant parameters are investigated in a step-by-step approach. Cold flow tests were followed by the characterization and qualification of the film applicator hardware and the measurement equipments. Prior to the conduction of hot fire tests; trial runs were conducted in order to see the sustainability and stability of combustion at off nominal mixture ratios. Subsequent to the trail runs, the injection pressures required for getting the rated flow rates of the fuel and oxidizer were calculated and respective tank pressures were set to get the required chamber pressure. Owing to the maximum temperature limit of SS304 the temperature of the hot gases were limited to 1000 °C. For safety reasons the experiments were planned for a combustion chamber pressure of 3 bar initially and in future the pressures can be gradually increased to a maximum of 10 bar provided, the hardware is not at any risk of getting damaged. The injection pressure of air flow line was set at 3 bar, and kerosene line at 4 bar in order to achieve an air flow rate of 25.12 g/s and kerosene flow rate of 1.32 g/s. The coolant gas was set to a pressure of 2 bar having a corresponding mass flow rate of 4.8 g/s. Hot test was done in 2 steps, with and without film cooling. The thermocouple positions for wall temperature measurement is shown in fig. 8.

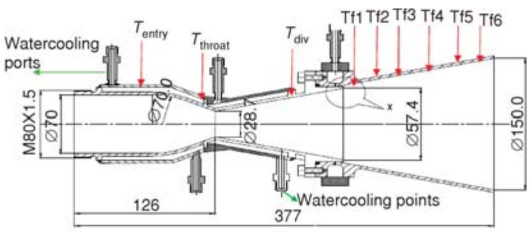

Figure 8. Thermocouple position; X – secondary injection module

The positions of thermocouples were altered and repositioned in order to capture the exact behavior of the film coolant. The present positions are liable for change under higher working temperatures and pressures. The total test duration was nine minutes, which allows sufficient time for the stabilization of combustion. In order to study the exact film cooling effect the nozzle divergent was made free of any other cooling mediums. The convergent section, throat section and part of the divergent are water cooled segments .Therefore in such an arrangement the thermocouples placed at the convergent section Tentry(entry level temperature) and throat section Tthroat (temperature of throat) were able to give steady-state temperature readings. The thermocouples placed downstream of film coolant injection namely Tf1, to Tf6 were in a transient state during the entire test period, as that section of the nozzle was unable to reach thermal equilibrium with the surroundings. The total test time could not be increased further owing to the safety of the hard ware. The hot fire tests were repeated in order to confirm the accuracy of the measuring instruments. The designated path of the film coolant is shown by arrows in fig. 9.

Figure 9. Film cooling along the wall

On completion of experiments, the data was analyzed for both set of tests. In open literature, the non-dimensional adiabatic cooling effectiveness η is widely used for investigating and evaluating the effectiveness of film cooling as shown in eq. (1). The local temperature difference between the adiabatic wall temperature T_{aw} and the temperature of the hot gas T_{c} is compared to the difference of film coolant injection temperature T_{cool} and the temperature of the hot gas. However, this approach has several disadvantages. The use of this definition is limited due to the transient nature of the tests performed. In the current application, the steady-state adiabatic wall temperatures would exceed safe operating temperatures for stainless steel and any transient adiabatic wall temperature cannot be measured, since the film cooling will always be combined with different cooling techniques like capacitive or convective cooling. Therefore, a different approach to describe the film cooling effectiveness as proposed by Christoph *et al*. [6] is favored:

$$\Theta(x) = \frac{T_{\text{wall, without film}}(x) - T_{\text{wall, with film}}(x)}{T_{\text{wall, without film}}(x) - T_{\text{cool}}} \tag{2}$$

Key experimental results

Post qualification and trial runs, the nozzle hardware was fixed to the combustor assembly for the hot fire tests. Steady-state operating conditions were achieved only at the water cooled segments, whereas at locations devoid of any water cooling (nozzle divergent) continued to be in transient state during the tests. More number of thermocouples are affixed immediately downstream of the injection point. The distance between successive thermocouples increases in the axial direction downstream. The best method to evaluate the wall temperature measurements and therefore film cooling efficiency in the transient tests is arguable. Although the comparison of the maximum wall temperature is possible, it was found that for the verification of the spatial characteristics of the film cooling effectiveness there need to be more number of thermocouples attached downstream of film applicator. For proper estimation of the effect of film coolant at the respective wall positions, hot fire tests were done in two steps; with and without film cooling. After completion of each set of hot fire tests the thermocouple readings at each of the axial positions were analyzed and thereby giving a clear picture of the effect of coolant. Special attention was required when interpreting the hot wall temperature data. Due to the high sensitivity of the thermocouple measurements to inaccurate mountings, to small gaps between sensor and chamber, to defects caused by thermal or mechanical overloads in the wall as well as due to the different signal delay times, the data always has to be checked for plausibility. The thermocouples T_{entry}, T_{throat}, and T_{div} showed almost identical profiles for the two sets of tests. In order to analyze the effect of film cooling temperature, readings of thermocouples Tf1 to Tf6 were compared for the two test cases *i. e*., with and without secondary injection. Temperature profiles for Tf1 and Tf6 are shown in figs. 10 to13 .The time shown in graphs is the actual test time.

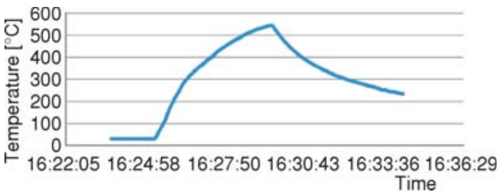

Figure 10. Tf1 without film cooling

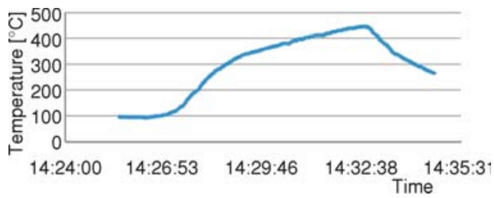

Figure 11. Tf1 with film cooling

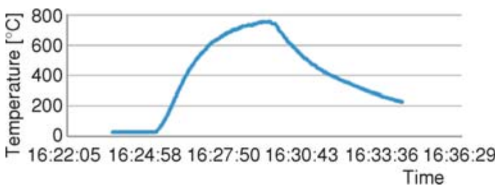

Figure 12. Tf6 without film cooling

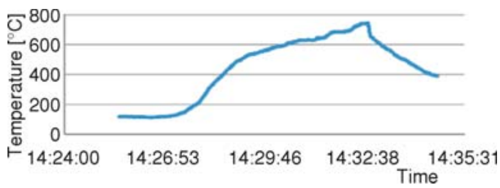

Figure 13. Tf6 with film cooling

It is clear that the highest temperature point is well above 500 °C in the absence of any film coolant. A dip of nearly 100 °C was observed owing to secondary injection.

It is evident from the graph that as the distance increases from the injection point the film cooling effectiveness correspondingly decreases. Data shown in tab. 3 corresponds to tem-

Table 3. Temperature distribution along axial length

Thermocouple	Axial length	Temperature without film cooling	Temperature with film cooling	Effectiveness
	[mm]	[°C]	[°C]	
Tentry	34.00	99.39	97.43	
Tthroat	121.00	41.09	42.00	
Tdiv	187.00	224.96	228.53	
Tf1	241.00	544.26	446.66	0.19
Tf2	247.00	566.95	485.00	0.15
Tf3	257.00	608.78	524.64	0.15
Tf4	277.00	714.48	654.63	0.09
Tf5	307.00	750.21	720.70	0.04
Tf6	337.00	759.85	745.38	0.02

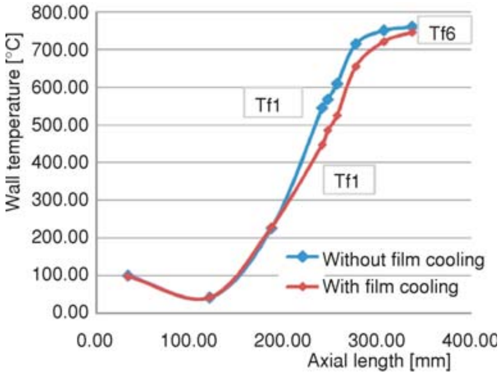

Figure 14. Temperature distribution along axial length

perature variation along the axial direction for the two test cases. The table shows the axial positions at which the thermocouples were placed .The distances were measured starting from the convergent section of the nozzle .The temperature readings correspond to test timings of 16:29:28 and 14:32:45, respectively. Effectiveness was calculated using (2).

Evaluating the thermocouples Tf1 and Tf6 from fig. 14, it was clear that the film cooling effectiveness gradually diminished downstream from the point of injection.

Comparison of experimental results and CFD prediction

CFD prediction

Suitable changes in the boundary conditions were incorporated in the computational model so as to match the actual test conditions. The boundary condition changes were: (1) inlet pressure of hot gas was changed to 3 bar and temperature o 1273 K, (2) outlet pressure was set to 1 atm, because test was carried out at ground conditions, and (3) mass flow rates of primary and secondary flow were changed based on experimental values.

The CFD simulation predicted the flow characteristics is shown on figs. 15 and 16.

The trends suggested that the effect of film coolant injection was limited to a small region. There was no continuous film formed on the wall surface suggesting extreme mixing of the two flow streams. Owing to the high temperature of hot gas, film seemed to be breaking downstream.

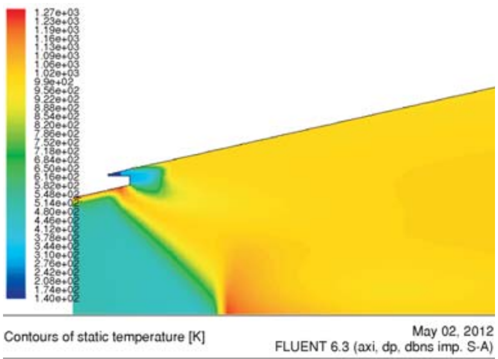

Figure 15. Contours of static temperature

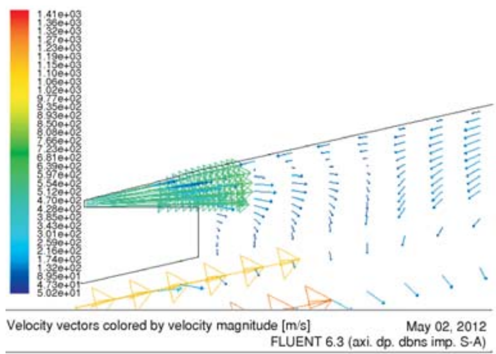

Figure 16. Contours of velocity vector

From fig. 16 there was clear indication of flow re-circulation occurring along the walls which severely dented the possibility of the formation of a continuous film over the wall surface. The computational model predicted a significantly low wall temperature when compared to the experimental data as is clear from the data shown in tab. 4 and fig. 17.

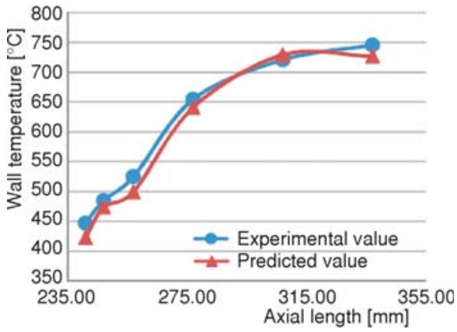

Figure 17. Simulation (predicted) and experimental results

Table 4. Data comparison

Thermocouple	Axial length	Experimental value	Software prediction
	[mm]	[°C]	[°C]
Tf1	241.00	446.66	422.64
Tf2	247.00	485.00	474.00
Tf3	257.00	524.64	499.14
Tf4	277.00	654.63	640.65
Tf5	307.00	720.70	729.00
Tf6	337.00	745.38	727.00

Conclusion and future work

The significantly higher temperatures obtained during experiment have been attributed to various factors like non-uniformity of the combustion process and the delay caused in initiating the secondary injection. On the whole a more sophisticated CFD model is proposed, incorporating additional features like thermal mass, chemical species interaction, *etc*.

Computational and experimental investigations were conducted in order to arrive at the key film cooling parameters and to develop and apply suitable CFD models to describe them. A film applicator has been designed and manufactured in order to experimentally validate the software predictions for supersonic film cooling. From the analysis it was observed that injection along the wall (angled injection) is better in establishing a more effective coolant layer than parallel injection. Hot tests were conducted on the designed nozzle after fitting it to an air-kerosene combustor. Due to the nature of the hardware, no steady-state operating conditions were achieved for the wall temperatures at locations downstream of film injection. It was found that the temperature measurements were highly sensitive to, an inaccurate mounting, to small gaps between sensor and chamber, to defects caused by overloads on wall as well as due to different signal delay times. Therefore experiments were repeated for same set of experiment conditions in order to avoid any plausibility. As demonstrated by the transient data, the effect of film was felt, but only for a small length. On the whole, a stable experimental set-up for carrying out film cooling experiments was established and a preliminary investigation into the effects of film coolant was done. The present CFD model needs modifications so as to incorporate real time experiment conditions.

Within the PRS group several studies both experimental and numerical are being carried out, related to the development of an effective configuration for supersonic film cooling. This paper summarizes the efforts taken to investigate the key film cooling parameters and to develop and apply suitable models to describe them. From these tests, information on heat flux density, length of the film and the film, cooling effectiveness beneficial for verification of the film cooling models are expected which could be useful in testing the film effectiveness for higher working chamber pressures and temperatures.

Nomenclature

M – mass flow rate [kgs^{-1}]
P – pressure, [bar]
T – temperature, [K]

Greek symbols

γ – ratio of specific heat
η – adiabatic film effectiveness
ρ – density, [kgm^{-3}]

Θ – film cooling effectiveness

Acronyms

LPG – liquefied petroleum gas
LPSC – liquid propulsion systems center
RTD – resistance temperature devices
SS304 – stainless steel

Subscripts

aw – adiabatic wall
c – combustion /hot gas
cool – coolant
SFC – supersonic film cooling
x – axial distance

References

[1] Goldstein, R. J., Wilson, D. J., Effect of Film Cooling Injection on Downstream Heat Transfer Coefficients in High Speed Flow, *J. Heat Transfer 95* (1973), 4, pp. 505-509
[2] Peng, W., Jiang, P.-X., Influence of Shock Waves on Supersonic Film Cooling, *Journal of Spacecraft and Rockets, 46* (2009), 1, pp. 67-73
[3] Schuchkin V., *et al*., Mixing and Film Cooling in Supersonic Duct Flows, *International Journal of Heat and Mass Transfer, 45* (2002), 22, pp. 4451-4461
[4] Hatchett, J. H., An Investigation of Effectiveness of Normal and Angled Slot Film Cooling in a Transonic Wind Tunnel, M. Sc. thesis, Virginia Polytechnic Institute and State University,Va., USA, 2008
[5] Ramesh, G., *et al.*, 'Mixing/Interaction Study of Supersonic Flow Streams in Rocket Engine Nozzle Cooling, *Proceedings*, 25th NCAE 2011, Birla Institute of Technology, Ranchi, Bihar, India
[6] Kirchberger, C., *et al*., Investigation on Film Cooling in a Kerosene/GOX Combustion Chamber, 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 2009, Denver, Col. USA, Paper number AIAA 2009-5406
[7] Sloop, J. L., Kinney, G. R., Internal Film Cooling of Rocket Nozzles, NACA RME8A29A, 1948
[8] Morrell, G., Investigation of Internal Film Cooling of 1000 Pound Thrust Liquid Ammonia Liquid Oxygen Rocket Engine Combustion Chambers, NACA, RME51E04, 1951
[9] Abramson, A. E., Investigation of Internal Film Cooling of Exhaust Nozzle of a 1000 Pound Thrust Liquid Ammonia Liquid Oxygen Rocket, NACA RME52C26, 1952
[10] Kinnney, G. R., Internal Film Cooling Experiments with 2 Inch and 4 Inch Tubes and Gas Temperatures to 2000 °F, NACA RME52B20, 1952
[11] Pai, B.R., Whitelaw, J. H., The Influence of Density Gradients in the Effectiveness of Film Cooling, Aeronautical Research Council, London, 1968
[12] Ramose, G., *et al*., Hydrogen Gas Injection on a Turbulent Supersonic Air Flow, 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 2002, Indianapolis, Ind., USA, Paper number AIAA 2002-3881
[13] Martelli, E., Thermo-Fluid-Dynamics Analysis of Film Cooling in over expanded Rocket Nozzles, 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 2006, Sacrmento, Cal., USA, Paper number AIAA 2006-5207
[14] Morris, C. I., Ruf, J. H., Validation of Supersonic Film Cooling Modeling for Liquid Rocket Nozzle Applications, 46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, 2012, Nashville, Tenn., USA, Paper number AIAA 2010-6657

Paper submitted: September 9, 2012
Paper revised: January 14, 2013
Paper accepted: June 1, 2013