Defence Science Journal, Vol. 67, No. 5, September 2017, pp. 497-503, DOI : 10.14429/dsj.67.10926

Simplified CFD based Approach for Estimation of Heat Flux over Wedge Models in High Speed Plasma Flows

L. Aravindakshan Pillai* and Praveen Nair

Vikram Sarabhai Space Centre, Indian Space Research Organisation, Trivandrum - 695 022, India
E-mail: aravind_vssc@yahoo.co.in

ABSTRACT

Analysis of plasma flows at hypersonic velocity over blunt bodies is quite complex and challenging as it involves complex flow physics and carries several uncertainties. Simultaneous simulation of all the parameters as existing in re-entry flight puts constraints on most of the ground based experiments. Numerical simulations, on the other hand, require modelling of ionisation and real gas effects and prove to be computationally costly. This paper highlights the development of unstructured, cell centred second order accurate parallel version of in-house computational fluid dynamics (CFD) solver where high temperature equivalent properties used from Hansen's 7 species model and establishment of a simplified procedure for estimation of heat flux over wedge models tested in Plasma Wind Tunnel facility, Vikram Sarabhai Space Centre. Numerical simulations were carried out for Plasma tunnel initially to get the flow properties inside the tunnel when operated without any model. A simplified CFD based approach is established for computing the heat flux over the bodies tested inside the tunnel and compared with the measured data. The comparison of numerical and measured values shows that the proposed methodology captures the flow physics and various parameters with acceptable levels of accuracy.

Keywords: Re-entry module; High enthalpy flows; Computational fluid dynamics; Heat flux; Plasma wind tunnel

NOMENCLATURE

CFL	Courant, Friedrichs and Lewy number
dt	Time step
ds	Area of elemental sides
F	Fluxes in x direction
G	Fluxes in y direction
H	Flux vector
M	Mach number
P	Static pressure (Pa)
R	Universal gas constant
r	Radial distance (m)
S	Source term
T	Static temperature (K)
t	Time (s)
U	Vector of conservative variables
V_i	Computational cell volume
Ω	Control volume
u	Velocity in x direction (m/s)
v	Velocity in y direction (m/s)
q_i	Velocity of fluid inside the cell (m/s)
ρ	Density (kg/m^3)
Δl_i	Characteristic dimension of quadrilateral element (m)
μ_l	Laminar viscosity (kg/ms)
Γ	Surface area (m^2)
γ	Ratio of specific heats
σ	Shear stress (N/m^2)

Received : 28 November 2016, Revised : 21 June 2017
Accepted : 21 June 2017, Online published : 19 September 2017

1. INTRODUCTION

During re-entry of a module into earth's atmosphere at very high velocities, it is subjected to severe aero thermal heating. The temperature of the air can be around 7000 K to 9000 K depending on the entry velocity[1]. At these elevated temperatures, air gets dissociated and ionised (partially or fully) and becomes plasma and flows over the module. Since the module is engulfed in plasma, the heat transfer and surface chemical reactions are very unique and will be different from the conventional heating. During reentry of a module into planetary atmosphere, the flight velocity and the air density varies throughout and hence the heating. The flow physics and the heating mechanism should be understood for making a foolproof thermal design of the module to protect it from heating. Depending on the altitude and the velocities at which body moves during flight, the flow regimes are shifting from rarified flow, non-equilibrium flow, frozen flow or continuum equilibrium flows. In addition to the flow transition, there are several known and unknown phenomena like laminar-turbulent transition, turbulent-air chemistry interactions, ablation, etc.

Such flows can be addressed using three tools: analytical, numerical and experimental. Analytical tools are best suited for simple geometries and are based on simplifying assumptions. Hence analytical tools are not fully dependable for practical design purpose. Experimental tools prove to be reliable tools for studying hypersonic flows since ground tests can duplicate actual flight conditions with minimum assumptions. However experiments are expensive and complete simulation of the flow parameters during re-entry cannot be simulated simultaneously as existing in flight in these facilities. These limitations on analytical and experimental techniques call for development of a robust, efficient and accurate computational fluid dynamics (CFD) solver to simulate the complex flows associated with re-entry modules.

The rapid and significant advances in computational facilities, both in terms of hardware and software, have identified CFD as an important and indispensable numerical tool for research and development in this field. Existing schemes and models used in CFD solvers perform reasonably well in the subsonic and supersonic flow regimes. When applied to hypersonic reacting flows, these encounter several limitations due to the characteristics of the high enthalpy high speed flows. A requirement of extremely dense grid near the wall limits the solver time-step and demands more computational time for convergence[2]. Moreover the source term from the chemical reactions poses an additional barrier to efficient computation. The reaction and the flow time scales are generally different and hence the solver must consider both to get a reasonable convergence rate. As far as flow turbulence is concerned, various models are currently in use. Each of them has various limitations in terms of implementation issues, computational efficiency and accuracy in different flow regimes. Such complexities involved in the simulation of the high enthalpy hypersonic flows constantly call for use of a robust, accurate and computationally efficient flow solver.

The 6 MW plasma wind tunnel facility available in Vikram Sarabhai Space Centre, capable of simulating all re-entry conditions, is made use of for validating the CFD code by measuring the heat flux in the test model.

This paper presents the details of a parallel version of in-house unstructured finite volume based CFD solver used for carrying out various numerical experiments and arriving at a simplified approach for estimating heat flux over bodies tested in 6 MW plasma tunnel facility and finally comparing the numerical results with the measured values.

2. GOVERNING EQUATIONS

The generic form of conservation equations describing a 2D/axisymmetric laminar flow can be expressed as[3]

$$\frac{\partial U}{\partial t}+\frac{\partial F}{\partial x}+\frac{1}{r^m}\frac{\partial (r^m G)}{\partial y}=S \tag{1}$$

For axisymmetric problems m = 1 and for 2-D problems m=0.

$$U=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho E\end{bmatrix} S=\begin{bmatrix}0\\ 0\\ m(p+\sigma_{\theta\theta})\\ 0\end{bmatrix} F=\begin{bmatrix}\rho u\\ \rho u^2+P-\sigma_{xx}\\ \rho_{uv}-\tau_{xr}\\ (\rho E+P-\sigma_{xx})u-v\sigma_{xr}-K\dfrac{\partial T}{\partial x}\end{bmatrix}$$

$$G=\begin{bmatrix}\rho v\\ \rho uv-\sigma_{rx}\\ \rho v^2+P-\sigma_{rr}\\ (\rho E+P-\sigma_{rr})v-u\sigma_{xr}-K\dfrac{\partial T}{\partial y}\end{bmatrix}$$

Turbulence is modelled using Goldberg's one equation turbulence model[4] in this paper.

The laminar viscosity is modelled using Sutherland's law given by

$$\mu_l=1.716\times10^{-5}.\left(\frac{T}{273}\right)^{3/2}\frac{384}{T+111}kg/(ms) \tag{2}$$

Turbulence viscosity is computed from turbulence model

3. NUMERICAL METHODOLOGY

A finite-volume based cell centred schemes is used for discretisation. In this method, the domain is divided into quadrilateral control volumes. The governing equations can be written as

$$\frac{\partial U}{\partial t}+\nabla.H-S=0 \tag{3}$$

where H includes fluxes F and G.

Integrating this equation over a control volume (Ω), we obtain

$$\int_\Omega\left(\frac{\partial U}{\partial t}+\nabla.H-S\right)d_\Omega=0 \tag{4}$$

Or

$$\int_\Omega\frac{\partial U}{\partial t}d\Omega+\int_\Omega\nabla.Hd\Omega-\int_\Omega Sd\Omega=0 \tag{5}$$

Applying Greens theorem, we get

$$\int_\Omega\frac{\partial U}{\partial t}d\Omega+\int_\Gamma H.nd\Gamma-\int_\Omega Sd\Omega=0 \tag{6}$$

where Γ is the surface area enclosing the respective control volume Ω. The above equation can be rewritten in an appropriate fashion for a typical computational cell as follows

$$\frac{dU_i}{dt}V_i+\sum_{faces}Hds-S_iV_i=0 \tag{7}$$

where V_i is the computational cell volume and ds is the area of elemental sides. Time integration is performed using Runge-Kutta method. This method is only conditionally stable as it is an explicit scheme. The convergence is accelerated by employing local time stepping where the maximum allowable time step is specified by stability criterion specified as:

$$\Delta t_i\le\left(\frac{\Delta l_i}{q_i+c_i}\right) \tag{8}$$

where Δl_i is the characteristic dimension of quadrilateral element and q_i and c are magnitude of fluid velocity cell and sound velocity in the cell and are given by

$$q_i=\sqrt{u_i^2++v_i^2}$$
$$c_i=\sqrt{\gamma RT_i} \tag{9}$$

hence the time step of explicit solution can now be expressed as:

$$\Delta t_i=CFL\times\left(\frac{\Delta l_i}{q_i+c_i}\right) \tag{10}$$

where CFL is the Courant, Friedrichs and Lewy number.

In present simulations AUSM+-UP scheme[5] has been chosen for the evaluation of convective fluxes wherein the in viscid flux is explicitly split into two parts: convective and pressure terms by considering convective and acoustic waves as two physically distinct processes.

Viscous fluxes are evaluated using conventional second order accurate least square based scheme[6]. For computing viscous flux components the derivatives of *u, v* and *T* are to be evaluated. This is done using least square based central difference scheme. Consider a pseudo cell ABCD (Fig. 1) with side n_3-n_4. Derivative is evaluated using the expression:

$$\left(\frac{\partial T}{\partial x}\right)_{n3-n4} = \frac{(T_A - T_C)(y_B - y_D) - (T_B - T_D)(y_A - y_C)}{(x_A - x_C)(y_B - y_D) - (x_B - x_D)(y_A - y_C)} \quad (11)$$

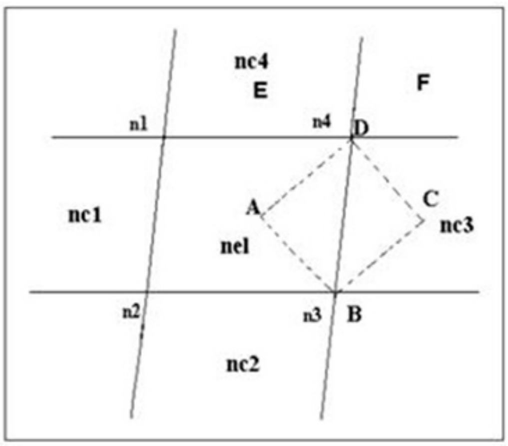

Figure 1. Finite volume connectivity for viscous flux evaluation.

4. VALIDATION OF IN-HOUSE CFD SOLVER

The solver has been validated for a series of test cases involving varying levels of geometric and flow complexities. This paper presents two test cases: one internal flow and another external flow.

4.1 NASA B2 Nozzle

A two-dimensional flow field simulation is carried for the NASA B2 nozzle[7] shown in Fig. 2. This nozzle is a standard facility nozzle used to supply supersonic flow at Mach 2 to various test facilities. The simulations are carried out using subsonic boundary conditions at inlet where stagnation pressure of 10 bar and stagnation temperature of 500 K are specified. The top wall is a no-slip adiabatic wall and a supersonic condition is maintained at the exit.

Figure 3 shows the Mach number distribution within the computational domain with upper half showing contours and the lower half representing fields. The maximum Mach number of 2.14 occurs at the nozzle exit away from the axis. Comparison of numerical wall pressure distribution with the measured values was carried out and it is observed that the

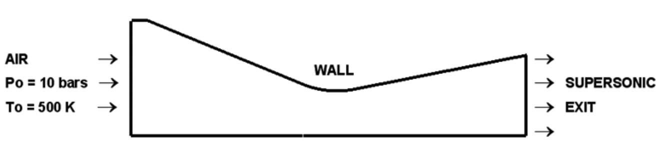

Figure 2. Geometry and boundary conditions for NASA B2 nozzle simulation.

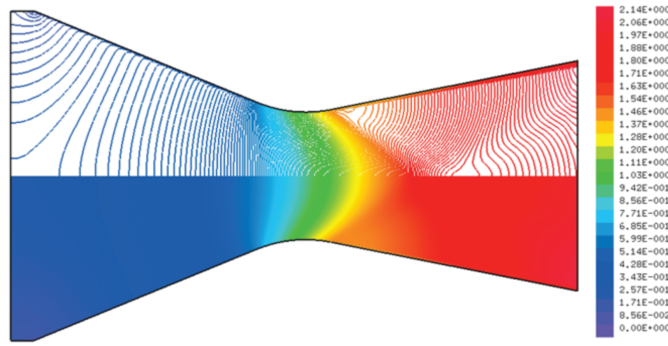

Figure 3. Mach number distribution for NASA B2 nozzle.

solver has captured the flow physics with reasonable and acceptable levels of accuracy.

4.2 Blunted Cone-Flare

Flow field around a blunted cone-flare[8] is of particular interest since it features most of the aspects of the hypersonic flow around re-entry vehicles. The region between the cone and the flare is critical with respect to the evaluation of the surface heat flux. Flow separation is induced by the shock wave-boundary layer-interactions, with subsequent flow reattachment that can dramatically enhance the surface heat transfer. The experiment was carried out in H3 hypersonic wind tunnel and the chamber conditions ensure that the flow is purely laminar. Hence this is one of the most suited test cases for validating in viscid flux evaluation schemes used in the solver, since complexities due to turbulence and reaction models are eliminated.

The geometry and boundary conditions for the blunted cone-flare are as shown in Fig. 4. Air at 580 K and 10 bar is passed through an axisymmetric nozzle where it expands to a nominal Mach number of 6 and enters the test section. This results in free stream values of P_{inf} = 673.67 Pa, T_{inf} = 67 K and M_{inf} = 6. The experiment is conducted in such a manner that the flow is laminar over the entire cone-flare.

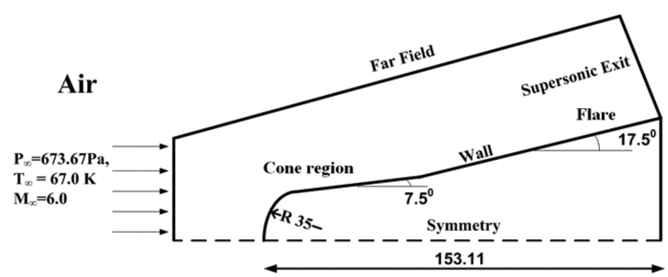

Figure 4. Geometry and boundary conditions for flow simulation over blunted cone-flare.

The simulations are carried out using constant wall temperature of 300 K to reproduce the experimental conditions. Grid independent studies are carried out using second order accurate solution mapped reconstruction. Accurate solution for this problem demand extremely fine mesh (micron level) near the cone flare junction.

Figure 5 shows the Mach number distribution for blunted cone flare. All features such as the detached bow shock in the stagnation region and shock-boundary layer interaction

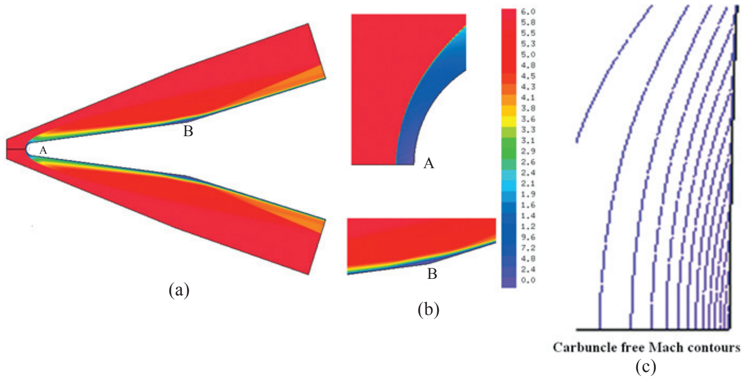

Figure 5. Mach number distribution for blunted cone-flare (a) entire computational domain (b) Enlarged view near region A and B (c) Carbuncle free zone near nose tip A.

in the cone-flare junction are captured. Mach contours near the stagnation zone are smooth proving that the inviscid flux evaluation scheme used in the solver makes it free from the carbuncle phenomena. From these results it can be concluded that the solver could capture the flow physics and predict heat flux with acceptable levels of accuracy.

5. NUMERICAL SIMULATION OF HIGH ENTHALPY FLOWS IN PLASMA WIND TUNNEL FACILITY

The 6 MW Plasma Wind Tunnel Facility available in Vikram Sarabhai Space Centre, ISRO is made use of for the model testing of re-entry thermal protection systems. The photograph of the facility is as shown in Fig. 6. In this facility, high temperature plasma is produced by heating air/ working medium to very high temperature using an electric arc maintained between the electrodes of the constrictor plasma generator. The facility essentially consists of a high power plasma generator, hypersonic nozzle, power supply system, coolant supply system, gas feed system, test chamber, and water cooled supersonic diffuser duct, heat exchangers, model injection system and vacuum pumping system. Blunt body tests simulate stagnation point heat transfer and blunted wedge tests simulate surface heat transfer and shear flows. The plasma produced in the generator is allowed to expand in a hypersonic nozzle that is exiting to a test chamber kept at high vacuum condition. The high velocity high temperature jet coming out from the nozzle when passes over the model simulates the re-entry conditions on the model.

Figure 6. Plasma wind tunnel facility.

The parameters that can be simulated depend on the facility operating conditions, nozzle area ratio, model configuration and finally the location of the model. The parameters like surface pressure, heat flux and in-depth temperatures of the model are measured and are used for validating the CFD code. In addition, all flow parameters that are existing on the model such as velocity, shear stress etc can be derived from the CFD analysis.

The numerical simulation of the flow field around the model in the plasma wind tunnel, involves the simulation of entire plasma generator, test chamber, test model, diffuser ejector system etc. The number of grids essential to capture all flow physics is very high, of the order of millions, which requires large computation time. Further for getting a grid independent solution is still time consuming. It is observed that usually plasma tunnel is operated for a standard test condition and calibration is performed to confirm the required simulation parameter. If the parameters are not meeting the requirement, facility will be operated for other conditions. Most of the time only model configuration and the location of the model will be changed. Hence a simple methodology is being evolved, wherein the analysis will be carried out in two steps. Initially only plasma facility without model is numerically simulated and the flow field in the test chamber is obtained. In the next phase, the numerical simulation of the model will be carried out wherein the flow field upstream of the model obtained from the first step and the test chamber pressure are used as the input. This helps to considerably reduce the computational time for specific cases.

To validate the methodology, two specific test cases, RLV (Reusable Launch Vehicle) nose cap joint simulation and MDA tile evaluation for HSP (Human Spaceflight Program) crew module, were numerically simulated and results were compared. Here, to obtain heat flux accurately, near wall grid resolution of one micron is considered and Y+ works out to be 4. The details are given as follows.

5.1 Numerical Simulation of Flow Field Inside 6 MW Plasma Facility without Model

Simulations are carried out using second order accurate in-house flow solver with Goldberg's one equation turbulence model. The computational domain consisting of the plasma generator, test chamber and the diffuser duct are considered for simulating the flow field in 6 MW facility with a model placed in the test section. Such a simulation requires extremely high computational time as we need to refine the grid near body to micron level. Hence simulation is initially carried out without model in the facility. Flow enters the facility nozzle at 3.22 bar and 7825 K. Here γ is taken as 1.23 and equivalent molecular weight as 21.0 based on past experience (based on Hanson's table values and earlier simulations). Entire computational domain is initialised with test chamber pressure of 100 Pa and slowly the pressure and temperature at inlet of the facility nozzle are increased to 3.22 bar and 7825 K. Simulation was continued till errors in conserved variables reduced by 4 orders. Figure 7 shows the Mach number distribution inside the computational domain. Flow expands to a Mach number of around 5 at the nozzle exit. Since the nozzle exit pressure is

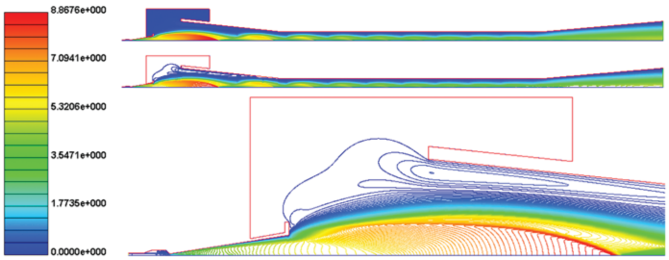

Figure 7. Mach contours inside plasma facility without model.

more than the test chamber pressure, the flow expands further in the test chamber to a Mach number of around 8.9. Expanded flow is confined to the diffuser with the help of catch cone where it undergoes a series of shocks. Mach number, static pressure and static temperature along the nozzle axis is obtained from CFD simulation so that one can directly obtain the flow conditions faced by model placed at different locations within the facility.

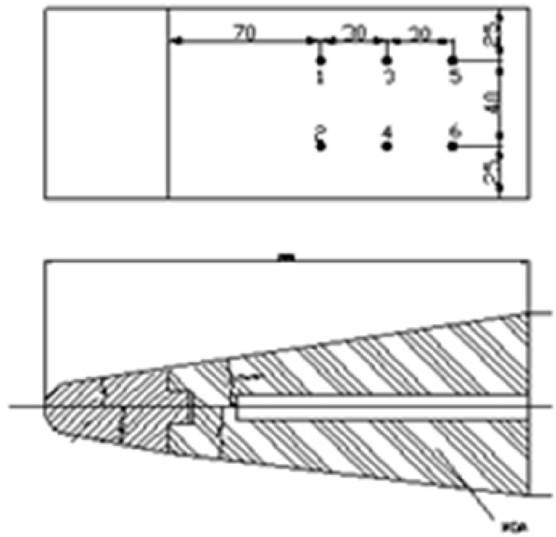

Figure 8. Calibration model.

5.2 Numerical Simulation of Flow Field Past Blunt Bodies Tested in 6 MW Facility

5.2.1 Testing of Double Wedge 2D Model in PWT Facility

As part of the evaluation of thermal performance of the RLV-TD nose cap to silica tile interface, a calibration test was conducted in PWT facility[9] wherein a double wedge model with 9° and 7° half angles is made use of for simulating the reentry heating simultaneously with shear flow condition. The detail of the calibration model used for determining the heat flux and to fix the facility operating condition is as shown in Fig. 8. Carbon Phenolic insert is used as the leading edge and rest of the body is made out of medium density ablative. 6 numbers of slug gauges are provided in the 9° surface and 4 numbers of slugs are provided on the 7° surface.

The calibration was carried out by injecting model into plasma jet when the facility was operating at 1150 Amp level. At this operating condition, air is heated to a total temperature of 7825 K. The model is kept in the plasma jet for 5 s - 6 s. The temperature rise of the copper slug is measured during its exposure to plasma. The heat flux at the top surface is computed from the temperature gradient and varies from 10 W/cm^2 to 9 W/cm^2 and at the bottom surface it varies from 8 W/cm^2 to 7 W/cm^2.

Numerical Modelling: Flow and fluid properties ahead of the calibration model is obtained from the numerical modeling of the plasma tunnel without model. Flow approach the body with static pressure of 85 Pa, static temperature of 1690 K and Mach number 5.658

Entire computational domain is initialised with Mach number of 5.658, static temperature of 1690 K and static pressure of 85 Pa. These values are also given as supersonic inflow boundary conditions. Wall of the model is given isothermal (300 K) no-slip boundary condition to get cold wall heat flux which will be the maximum heat flux experienced by the model during the test. A bow shock formed ahead of the model is well captured without any carbuncles. Figure 9 gives the cold wall heat flux over the model surface. Stagnation point heat flux is around 260 W/cm^2 and it decreases to as low as around 10W/cm^2 near the nose joint.The measured heat flux in the top surface varies from 11 W/cm^2 - 10 W/cm^2 whereas in the bottom surface it varies from 9 W/cm^2 - 8 W/cm^2. Measured heat flux are in close agreement with the numerical value proving that the simplified methodology is capable of capturing the flow physics and flow parameters with acceptable levels of accuracy.

5.2.2 Testing of 2D Wedge Model with Top Surface at 14° to Nozzle Axis.

As part of the qualification testing of MDA tiles in the boat tail region of CARE re-entry module of Human Space Program, a calibration test was carried out to determine the heat flux & shear stress on the wedge surface and fix the facility operating conditions[10]. The details of the calibration model are as shown in Fig. 10. A water cooled leading edge is used and 5 numbers of slug gauges are provided on the top surface of the wedge model to map the heat flux. The model is fixed to the model injection system in such a way that the top surface

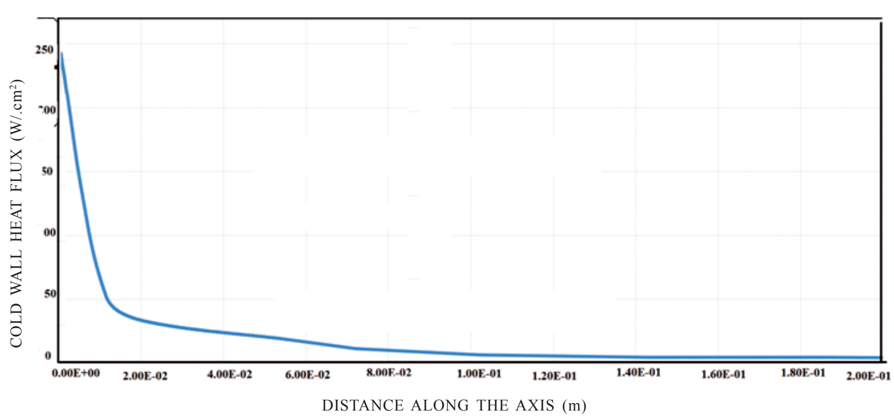

Figure 9. Variation of cold wall heat flux over the model.

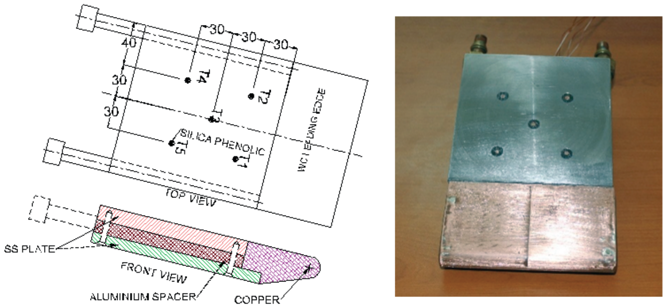

Figure 10. Details of the calibration model.

is inclined to 14 deg to the nozzle axis. The leading edge is kept at 300 mm from the nozzle exit. The facility is operated at a power level and air flow rate corresponding to a plasma temperature of 7600 K and arc column pressure of 3 bar. The stream calibration was carried out by injecting the model to the plasma jet and kept there for ~5 s. The heat fluxes were computed from the temperature rise rate of the slug gauges and are in the range of 22 W/cm^2 - 20 W/cm^2.

Numerical Modelling: The entire plasma generator test chamber and diffuser duct are included as the computational domain as in the previous case. As in case in section 5.2.1, numerical analysis of the plasma tunnel without model with nozzle inlet pressure and temperature of 3 bar and 7600 K was carried out. Since the nozzle exit pressure is more than the test chamber pressure, the flow expands further in the test chamber to a Mach number of 8.4.The Mach number at the point (300 mm from nozzle exit) where the model is placed is around 5.5.

Simulation of Flow Field over Wedge Shaped Model: Flow and fluid properties ahead of the model are calculated from tunnel flow simulation and the corresponding static pressure, temperature and Mach number are 99 Pa, 1700 K and 5.5 respectively. The entire computation domain is initialised with Mach number of 5.5, static temperature of 1700 K and static pressure of 99 Pa. These values are given as supersonic inflow boundary conditions. Wall of the model is given isothermal no-slip boundary conditions to get cold wall heat flux.

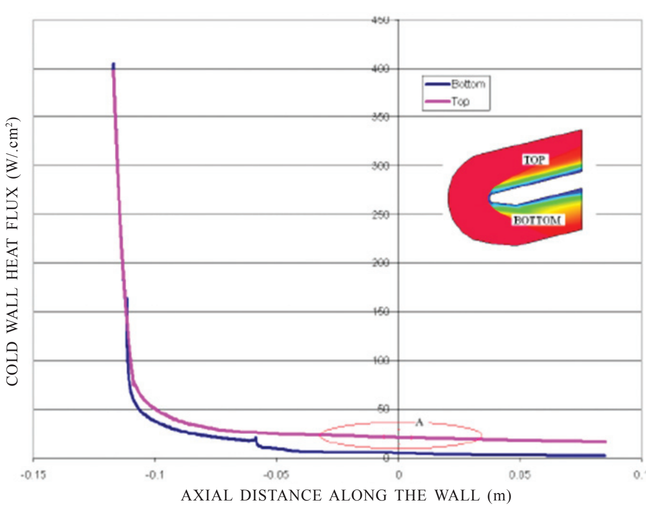

Figure 11. Variation of cold wall heat flux over the top and bottom model surface.

From the analytical results, it was observed that, bow shock formed ahead of the model is well captured. Figure 11 gives the cold wall heat flux over the model surface. As can be seen, the stagnation point heat flux is around 400 W/cm^2 and it decreases to as low as 20 W/cm^2 at the farthest point on the top surface of the model. Heat flux measured near point A is around 22 W/cm^2 – 20 W/cm^2 and the same predicted by the numerical method is 22 W/cm^2.

The comparison of the heat flux (W/cm^2) measured with CFD prediction is given as follows:

	Double wedge 2d model		2D wedge model	
	Top surface	**Bottom surface**	**Fore end**	**Aft end**
CFD	11.82 - 10.71	9.55- 8.62	21.81	20.3
Measured	11-10	9-8	22.3	19.9

6. CONCLUSIONS

In the present study, initially an unstructured cell centred based N-S solver developed in-house is numerically validated with experimental results for two standard test cases. Test cases addresses external flows as encountered in plasma wind tunnel facility. Simulations are carried out for the tunnel without any model to get the flow parameters at various locations in the test chamber where model is to be tested. A series of numerical simulations are carried out for various models tested in the facility to compare the numerical and measured model surface heat flux values. The results of these simulations show that the measured and numerical heat flux values are in close agreement establishing that the present simplified methodology captures the flow physics and flow parameters with acceptable levels of accuracy. The main advantage with this simplified methodology is that simulations need to be carried out only for body (neglecting the tunnel geometry) assuming single species, saving enormous computational time.

REFERENCES

1. Reynier, Ph; Beck, J.; Bouilly, J.-M.; Chikhaoui, A.; Kosarev, M.; Lino Da Silva, M.; Marraffa Surzhikov, L. & Vacher, D. Validation of aerothermal chemistry models for re-entry applications : Synthesis of entry experimental achievements, Pres-02-2015, pp.1-24
2. Wang, Z.J. High-order computational fluid dynamics tools for aircraft design. *Philosoph. Trans. Royal Society*, 2014. pp. 1-18.
3. Veersteeg, H.K. & Malalasekera, W. An introduction to computational fluid dynamics: The finite volume method, John Wiley and Sons, 1995, pp. 10-102.
4. Goldberg, U. Hypersonic flow heat transfer prediction using single equation turbulence models. *ASME J. Heat Transfer*, 2001, **123**(1), 65-69. doi: 10.1115/1.1337653
5. Liou, M.S. A sequel to AUSM, Part II: AUSM+-UP for all speeds. *J. Comput. Phy.*, 2006, **214**(1), 137-170. doi: 10.1016/j.jcp.2005.09.020
6. Munikrishna, N. & Balakrishnan, N. On viscous flux discretization procedure for finite volume and meshless solver. No. G21648, Indian Institute of Science, 2009, pp. 108. *PhD Thesis*.

7. Mary, L.M.; Lawrence, E.P. & Richard, J.R. The effect of throat contouring on two- dimensional converging-diverging nozzles at static conditions, *NASA-TP* No. 1704, 1993.
8. Savino, R. & Paterna, D., Blunted cone-flare in hypersonic flow. *Computers and Fluids*, 2005, **34**(7), 859-875. doi: 10.1016/j.compfluid.2004.05.012
9. Pillai, Aravindakshan Krishnaraj, K.; Mahendran, Praveen Kumar; Sreenivas, N.; Dileep, K.N, & Krishnakumar, G. Testing of C-C to silica tile joint of the RLV-TD in plasma wind tunnel facility. Quick look report on the test results. VSSC Internal document, Report No. APLD:TR: 35/15.
10. Pillai, Aravindakshan; Krishnaraj, K. & Sreenivas, N. Performance testing of thermal protection system of HSP crew module for CARE mission in plasma wind tunnel facility - test results. VSSC Internal document, APLD/TR/27/14.

ACKNOWLEDGMENTS

The authors acknowledge the valuable suggestions and technical support provided by Dr. T. Jayachandran, Deputy Director, PRSO, and other engineers of Propulsion Research Group, Vikram Sarabhai Space Centre, Indian Space Research Organisation, Thiruvananthapuram.

CONTRIBUTORS

Mr L. Aravindakshan Pillai is post graduate in Mechanical Engineering from IISc, Bengaluru and heading the Propulsion and Plasma Research Group in Vikram Sarabhai Space Centre, ISRO. He is working in the area of design and development of plasma wind tunnels for reentry simulation and numerical modeling of high temperature flows.
Contribution in the current study is the grid sensitivity study and generation of data for validation of the code through the conducting model tests in the Plasma Wind Tunnel facility.

Dr Praveen Nair took his PhD in Computational Fluid Dynamics from IIT, Bombay and now heading the CFD Section, FMT Division of Vikram Sarabhai Space Centre, ISRO. He is also working in the area of parallel computing.
In the present study he has developed the CFD code and validated the code using the wind tunnel tests.