INVESTIGATION OF THERMAL PERFORMANCE OF ACTIVE COOLING CONFIGURATIONS WITH REGENERATIVE AND EFFUSION COOLING FOR HIGH SPEED COMBUSTION CHAMBER

Siva Karthik C V S S[1], Santhosh Kumar N[2], T Kishen Kumar Reddy[3], C. Chandrasekhar[4],
R.K Sharma[5]
[1.2.3.4]Mechanical Engineering Department, Jawaharlal Technological University, Hyderabad,
[5] Defense Research Development Organization, Kanchanbagh, Hyderabad
Email:[1]seva.karthik@gmail.com, [2]sknagula1991@gmail.com , [3]reddykishen@yahoo.com,
[4]chandraaditya@yahoo.com, [5]rksharma_hstdv@drdl.in

Abstract— The combustor liners of high speed combustion chamber are subjected to high thermal loads. Active cooling of such liners is seen as a viable option and research in this area is currently underway in many countries due to the advantages it offers. The main method of heat transfer is the regenerative cooling, where in coolant is passed through a channel inside the combustor. But, the configuration of such liners has to be optimized in terms of providing desired cooling efficiency with low mass flow rate of the coolant, so as to reduce the overall weight per unit area. For the current investigation rectangular channel is chosen. In this context, various configurations are verified with the help of MATLAB program and suitable configurations are simulated using ANSYS Fluent. It has been found that width of the channel has a profound effect on the heat transfer rates. As the width decreases the desired cooling efficiency is achieved at low mass flow rates of the coolant. In addition a comparative study is made between standalone regenerative cooling with that of coupled cooling strategy of regenerative and effusion cooling / film cooling. Effusion / full film coverage cooling has been the state-of-art cooling technology for the modern combustor liners in aero-engines, due to its advantages to mitigate the combustion instabilities and provide better thermo-acoustic performance. In the coupled strategy a series of holes are provided on the combustion side the coolant channel, to allow the coolant to effuse through the holes, thus forming protective layer on the surface. In the simulations that were carried out it was found that the coupled cooling strategy is effective to obtain the desired cooling efficiency at low mass flow rates than when compared to stand alone regenerative cooling. Thus meeting the performance requirements with reduced overall weight.

Index Terms— active cooling, film cooling, high speed combustion chambers, regenerative cooling

Introduction: Active cooling of high speed combustion chamber is being pursued as a viable option by many researchers. The primary reason for such a pursuit being the advantages offered by active cooling, which are not possible with other methods of cooling. The main advantage is to avoid carrying additional coolant on board which will add to weight. The use of Hydrocarbon fuels as coolant offer the additional heat sink due to their property of cracking, in which long chain molecules are broken into smaller ones, thus aiding the combustion in high speed flows inside the combustion chamber. But the major challenge being the configuration of the actively cooled heat exchanger design, which can withstand the high thermal and mechanical load with low weight per unit area. Due to the complex nature of the heat transfer and number of parameters that influence the process of active cooling, the design of such heat exchanger is a daunting task. Many proposals have been made, to arrive at a suitable configuration which can effectively cool the high heat fluxes encountered during the combustion process inside the combustion chamber.

Valdevit *et al.* [1] have shown that the geometry of the coolant channel, the thermal and physical properties of the coolant, combustion medium and material of the combustor and their interaction , conditions that are prevailing in the combustion chamber all influence the heat transfer rates. They have carried out parametric studies for different materials employing thermal barrier coatings, over a wide range of heat transfer coefficients on the combustion side and coolant flow rates inside the channel. Here the cooling strategy mainly focused on the usage of sensible heat to cool the channels and usage of Thermal Barrier Coatings (TBCs) to reduce the heat load reaching the surface of the panel.

Antonio *et al.* [2-3] studied effusion cooling with emphasis on the thermal effectiveness in multi-perforated plates for combustor liners of aero-engines, for different geometric and fluid dynamic conditions and found that the effective design leads to effecient thermal management.

Some other strategies include usage of endothermicity of the fuel as a heat sink but, that needs a thorough understanding of the fuel and its properties, and is also an area of research.

The purpose of the present paper is to study the ways to improve the heat transfer rates in a active cooling channel for a given thermal load, while minimizing the coolant flow rates requirement, thereby reducing the weight of the structure. In the first section of the paper regenerative cooling is studied and analyzed using 1D MATLAB analysis, validated by the 3D CFD analysis. In the next section, a coupled cooling strategy of regeneration and effusion / film cooling is used. Initial study is made to understand the effect of geometric parameters which can help reduce the coolant requirement for regenerative cooling and thereby reduce weight and to further extend to incorporate the effusion cooling (full-coverage film cooling) which is the state-of-art cooling technology for liners of the aero-engines, to the high speed combustion chamber in order to improve the heat transfer rates with minimum coolant flow rates.

The objective set above is achieved by study and comparison of three rectangular geometries channels with variable widths. The investigation is made first using MATLAB by adapting the approach of Valdevit *et al.* [1] to assess the influence of the geometrical parameters and then perform the CFD simulations to validate the same. The material of the coolant channel considered is Inconel X-750 and the fuel is JP-7. The results obtained above can be generalized for a wide variety of metal – fuel combinations.

The structure of the paper is as follows:

- Provide the description of the geometric parameters of the coolant channel.
- Present a study on the active cooling configurations by adapting Valdevit *et al.* [1] analysis using MATLAB program to arrive at three most suitable configurations of the coolant channel.
- Validate the above analysis with 3D CFD analysis using Fluent.
- Determine the important geometric parameters that have influence on the coolant flow rate requirements.
- Present a coupled cooling strategy of regenerative and effusion cooling followed by the results from CFD analysis of the coupled strategy.
- Discuss the effectiveness and the implications due to the coupled strategy.

Description of cooling channel configuration: The channel shape chosen for the purpose of active cooling is a rectangular one. The following is the description of the channel geometric parameters. The width of the internal flow area is denoted by ‘w’, height of the flow area is ‘l’, thickness of the face tf , thickness of the core is tc, total width of the channel is $b = w + tc$, total height of the channel is $H = l + 2tf$. Schematic of the cooling channel is provided in Fig. 1 for above geometric parameters. The following table summarizes the three suitable configurations studied.

Configuration	W	l	t_f	t_c
Type – 1	0.0025	0.005	0.0015	0.00125
Type – 2	0.002	0.005	0.0015	0.00125
Type – 3	0.0015	0.005	0.0015	0.00125

Note: All units in m.

The analysis is done for a single channel with the above parameters and length of the channel 0.5m. The inputs that are needed for the purpose are the realistic adiabatic wall temperature (Taw), wall temperature on the combustion side (Tw), heat transfer coefficient on both combustion side (hG) and the coolant side (hc), coolant flow rate per unit width (Veff), inlet temperature of the coolant (Tfuel^{inlet}) as encountered in scramjet experimental test conditions.

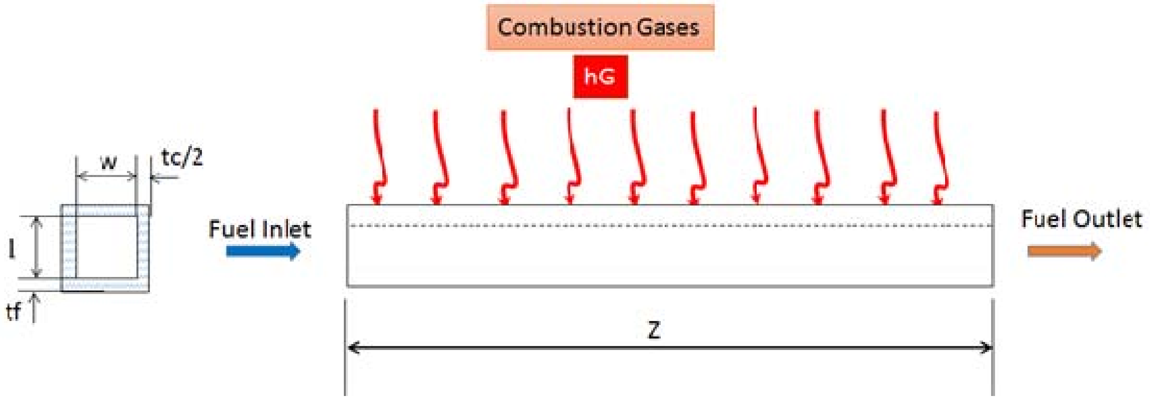

Fig.1 Cooling channel configuration diagram

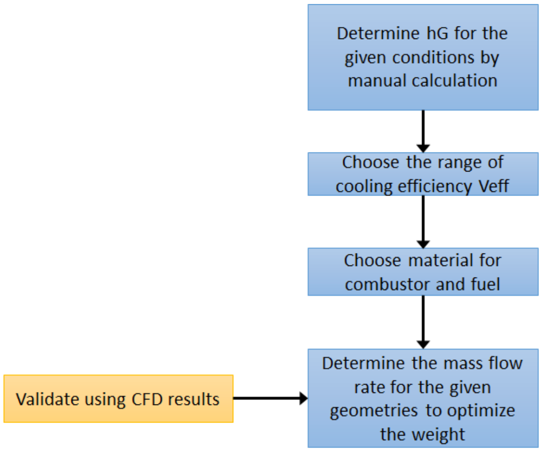

Fig. 2 Flow chart of 1-D Matlab program

The heat transfer coefficient on the combustion side hG is calculated using Eckert's Reference Enthalpy method. The conditions considered on the combustion chamber side are that of prevailing in the actual scenario as described in reference [1] with some variations. On the coolant side the inlet temperature is 400 K. For any given channel configuration, the mass flow rate required is calculated such that the temperature of the channel is within the material temperature limit for a given length of the channel (Z).

Analytical Model: MATLAB code is developed in line with the Valdevit *et al.* [1], which is used as a tool to arrive at suitable configurations. The Fig. 2 shows the flow chart for MATLAB program. For the given geometry and boundary conditions such as heat transfer coefficient of the coolant and combustion gases, the amount of the coolant mass flow required to keep the maximum temperature of the metal within the material temperature limit is obtained. Predictably maximum temperature occurs at the combustion side of the channel and hence this temperature is an important measure to check, in the design of a suitable configuration. **MATLAB Results:** Analytical results using MATLAB program for the three configurations are as follows, for the inlet temperature of the coolant of 400 K, heat transfer coefficient 701 W/m2K, adiabatic wall temperature Taw 3403 K.

Each of the configurations is checked for 3 different coolant mass flow rates. The mass flow rate of the coolant is derived from cooling efficiency Veff as described in Valdevit *et al.* [1].

Table I. Depicts the fuel mass flow rate and wall and fuel temperatures variations for various configurations.

Configuration	Mass flow rate Kg/s	Max Tfuel (K)	Max Twall (K)
Type 1	0.006	623	896
Type 1	0.0048	675	946
Type 1	0.0036	761	1026
Type 2	0.006	597	838
Type 2	0.0048	643	892
Type 2	0.0036	719	953
Type 3	0.006	569	779
Type 3	0.0048	609	818
Type 3	0.0036	676	895

The aim is to find the flow rate of coolant which will help maintain the maximum wall temperature around 900 K. This limit is chosen for MATLAB calculation, instead of the material temperature limit of 1100 K for Inconel, as in the 3D simulation the temperature raise will be more. Hence, this consideration will build in factor of safety, while arriving at the required mass flow rate. From the results obtained, it can be seen that decrease in the width of the channel decreases the amount of coolant required for cooling the channel to maintain the wall at the required temperature. Fig. 3 shows the comparison between the three configurations for the maximum temperature reached by combustion side of the metal and fuel, for different mass flow rates. From the chosen mass flow rates it can be is observed that the suitable mass flow rates of coolant for each of the configurations are 0.006 Kg/s for Type1, 0.0048 kg/s for Type 2, 0.0036 Kg/s for Type 3 configuration.

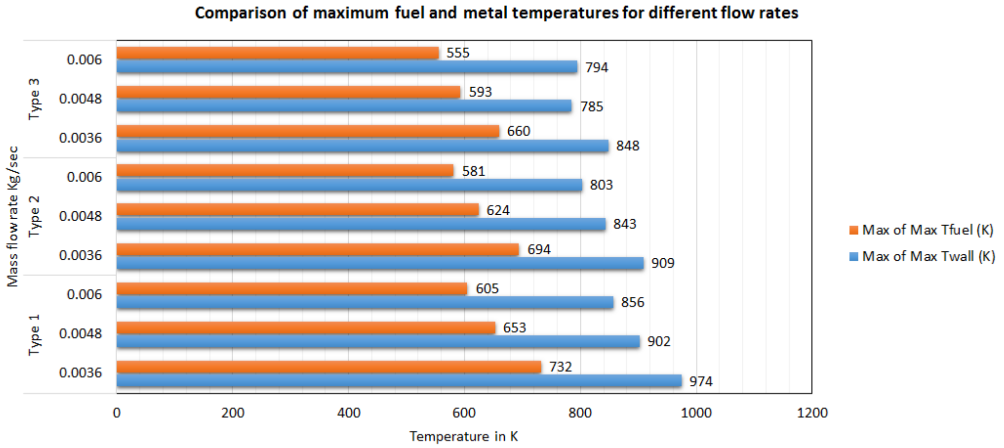

Fig. 3 **Comparison between the three configurations for the maximum temperature attained by combustion side of the channel and fuel, for different mass flow rates.**

Table II: Comparison between the highest material and fuel temperatures for 3D CFD and MATLAB

Configuration	Mass flow rate (kg/s)	Fuel Temperature (K)		Metal Temperature (K)	
		MATLAB	3D CFD	MATLAB	3D CFD
Type 1	0.006	623	560	896	1250
Type 1	0.0084	561	525	836	1100
Type 2	0.0048	643	590	892	1140
Type 2	0.006	597	550	838	1080
Type 3	0.0036	676	620	895	1140
Type 3	0.004	649	590	856	1060

The amount of coolant required for Type 2 and Type 1 configurations is in excess of the coolant required by Type 3 configuration by

34.5% and 66.7%, respectively; thus contributing to the decrease in the overall weight of the type 3 configuration. In the three cases studied Type 3 configuration with a decreased width of the coolant channel has a better performance as it is able to provide the required cooling efficiency at lower mass flow rate. Any further decrease in the channel width will be constrained by the manufacturing consideration. Hence, with all other dimension remaining constant, the width of the channel is found to play an important role in improving the heat transfer rate, while effectively keeping the metal temperature within the desired material limit. It is also found to reduce the amount of the coolant required thus contributing to the reduction in the weight of the structure.

Model Description:

To extend the analysis and validate the above results, 3D CFD analysis is performed in ANSYS Fluent 14.5. The three configurations of the coolant channels are tested with following boundary conditions.

In order to reduce the computation time, heat flux of 1600000 W/m^2 is applied on the face of the combustor side of the channel which is equivalent to the thermal load due to the hot gases having heat transfer coefficient hG, calculated using Eckert's Reference enthalpy method, as 701 W/m^{2}K with adiabatic wall temperature of 3403 K. Though the target length is 0.5, the simulation is carried over 1m length of the channel, as it is not possible to predict the exact mass flow rate which can meet the 0.5 m target length.

Boundary Conditions:

- Inlet Boundary condition:
    - Mass flow inlet
    - Temp = 400 K
    - Pressure = 3e6
- Outlet Boundary Condition:
    - Outflow
- Combustion side of the channel
    - Heat flux 1600000 W/m^2

Materials:

- Channel Material - Inconel X-750
- Coolant - JP-7

Turbulence Model: K-Є Turbulence model is used.

Type of analysis: Transient with pressure based solver

Results: The following section describes the results obtained from the CFD simulation.

Effect of width of the channel on the coolant mass flow rate: For the steady state condition, Table II provides the comparison between 3D CFD and MATLAB results. Considering the material temperature limit of Inconel X-750 to be 1100 K, the simulations are started with a mass flow rate determined in the MATLAB analysis for each of the configurations and varied till the maximum temperatures are below material temperature limit of 1100K. As can be seen from the results, that for the flow rates determined from MATLAB program are not sufficient for 3D simulation and that for 3D simulation, the flow rate has to be more than required, since the calculation in MATLAB is based on one-dimensional heat transfer and the variation between the 3D CFD and MATLAB is on the expected lines. The mass flow rates for CFD vis-à-vis MATLAB are higher by 26.667%, 20% and 11.11% for Type 1, 2, 3 configurations respectively. Though the estimation of the MATLAB has marked variation with those of the results from 3D CFD results, nonetheless, it serves as a starting point for the study. The trend in both 3D and analytical simulations shows that as the width decreases, the mass flow rate decreases. Thus demonstrating the effect of the width of the channel on the heat transfer and flow rates required to achieve the desired cooling efficiency. It also justifies the reason and explanation given above for choosing a lower temperature of 900 K as the material temperature limit, for MATLAB calculation.

Effect of width of the channel on the coolant heat carrying capacity: In Type 1 configuration, which has the largest width of the three geometries, the desired heat transfer rate can be obtained only through the increased mass flow rate of the fuel. Also, the rise in temperature of the coolant in middle of the channel is less in this case which can be observed from the graphs given in 4(a) (b) (c). It means that the coolant at the middle of the channel is not participating in the heat transfer process. This is attributed to the low residence time of the fuel inside the channel and not enabling the fuel to absorb the heat; while, the effectiveness of the channel to act as a fin has improved due to reduced gap between fins. This trend is following the fin analogy.

The phenomenon explained above can be visualized through the graphs shown the Fig. 4a, 4b, 4c. Here it can be observed that due to higher temperatures of the fuel achieved in Type 3 configuration, endothermicity of the fuel may come into play and may lead to higher heat sink capacity. So, decreased width of the channel can contribute to improved heat transfer rate by the way of the improved endothermicity thus protecting the structure from damage.

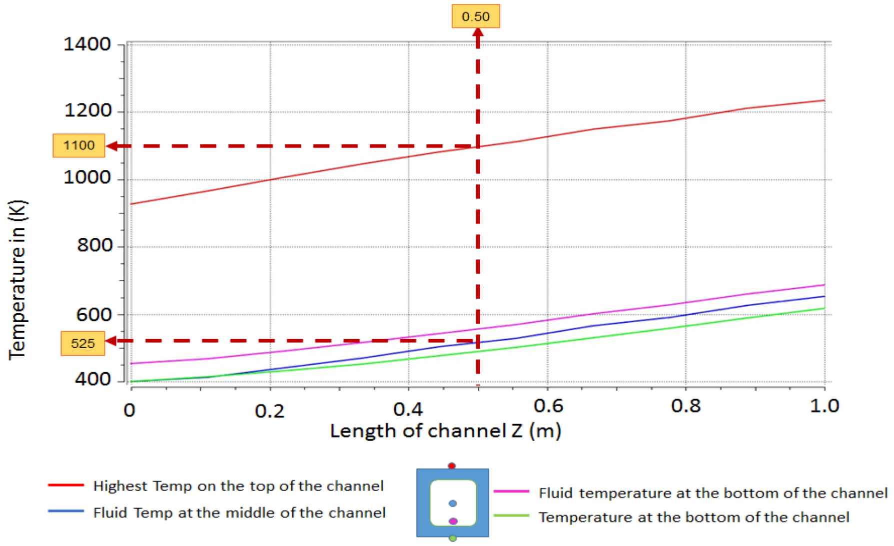

Fig. 4(a)

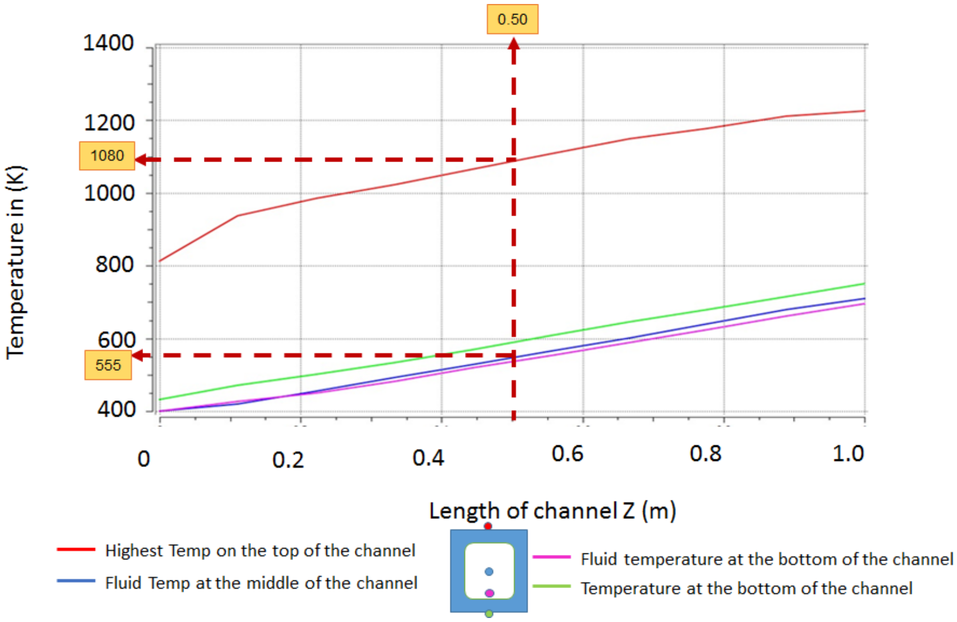

Fig. 4(b)

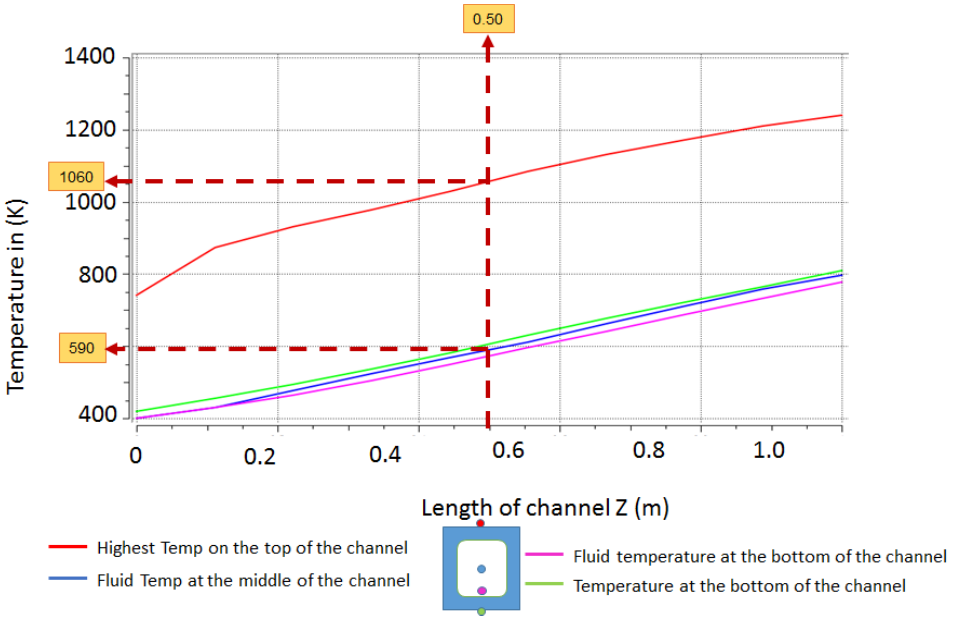

Fig.4 (a) (b) (c) Temperature distribution from CFD analysis of Type 1, 2, 3 configurations with regenerative cooling

Effect of width of the channel on the weight per unit area: The weight per unit area is an important parameter that needs to be optimized. The below section discusses the comparison in terms of weight per unit area (fuel + metal) for the three configurations.

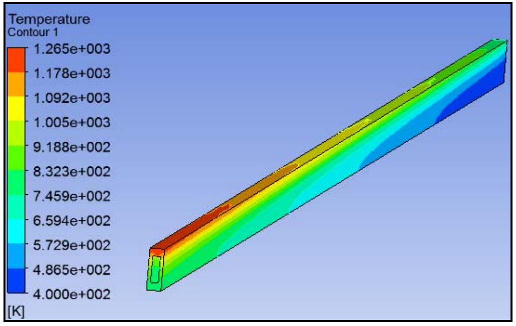

Fig. 5 Regenerative cooling temperature distribution over the length of the long channel

If a panel is considered of width say B. For the purpose of comparison let B be 0.1 m, then the number of channels of each type of configuration that fit in are N = B/b. The Fig. 6 explains the notations described above. The number will be rounded off to the nearest integer. Then, the weight of the panel (WPanel) can be calculated which is the volume times the density. The weight of the fuel (Wfuel) is the fuel required for the duration of operation, say 25 seconds. Then the total weight of the fuel required is mass flow rate times the duration.

The above two quantities are summed up to obtain the total weight (Wtotal = WPanel + Wfuel). For the comparison, the weight is measured in terms of weight per unit area, which is Wtotal / (B*Z).

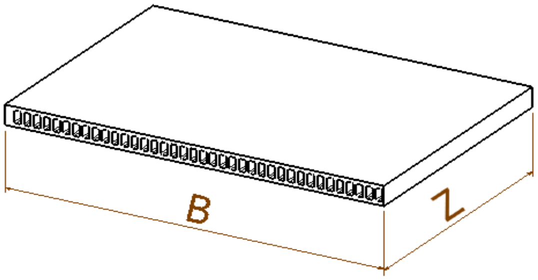

Fig. 6 Combustor panel

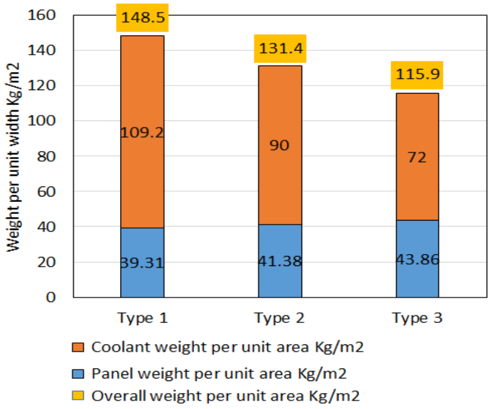

Fig. 7 Weight per unit area comparison for different configurations.

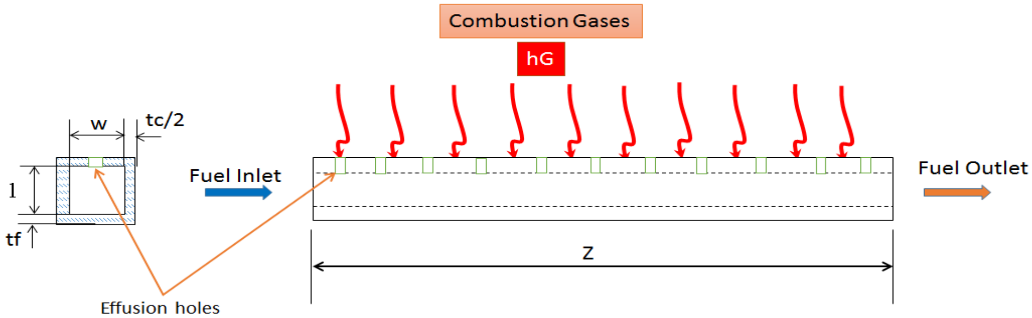

Fig. 8: Schematic of effusion cooling

Cooling strategy combining regenerative and effusion cooling: From the above results and analysis it can be observed that better heat transfer rates can be obtained if the coolant is allowed to absorb more heat. This can be accomplished by either increasing the residence time of the coolant inside the coolant channel or by bringing the colder coolant from the bottom of the channel towards the combustion side of the channel as much as possible to absorb the more heat per unit mass flow rate. In the above sections so far, regenerative cooling strategy has been used to cool the coolant channel and was found that increasing the mass flow rate may not be an efficient method. Now, in order to improve the efficiency of the heat transfer and to bring more coolant from bottom towards the combustion side of the channel, effusion cooling is proposed. In this the coolant is passed through a channel with series of holes drilled on the combustion the coolant channel facing the combustion side, so that as the coolant travels along the length, a portion of the coolant is effused through the holes. The coolant thus effused will form a protective layer on the channel and prevents the heat from reaching the surface of the coolant channel. This form of cooling through effusion is currently used in the combustor liners of aero-engines and is an area of interest for the researchers in this area. Effusion cooling is currently being pursued as an option for the aero-engine combustor liners due to its better thermo-acoustic performance [2]. In the hypersonic systems as well the use of such similar cooling strategy has been explored by many researchers and found that the cooling mainly depends on the number of holes, their spacing and the blowing ratios but still this subject is an evolving one.

Linn *et al.* [4] has performed numerical investigation of the effusion cooling and compared them with the experimental ones.

K.A. Heufer *et al.* [5] have studied film cooling for hypersonic flow conditions.

K.A Juhany *et al.* [6] have studied the influence of injectant Mach number and temperature on supersonic film cooling.

David E.Glass *et al.* [7] have performed numerical analysis of convection and transpiration cooling through the use of natural porosity of the CMC using Hydrogen as a coolant by using boundary layer code and porous media finite difference code.

Wolfgang Dahmen *et al.* [8] have performed 2D -numerical simulation of transpiration cooling through porous media using CMCs

S.Gulli *et al.* [9] have performed investigation of transpiration cooling effectiveness for air-breathing hypersonic vehicles.

But to the knowledge of the authors, it is not found in the open literature where in the coupled cooling of effusion and regenerative cooling method has been applied to the coolant channel. The purpose of the following section is to obtain a basic understanding of the performance of the coupled cooling strategy using the CFD analysis.

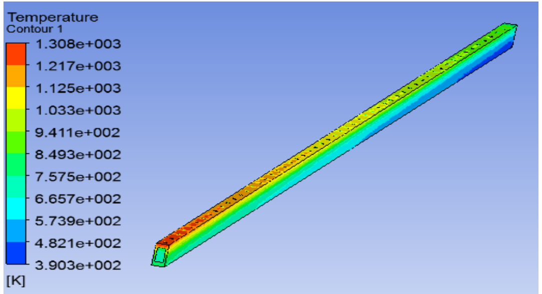

Fig 9: Effusion cooling of 1m channel

Model Description:

The Fig. 8 shows the schematic of the coolant channel with effusion cooling along with the regenerative cooling. CFD simulation is performed combining Transpiration and effusion cooling. Effusion holes are provided on the Type1 configuration and compared with the regenerative cooled configuration. The holes of diameter 0.001 m and with equal spacing of 0.02 m are provided on the face of the channel exposed to the combustion gases. At the effusion holes, mass flow outlet boundary condition with 15% mass outflow through the holes is imposed. The iterations are started with the mass flow rate of 0.006 Kg/s with an objective to maintain the temperature within the material temperature limit. Rest of the conditions are similar to those described above. It was found from 3D CFD analysis that at 0.007 Kg/s, desired cooling efficiency is obtained. Fig. 9 shows the graphical results of the temperature distribution along the length of the channel. Fig. 10 shows the 3D temperature distribution along the channel.

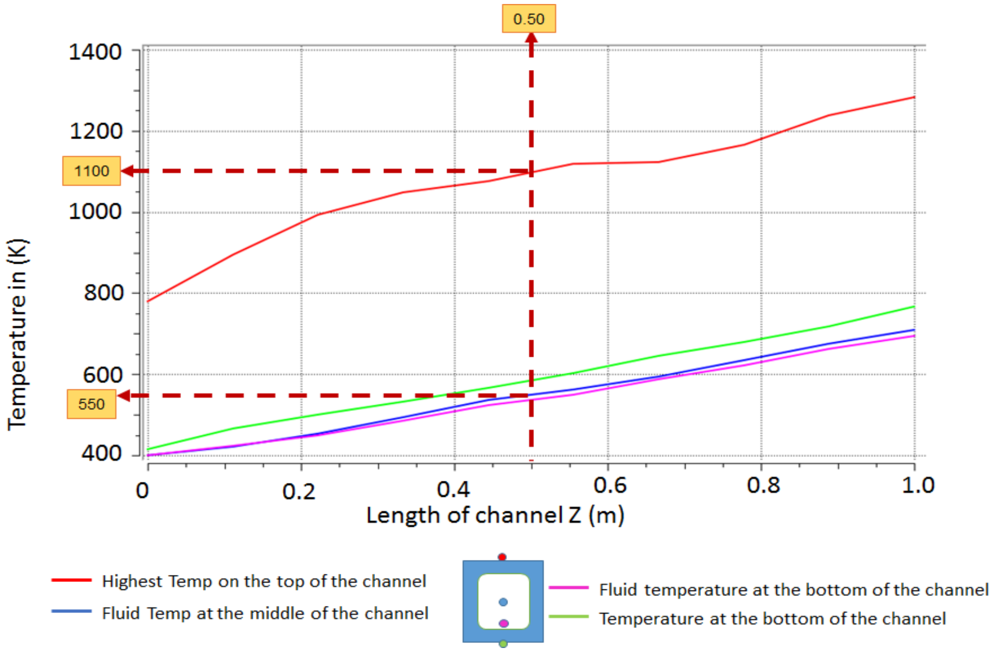

Fig. 10 The temperature profile along the length of the channel for effusion cooling

Results: The following section, discusses the comparative analysis between the two cooling strategies viz., regenerative alone and the coupled cooling for the model described in the above section.

Effect of mass flow rate:

1. For 0.006 Kg/s of the mass flow rate the highest temperature reached at 0.5 m length is 1250 K and 1150 K for regenerative and coupled cooling strategies, respectively. For the desired cooling efficiency to keep the material temperature limit of 1100 K, the coolant flow rate required is observed to 0.0084 Kg/s for the former and 0.007 Kg/s for the later.
2. In both cooling strategies, the channels reached highest temperature occurs at the outlet.
3. The steady state temperature in both the cases with and without effusion cooling is observed at the around 25 seconds.

Conclusions:

1. The coupled cooling strategy is found to be advantageous when compared to regenerative cooling as discussed below.

a. In terms of the coolant mass flow required to achieve the desired cooling performance the mass flow rate for coupled cooling required 20% less than the standalone regenerative one. This implies a considerable weight saving, as 20% less coolant needed to be carried on board.

In terms of overall weight (Wtotal), it can be observed from Fig. 11, that the coupled strategy provides 12 % weight saving compared to the regenerative one.

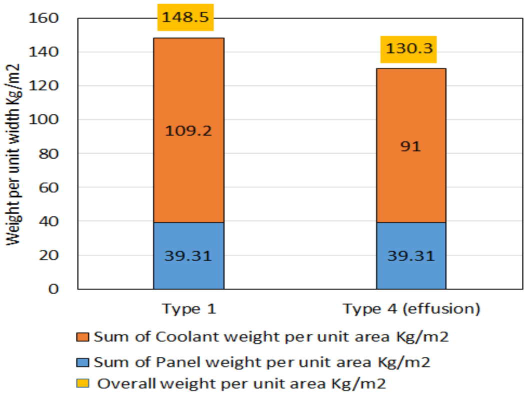

Fig. 11 Weight comparison between regenerative and effusion cooling

b. Due to the manufacturing constraints involved, the width of the channel cannot be reduced to extent required to improve the cooling efficiency. Hence higher mass flow rates are required to improve the cooling efficiency. With the coupled strategy and by providing suitable number of holes on the coolant channel the required cooling can be achieved at lower mass flow rates.

c. By providing the pores at intermediate distances along the channel the flow rate could be reduced, therefore the coolant residence time is increased and hence effectively utilizing the coolant heat sink capacity. The increase in the residence time also provides an opportunity to increase the contribution of the coolant to cool the structure by the way of increasing the endothermic capacity of the fuel at higher temperatures of the fuel.

Implications with effusion cooling

- The presence of pores can reduce the structural integrity and that needs to be verified. This will be part of the future work.
- The presence of fuel at higher temperatures can lead to coking and thus may block the pores. It is important that the temperature of the coolant does not exceed the coking temperature inside the pores.

The effect of the fuel effusion through the pores on the combustion is also an important aspect which need to be understood.

Nomenclature:

CFD – Computational Fluid Dynamics
CMC – Ceramic Matrix Composites

REFERENCES:

[1] Lorenzo Valdevit, Natasha Vermaak, Frank W. Zok and Anthony G. Evans, “A materials selection protocol for light weight actively cooled panels,” Journal of applied mechanics, vol. 75, pp. 061022-1 – 061022-15, 2008.

[2] Antonio Andreini, Bruno Facchini, Alessio Picchi, Lorenzo Tarchi, Fabio Turrini, “Experimental and theoretical investigation of thermal effectiveness in multiperforated plates for combustion liner effusion cooling,” Journal of Turbomachinery, vol.136, pp. 091003-1 - .091003-13, September 2014.

[3] Antonio Andreini, Riccardo Da Soghe, Bruno Facchini, Lorenzo Mazzei, Salvatore Colantuoni, Fabio Turrini, “Local Source Based CFD modelling of effusion cooling holes: Validation and application to an actual combustor test case,” Journal of Engineering for Gas Turbines and Power, vol.136, pp. 011506-1 – 011506-11, 2014.

[4] Linn, J., Kloker, M.J. “Numerical investigations of effusion cooling in hypersonic boundary-layer flow. In: New Results in Numerical and Experimental Fluid Dynamics VI (ed. S. Jakirlic, C. Tropea, H.-J. Heinemann, R. Hilbig), NNFM series, Springer,” 2007

[5] K.A. Heufer, H. Olivier, “FILM COOLING FOR HYPERSONIC FLOW CONDITIONS,” Proceedings 5th European Workshop on Thermal Protection Systems and Hot Structures Noordwijk, The Netherlands, 17 - 19 May 2006 (ESA SP-631, August 2006)

[6] K.A. Juhany, M.L. Hunt, J.M. Sivo, “Influence of injectant mach number and temperature on supersonic film cooling” , Journal of thermophysics and heat transfer , March -1994

[7] David E. Glass, Arthur D.Dilley, H. Neal Kelly, “Numerical anlysis of convection / transpiration cooling NASA / TM-1999-209828 ,” 1999.

[8] Wolfgang Dahmen, Thomas Gotzen, Siegfried Muller, “Numerical simulation of Transpiration cooling through porous material,” Bericht Nr. 374, Institut fur Geometrie und Praktische Mathematik, September 2013.

[9] S Gulli, L. Madddalena, S. Hosder, “Investigation of transpiration cooling effectiveness for air breathing hypersonic vehicles,” AIAA 2011.

[10] William H. Heiser, David T. Pratt, “Hypersonic Airbreathing Propulsion,” AIAA, 1994.

[11] Jerrold D. Wear, Arthur M. Trout, John M. Smith, “Performance of Semi-Transpiration Cooled Liner in High Temperature Rise Cobustor,”NASA Technical Paper 1806, 1981.