Application of transpiration cooling in a 2D-supersonic/hypersonic flow by injection over a flat plate

Sujan B Thapa*, Kalyan Deepak G, Suchet B Thapa

Abstract: The advancement in space exploration and aviation industry have made hypersonic flights such as scramjets to be imminent. Such flow regimes with high Mach numbers are characterized by high heat flux and wall temperatures where well designed thermal protection systems are crucial for avoiding material damage and assuring the structural integrity of the aircraft. This holds true for other applications as well such as cooling of re-entry vehicles, jets including rocket engines and much more. Till date, such concepts and applications have been at most incorporated for subsonic flow regimes such as cooling turbine blades. However, the vortical structures generated in such three dimensional cases reduces the average cooling effectiveness. Thus this research is designed to present an optimum cooling system by analyzing the interaction of a cold fluid injected into a flat plate supersonic and hypersonic boundary layer through the numerical simulations of the Naiver-Strokes equations. The interaction of cold fluid into a supersonic boundary layer over a flat plate through injections acts a promising concept for thermal protection systems in the design of innovative cooling systems. This is known as transpiration cooling. This cooling technique is adapted into an automatic code generation framework known as OpenSBLIFVM. In this work, the underlying cooling concept and its efficiency realization are briefly summarized and validated for the application at hand. In this study a numerical analysis of the transpiration cooling mechanism is presented on a two dimensional flat plate specimen with injection from the bottom surface to achieve cooling effect for flow in supersonic regime. Also, with the results obtained from the simulations an optimum angle injection was found to be 90°.

Keywords: Supersonic flow; Hypersonic flow; Subsonic flow; Scramjets; heat flux

I. INTRODUCTION

Most components especially the outer wall of atmospheric re-entry vehicles or supersonic space aircraft are partially exposed to severe aerothermal loads. Thus, components need sophisticated thermal protection to sustain very high temperatures. The peak heating rates generated in the combustion chamber of such aircraft during supersonic flights impose the most difficult requirement to accommodate for the thermal structural design of the engine. This is so because unlike in scramjets, the passive ablative thermal protection systems (TPS) cannot be used for such re-entry application due to inadequate reusability requirements, maintenance of precise geometric contours and significant ablative loss of combustor material [16], [8], [14], [6], [17].

Therefore, in order to reduce the wall heating effects, various active cooling methods can be adapted such as film cooling, convection cooling, or transpiration cooling. There has been extensive research over the past decade regarding such cooling concepts as well as the interaction of the injected cooling fluid with the main flow field [3], [5], [7]. However, these works have only been able to address practical applications such as turbine flows and subsonic studies. Only recently, studies and development on active cooling strategies for supersonic and hypersonic flow regimes have been explored. For instance, DNS of hypersonic boundary layer over a flat plate with different domain configurations by Adriano explains the cooling mechanism and the boundary layer interaction upon slot coolings and plenum chamber [1]. Similarly, film cooling for supersonic flows in scramjet applications has been studied and established that with the turbulent flows involved, tangential injections are the preferred choice [9]. Moreover, in summary, it is fair to state that there still lacks a severe understanding of the fundamentals of the flow field and thermal interactions.

In this study, we provide further exploration into one of such cooling techniques particularly transpiration cooling which consists of injecting cold fluid into the boundary layer through an adiabatic wall that is particularly used to reduce the wall heat flux. The reason being, this technique meets the reusability requirements as well as is suitable for continuous and repeated operation of supersonic vehicles. Also, it has been established that among the various cooling techniques stated, transpiration cooling offers the highest efficiency under the similar geometric specimens carried out in this study [2].

The basic principle of this cooling technique involves injecting a cold fluid, commonly air, through insertions or slots in the wall surface such that the heat load at the wall is reduced. This application has been adapted into the flat plate boundary analysis done in [12], [4] which employs OpenSBLIFVM numerical solver. This solver has the capability to automatically parallelise the code to various architectures like Central Processing Units (CPU's) using Sequential and Message Passing Interface (MPI) and using Compute Unified Device Architecture (CUDA) on the GPU's. The brief methodology of the discretisation schemes and their implementation are highlighted in section II. In section III the numerical setup, the injection criterion and the corresponding results haven been discussed. The conclusions drawn from the reuslts have been mentioned in section IV.

II. METHODOLOGY

In this work, the governing equations used are the two dimensional compressible Navier Stokes equation in their integral form. They are as follows:

$$\frac{\partial}{\partial t}\int_{\Omega}\vec{W}d\Omega + \oint_{\partial\Omega}(\vec{F_c}-\vec{F_v})dS = 0 \tag{1}$$

where, $\vec{W}$ is the conservative variables vector, $\vec{F_c}$ and $\vec{F_v}$ are the convective flux and the viscous flux vectors respectively.

$\vec{Q}$ is the source term.

$$\vec{W} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}, \vec{F_c} = \begin{bmatrix} \rho V \\ \rho u V + n_x p \\ \rho v V + n_y p \\ \rho H V \end{bmatrix}, \vec{F_v} = \begin{bmatrix} 0 \\ n_j \tau_{xj} \\ n_j \tau_{yj} \\ n_j \Theta_j \end{bmatrix} \quad (2)$$

where, ρ, u, v and p are density, components of the velocity vector ($\vec{v}$) and pressure respectively, E and H are internal enrgy and enthalpy of the fluid and V is contravariant velocity ($\vec{v}.\vec{n}$), i.e. the velocity normal to a surface element and is given by,

$$V = \vec{v}.\vec{n} = n_x u + n_y v \quad (3)$$

where, $\vec{n}$ (n_x, n_y) is the unit normal vector. The shear stress (τ_{ij}) is given by,

$$\tau_{ij} = \frac{\mu}{Re}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right) \quad (4)$$

where, Re is the Reynolds number. The equations are non-dimensionalised with the free stream values. The viscous heat dissipation and heat conduction terms are included in Θ, given by,

$$\Theta_j = u_i \tau_{ij} + \frac{\mu}{RePr(\gamma - 1)M_\infty^2}\frac{\partial T}{\partial x_j} \quad (5)$$

where, T, Pr, γ and M_∞ are the temperature, Prandtl number, ratio of specific heats and freestream Mach number respectively.

A. *AUSM*

In order to discretise the convective fluxes in the governing equation 2, Advection Upstream Splitting Method (ASUM) has been incorporated. This scheme was introduced by Liou and Steffen [10]. In this scheme, the convective flux is split into convective and pressure part given by,

$$\vec{F_c} = (M_N)_{i+1/2} \cdot \begin{bmatrix} \rho c \\ \rho c u \\ \rho c v \\ \rho H c \end{bmatrix}_{L/R} + \begin{bmatrix} 0 \\ n_x p \\ n_y p \\ 0 \end{bmatrix}_{i+1/2} \quad (6)$$

where, c is the speed of sound, $M_{N\,i+1/2}$ is the interface Mach number and is evaluated as,

$$(M_N)_{i+1/2} = M_L^+ + M_R^- \quad (7)$$

M_L^+ and M_R^- are the split Mach numbers.

The discritiztion of viscous terms is performed on the basis of Auxilary Cell Approach [12] implementation.

B. *OpenSBLIFVM Application*

OpenSBLIFVM [12] is an automatic code generation framework based on OpenSBLI [11] and Sympy [13] as its building blocks. This framework is developed using the Finite Volume Method, making use of the basic functionality present in OpenSBLI. The discretisation of the convective fluxes is implemented using various Flux Vector Splitting schemes such as ASUM, AUSM δ, AUSM$^+$ and Flux Difference Splitting Schemes Vanleer, Roe and HELLE with entropy fix terms respectively. This auto generated code framework coupled with the OPS [15] library can run on various architectures. In the present work, only CUDA version is used.

III. SIMULATIONS

A. *Numerical Setup*

A two-dimensional flat plate of domain 2 x 0.5 non-dimensional units in x and y direction respectively is subjected to a supersonic inflow of Mach 2.7. The left of the domain is set as inflow, top and right of the domain as outflow and bottom wall is an adiabatic wall. The mesh of sizes $180 \times 150, 240 \times 180, 300 \times 220$ have been considered with local refinement at inflow, the bottom wall and around the injection area. The mesh of size 180×150 with respective boundary condition is shown in fig. 1.

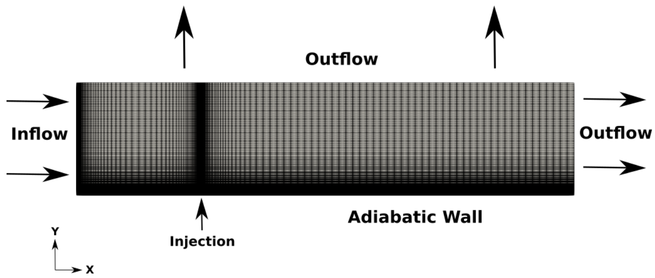

Figure 1: **Mesh of size** 180×150.

B. *Injection*

The setup is subjected to fluid same as inflow through a thin slot between the location 0.497 to 0.503. The injection is performed at varied angles of 90°, 60°, 45°, 30° with an sonic inflow velocity.

The simulations are performed with density (ρ) of 1.0 at Reynolds number (Re) 300000. The gamma (γ) value and reference temperature are set at 1.4 and 288.0 K respectively. Likewise, the Sutherland's constant (μ) and Prandtl number (Pr) are set at 110.4 and 0.72 respectively.

C. *Results*

To validate our results, we achieved grid independence relatively in the pressure and temperature domain among the different angles of injection as mentioned. The pressure plotted is the total pressure at the end of the flat plate along the y-direction, whereas the temperature is plotted across the bottom wall of the flat plate. It can be inferred from the fig. 2 and fig. 3 that grid size of 300×220 was able to capture the shock location and the upstream disturbances precisely comparatively than other grid sizes. The simulations were run until the Linf norm of $1 \times e^{-12}$ with a time step of 0.0001.

The fundamental aim of the experiment is to decrease the wall temperature. The line plots shown at fig. 4, displays the temperature fluctuation along the x-axis at different angles of injection. The plot exhibits a sharp temperature drop at the site of injection followed by a slight temperature rise lower than the initial temperature along the domain length. It

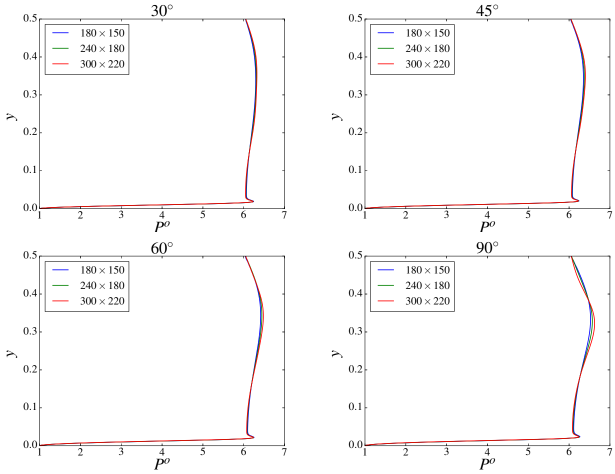

Figure 2: Grid Independence results of total pressure at outlet.

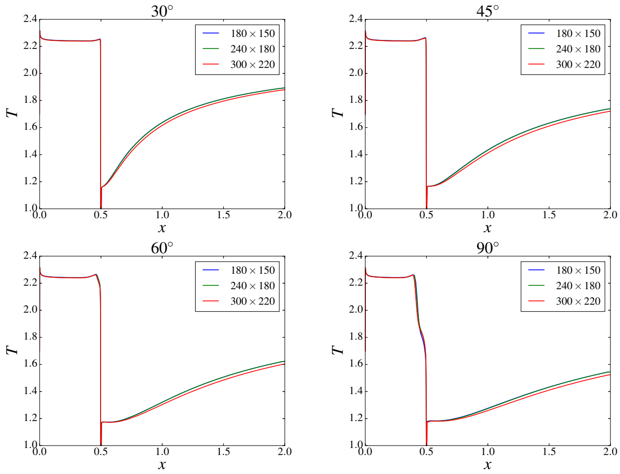

Figure 3: Grid Independence results of temperature at bottom wall.

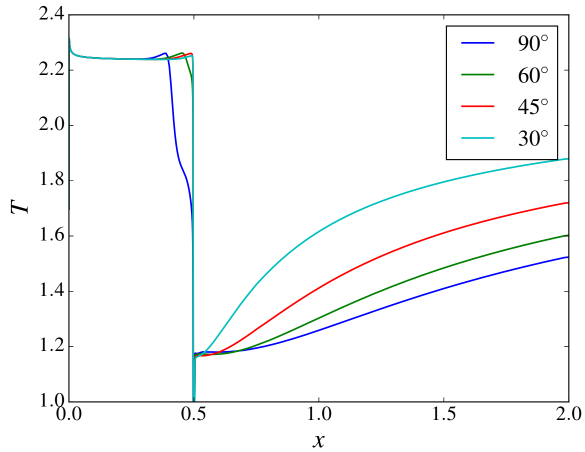

Figure 4: Temperature at bottom wall for various angles of injection 300×220 **grid size.**

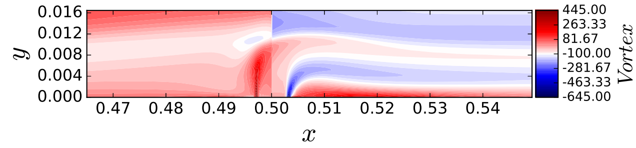

Figure 5: Vortex formation in 300×220 **grid for 90° angle of insertion.**

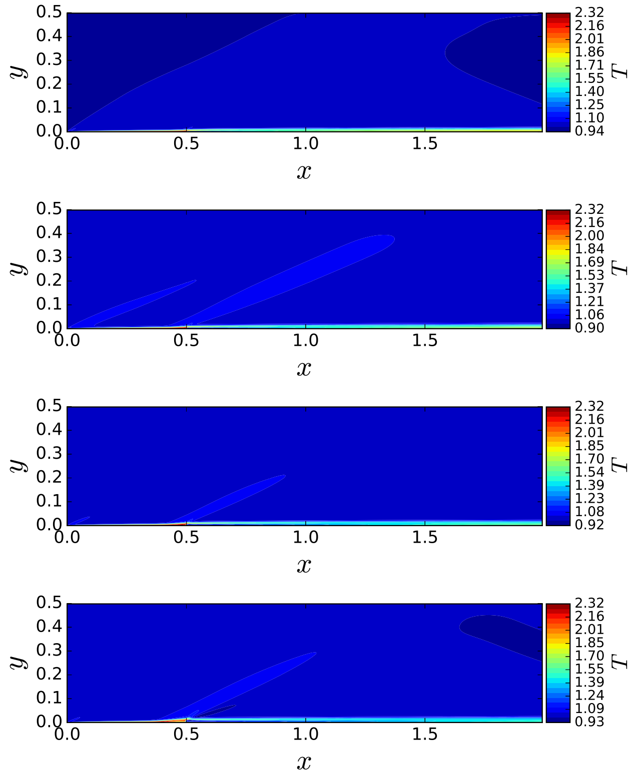

Figure 6: Contours of Temperature for grid size 300×220**.**

can be inferred from the plot that the maximum temperature difference can be achieved at an angle of 90°.

The solver has captured vortex formation as show in fig. 5 before and after the injection. The formation of the vortex before the injection is due to interaction of the inflow boundary layer with the injction. While the vortex after injection is due to the interaction of the injection with the boundary layer after flow seperation. The shock generation at the inlet and site of injection can be realized via the contour plots as shown in fig. 6.

The line plot shown at fig. 7 depicts the pressure behaviour at the outlet of the domain. The additional deflection in the pressure distribution at 90° angle is due to the insufficient domain length. The overall increase in the pressure at the outlet can be understood as an effect of the shock formation. The highest pressure alteration can be realized in the contour plot shown in fig. 8 (90° contour plot).

The cooling efficiency and the pressure losses realized from the above performed numerical simulations have been compiled in the table. 1. The final cooling effect is a result of the temperature difference between the free stream

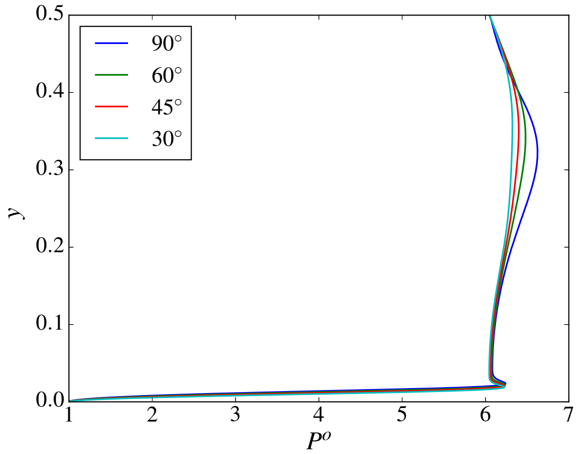

Figure 7: **Total pressure at outlet for various angles of injection for** 300×220 **grid size.**

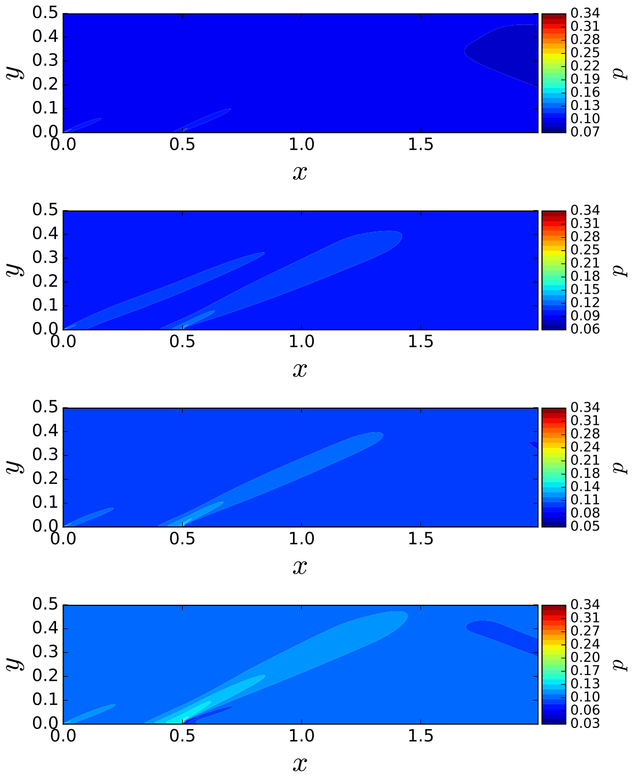

Figure 8: **Contours of pressue for grid size** 300×220.

temperature and the adiabatic wall temperature at the end of the flat plate. The pressure losses is calculated by $P_{of} - p_o$ where P_{of} is the free stream pressure and p_o is total pressure.

The iterations taken to converge at different angles of injection is shown in fig. 9. They are plotted for the grid size 300×220. We can notice that 90 takes 38,20,744 number of iterations to converge and 30 takes the least number of iterations of 23,27,915 for convergence.

Table 1: **Comparision between cooling effect and pressure losses**

S. No.	Temperature (%)	Pressure (%)
30°	18.83	3.17
45°	25.70	3.22
60°	30.78	3.27
90°	34.21	3.37

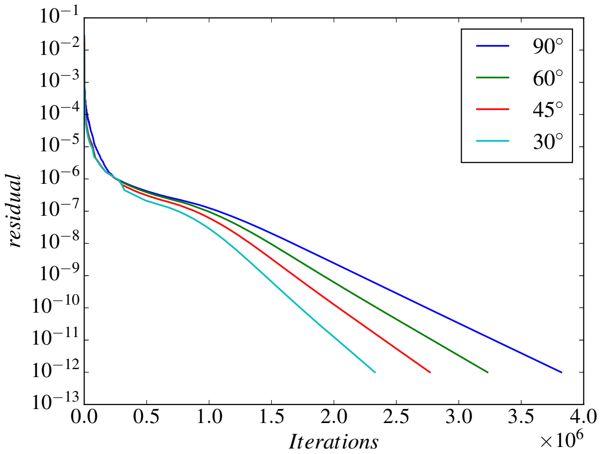

Figure 9: **Iterations taken to converge for different insetion angles.**

D. Computational Run Time

The simulations are run on NVIDIA DGX V100 workstation. The computational time taken for simulation is shown in table. 2.

Table 2: **Computational runtime (sec) for different angles of injection and grid sizes on GPU**

S. No.	30°	45°	60°	90°
180×150	465.8584	525.1016	620.1948	728.2115
240×180	561.0108	631.7047	758.2854	858.3668
300×220	662.7118	806.0729	923.8014	1075.0263

IV. CONCLUSIONS

Our numerical simulations confirm that transpiration cooling using injection is a promising strategy for supersonic flows as well. The presented investigations show that the AUSM scheme has been able to successfully capture the shock formations at the leading edge and the injection slot along with boundary layers.

The analysis revealed adequate temperature differences among the different angles of injection under study. The maximum cooling efficiency of $\approx 35\%$ has been achieved at $90°$ angle of injection but with a maximum pressure loss of 3.4% within the specified domain length. While the second best angle of injection was found to be $60°$ providing a moderate cooling efficiency of $\approx 31\%$ with a pressure loss of 3.3%.

In regards to the computational time, as it has been acknowledged that the simulation at the $90°$ angle of injection gives the best cooling effect however it also uses more number of iterations for convergence due to maximum flow obstruction. Hence higher computational time.

ACKNOWLEDGMENT

This work has been carried out in the frame of NVIDIA DGX V100 GPU. The authors would like to thank Mr. Raja Mangalagiri for helping in running the simulations on GPU and in understanding OpenSBLIFVM [12][4].

REFERENCES

[1] A. Cerminara, R. Deiterding, and N. Sandham. Direct numerical simulation of blowing in a hypersonic boundary layer on a flat plate with slots. In *2018 Fluid Dynamics Conference*, page 3713, 2018.

[2] E. R. G. Eckert and J. N. Livingood. *Comparison of effectiveness of convection-, transpiration-, and film-cooling methods with air as coolant*, volume 1182. National Advisory Committee for Aeronautics, 1954.

[3] A. Fitt, J. Ockendon, and T. Jones. Aerodynamics of slot-film cooling: theory and experiment. *Journal of fluid mechanics*, 160:15–27, 1985.

[4] K. D. G, S. B. Thapa, R. Mangalagiri, and S. P. Jammy. Performance of various shock capturing schemes on cpu's and gpu's. In *8th International and 47th National Conference on Fluid Mechanics and Fluid Power (FMFP)*, number FMFP2020-196, IIT Guwahati, Assam, India, December 2020.

[5] A. Gülhan and S. Braun. An experimental study on the efficiency of transpiration cooling in laminar and turbulent hypersonic flows. *Experiments in fluids*, 50(3):509–525, 2011.

[6] A. Hadim and L. C. Burmeister. Onset of convection in a porous medium with internal heat generationand downward flow. *Journal of Thermophysics and Heat Transfer*, 2(4):343–351, 1988.

[7] K. Heufer and H. Olivier. Experimental and numerical study of cooling gas injection in laminar supersonic flow. *AIAA journal*, 46(11):2741–2751, 2008.

[8] H. N. Kelly and M. L. Blosser. Active cooling from the sixties to nasp. In *NASA Conference Publication*, pages 189–189. NASA, 1992.

[9] M. Konopka, M. Meinke, and W. Schröder. Large-eddy simulation of supersonic film cooling at finite pressure gradients. In *High Performance Computing in Science and Engineering'11*, pages 353–369. Springer, 2012.

[10] M.-S. Liou and C. J. Steffen. A new flux splitting scheme. *Journal of Computational Physics*, 107(1):23 – 39, 1993.

[11] D. J. Lusher, S. P. Jammy, and N. D. Sandham. OpenSBLI: Automated code-generation for heterogeneous computing architectures applied to compressible fluid dynamics on structured grids. 173(2018):17–21, 2020.

[12] R. Mangalagiri, S. B. Thapa, and S. P. Jammy. Opensblifvm: An automatic code generation framework for finite volume methods on cpu's and gpu's. In *8th International and 47th National Conference on Fluid Mechanics and Fluid Power (FMFP)*, number FMFP2020-198, IIT Guwahati, Assam, India, December 2020.

[13] A. Meurer, C. P. Smith, M. Paprocki, O. Čertík, S. B. Kirpichev, M. Rocklin, A. Kumar, S. Ivanov, J. K. Moore, S. Singh, T. Rathnayake, S. Vig, B. E. Granger, R. P. Muller, F. Bonazzi, H. Gupta, S. Vats, F. Johansson, F. Pedregosa, M. J. Curry, A. R. Terrel, v. Roučka, A. Saboo, I. Fernando, S. Kulal, R. Cimrman, and A. Scopatz. Sympy: symbolic computing in python. *PeerJ Computer Science*, 3:e103, Jan. 2017.

[14] P. Raghuraman, B. Anderson, S. Sieger, D. Rousar, J. Hidhal, and K. Baxter. The hedi platelet transpiration cooling concept-lightweight, durable, flight proven. In *6th Joint Thermophysics and Heat Transfer Conference*, page 2005, 1994.

[15] I. Z. Reguly, G. R. Mudalige, M. B. Giles, D. Curran, and S. McIntosh-Smith. The OPS domain specific abstraction for multi-block structured grid computations. *Proceedings of WOLFHPC 2014: 4th International Workshop on Domain-Specific Languages and High-Level Frameworks for High Performance Computing - Held in Conjunction with SC 2014: The International Conference for High Performance Computing, Networking, Storage and Analysis*, pages 58–67, 2014.

[16] K. D. Song, S. H. Choi, and S. J. Scotti. Transpiration cooling experiment for scramjet engine combustion chamber by high heat fluxes. *Journal of propulsion and Power*, 22(1):96–102, 2006.

[17] T. Tanahashi, H. Kawai, J. Masuzawa, T. Sawada, and T. Ando. Flow of the entrance region in a porous pipe: 3rd report: Transient flow. *Bulletin of JSME*, 25(205):1070–1078, 1982.

AUTHORS PROFILE

Sujan Bahadur Thapa is a final year Mechanical Engineering student at SRM University- AP, Andhra Pradesh. He graduated high school with a perfect score from Xavier Academy and now is a co-author of a recently published paper at the 8th International and 47th National conferences on Fluid Mechanics and Fluid Power (FMFP-2020) held on 10th December regarding shock-capturing schemes. He also has reasonable experience in the field of numerical computations and fluid dynamics via workshops and hands-on experience with various projects under his B.Tech tenure.

Gogula Sri Kalyan Mani Deepak is a final year B.tech Student at SRM University- AP, Andhra Pradesh in the mechanical wing. With his excellent performance in his high school Sri Chaitanya, he is now a lead author and holder of a conference paper at the 8th International and 47th National conferences on Fluid Mechanics and Fluid Power (FMFP-2020) held at IIT Guwahati 2020. Currently, he is also employed at Cognizant Technology Solutions (CTS) and deals with digital backgrounds particularly at digital engineering and IoT.

Suchet Bahadur Thapa is a final year Mechanical Engineering student at SRM University- AP, Andhra Pradesh. He graduated high school with distinction marks from Xavier Academy and now is a co-author of a recently published paper at the 8th International and 47th National conferences on Fluid Mechanics and Fluid Power (FMFP-2020) regarding automatic code generation framework for Finite volume methods on CPU's and GPU's. He also has reasonable experience in the field of numerical computations and fluid dynamics via workshops and hands-on experience with various projects under his B.Tech tenure.