Defence Science Journal, Vol. 66, No. 3, May 2016, pp. 203-209, DOI : 10.14429/dsj.66.8873

RESEARCH PAPER

A Novel CFD Method to Estimate Heat Transfer Coefficient for High Speed Flows

A. Bhandarkar, M. Dharavath, M.S.R. Chandra Murty, P. Manna, and D. Chakraborty*

Directorate of Computational Dynamics, Defence Research and Development Laboratory, Hyderabad – 500 058, India

**Correspondence e-mail: debasis_cfd@drdl.drdo.in*

ABSTRACT

Accurate prediction of surface temperature of high speed aerospace vehicle is very necessary for the selection of material and determination of wall thickness. For aerothermal characterisation of any high speed vehicle in its full trajectory, it requires number of detailed computational fluid dynamics (CFD) calculations with different isothermal calculations. From the detailed CFD calculations for different flow conditions and geometries, it is observed that heat transfer coefficients scale with the difference of adiabatic wall temperature and skin temperature. A simple 'isothermal method', is proposed to calculate heat flux data with only two CFD simulations one on adiabatic condition and other on isothermal condition. The proposed methodology is validated for number of high speed test cases involving external aerodynamic heating as well as high speed combusting flow. The computed heat fluxes and surface temperatures matches well with experimental and flight measured values.

Keywords: Aerodynamic heating, heat flux, thermal analysis, computational fluid dynamics, scramjet combustor

1. INTRODUCTION

Certain critical applications where working temperature is very high, it is necessary to calculate the temperature distribution in the solid to ensure it should not cross the metallurgical limit. The selection of material and wall thickness of high speed aerospace vehicle flow is largely dependent on accurate estimation of wall temperature distribution in the solid. Aerospace vehicles during ascent phase trajectory experience aerodynamic heating caused by viscous dissipation and the conversion of kinetic energy into thermal energy in the boundary layers surrounding the object. The combustion of fuel in high speed air stream (Ramjet/scramjet engines) also cause significant heat transfer to the combustor walls and fuel injecting devices.

In the past, the problem was simplified by calculating aerodynamic field and the temperature inside the solid body separately by imposing a prescribed wall heat flux or temperature at the interface. Thus, the complexity of heat transfer processes between the fluid and the vehicle structure is described by a pre-determined heat transfer coefficient which shows that the heat transfer process is independent of the solid properties. Different engineering methods[1-3] are proposed in the literature based on empirical formulae to estimate heat transfer coefficients for high speed flows. Eckert's method[1] uses reference enthalpy and boundary layer edge data for the estimation of heat transfer coefficient. Van Driest method[2] are based on flow past flat plates under zero pressure gradient and uses Reynold's analogy principle to calculate heat transfer coefficients. Fay and Riddle method[3] calculates the stagnation point heat flux from the numerical solution of chemically reacting boundary layer flows under equilibrium conditions. Although, these methods predicts heat transfer coefficients for engineering design for simple attached flows, but their applications to complex separated flows are not beyond doubt. To cater the uncertainty in the methodology, various design factors are introduced and vehicle structures become heavier. With the advent of powerful computer and robust numerical algorithms, conjugate heat transfer (CHT) methods[4-6] are introduced in high speed fluid flow by solving simultaneously Navier Stokes Equations in fluid region and heat conduction equation in solid region. Although these method are promising, but they are yet to fully mature for engineering design. Moreover, these methods require extensive computer speed and memory.

Due to inadequacy of empirical correlations and high turn about times of CHT methods, transient thermal analysis of high speed aerospace vehicle structures mostly depend on heat flux calculated from high fidelity CFD solution. CFD based heat transfer prediction requires number of isothermal calculations to determine the heat flux data base for different wall temperatures. From detailed flow calculations for different geometries, it is observed that heat transfers coefficients (h) scale with the difference of adiabatic wall temperature (T_{aw}) and skin temperature (T_w) as $h = \dot{q} / (T_{aw} - T_w)$. Based on this observation a simple method, referred as 'isothermal method', is proposed to calculate heat flux with only two CFD simulations one on adiabatic condition and other on isothermal condition. The proposed methodology is validated for number high speed test cases for both external and internal flows.

2. ANALYSIS

The heat transfer to a surface in high-speed flow[7] can be calculated as,

$$q = S_t * \rho_e * U_e * (E_{aw} - E_w) \quad (1)$$

where S_t is the stanton number and E_{aw} and E_w are the adiabatic

Received : 03 June 2015, Revised : 28 March 2016
Accepted : 01 April 2016, Online : 25 April 2016

wall enthalpy and the wall enthalpy respectively. To take into account irreversible processes in the boundary layer flow, the adiabatic wall enthalpy may be expressed in terms of a recovery factor r as

$$E_{aw} = E_e + r * (E_0 - E_e) \quad (2)$$

where E_e and E_0 are the free-stream static and total enthalpies, respectively. The recovery factor (r) is typically written in terms of the Prandtl number $\left(\Pr = \frac{\mu \times C_p}{K}\right)$ and is taken as $Pr^{1/2}$ for laminar flow and $Pr^{1/3}$ for turbulent flow[7]. Where C_p is the specific heat at constant pressure, μ is the free-stream dynamic viscosity and K is the thermal conductivity of the fluid.

High fidelity RANS simulations with fine resolution of boundary layer (Y^+~1), gives directly the wall heat flux for a fixed wall temperature. Heat flux obtained from one fixed wall temperature is not same for different wall temperatures. In the proposed 'isothermal method' methodology, cold wall heat flux ($\dot{q}_{cw}$) is calculated based on cold wall temperature (T_{cw}) and wall heat fluxes ($\dot{q}_w$) at other wall temperatures are scaled to the local wall temperature utilizing adiabatic wall temperature as given below,

$$\text{Local wall heat flux } \dot{q}_w = \dot{q}_{cw} \times \left[\frac{T_{aw} - T_w}{T_{aw} - T_{cw}}\right]$$

This 'isothermal method' requires one CFD simulation with isothermal wall and another CFD simulation with adiabatic wall at every trajectory point.

2.1 The CFD Methodology

Commercial CFD software Ansys Fluent[8] is used for simulating three validation cases namely, turbulent flow past circular cylinder at Mach 6.47, Kinetic heating of nose cone of an aerospace vehicle from subsonic to Mach 8 and Hydrogen fuelled scramjet combustor with wall injection; while, CFX-Tasc flow[9] is used for the simulation of hydrocarbon fueled scramjet combustor with strut injection. Ansys Fluent[8] solves three-dimensional Navier Stokes equations (conservation of mass, momentum, energy and species continuity) in a hybrid grid system using a collocated variable arrangement. Density based solver along with Roe flux difference splitting scheme[10] is used to simulate high Mach number compressible flow.

CFX is a fully implicit three dimensional Reynolds Averaged Navier-Stokes (RANS) code. The code is fully implicit, finite volume method with finite element based discretisation of geometry. The method retains much of the geometric flexibility of finite element methods as well as the important conservation properties of the finite volume method. It utilises numerical upwind schemes to ensure global convergence of mass, momentum, energy and species. The convective terms are discretised using a 2nd order scheme. Turbulence was modeled using k-ε model[11] and wall functions were used to model flow near the walls.

For combustion, the eddy dissipation model (EDM)[12] is used for its simplicity and robustness in predicting the performance of reactive flows in many engineering applications. The eddy dissipation model is based on the concept that chemical reaction is fast compared to the transport process in the flow. The model assumes that the reaction rate may be related directly to the time required to mix reactants at molecular level. In turbulent flows, the burning rate is assumed to be proportional to the rate at which turbulent kinetic energy is dissipated i.e., reaction rate is proportional to ε / k, where, k is the turbulent kinetic energy and ε is its rate of dissipation. Both hydrogen fuel and kerosene fuels are used for the reacting flows in the present simulation. Hydrogen oxidation is modelled with single step reaction $2H_2 + O_2 \rightarrow 2H_2O$, while, kerosene ($C_{12}H_{23}$) oxidation is represented on a molar basis by $C_{12}H_{23} + 17.75O_2 = 12CO_2 + 11.5H_2O$. The mixing rate determined from the EDM is given as.

$$R_{k,EDM} = -A_{ebu}\,\rho\,\frac{\varepsilon}{k}\,\min\left\{y_f, \frac{y_{ox}}{r_k}, B_{ebu}\frac{y_p}{1+r_k}\right\}$$

where ρ and y_f, y_{ox} and y_p are the density and mass fractions of fuel, oxidizer and products, respectively, A_{ebu} and B_{ebu} are the model constants and r_k is the stoichiometric ratio.

Lagrangian particle tracking method (LPTM) is used for discrete phase model to characterise the flow behavior of the dispersed phase fluid (kerosene liquid) alongwith the flow of the continuous phase predicted using a discretised form of the RANS equations. Log-normalised maximum residue of -04 is considered as the convergence criteria. The software is thoroughly validated for nonreating flows as well as reacting flow with hydrogen and hydrocarbon fuels and the simulation results were published[13-16].

3. VALIDATION TEST CASES

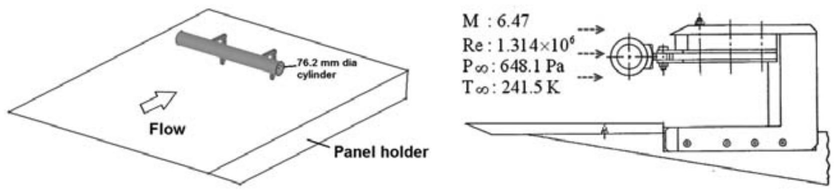
Following four test cases are selected for the validation of the proposed method described above.

(i) Turbulent flow past circular cylinder at Mach 6.47
(ii) Kinetic heating of nose cone of an aerospace vehicle from subsonic to Mach 8
(iii) Hydrogen fuelled scramjet combustor with wall injection.
(iv) Hydrocarbon fuelled scramjet combustor with strut injection.

First two test cases pertain to external aerodynamic heating where free stream Mach number ranges from low supersonic to hypersonic; while next two cases relate to internal heating in a scramjet combustor where flow velocities are of the order of 1000 m/s. Computed heat fluxes and surface temperatures are compared with the available literature and flight data.

3.1 Test case-1: Turbulent Flow Past Circular Cylinder at Mach 6.47

A 76.2 mm diameter, 12.7 mm thick 610 mm long cylinder made of AISI-321 is tested at Mach 6.47 in the 8 ft high temperature hypersonic blowdown tunnel[17] as shown in Fig. 1. The vitiated gas at test section is having Mach number, pressure, temperature, and Reynolds number of 6.47, 648 Pa, 241.5 K, and 1.312 X 10^6 per m, respectively. Pressure transducers and high response coaxial thermocouples were used to measure the flow field and heating rate distribution at various locations. Details of the experimental configurations, the tunnel flow conditions, and the experimental results are given[17]. As shown in Fig. 1.



(a) (b)

Figure 1. Schematic of (a) experimental setup[17] and (b) tunnel flow conditions.

Very fine structured grid involving 140 X 110 grid points with minimum grid spacing of 1 μm is employed. To predict the heating rate accurately, wall boundary layer is resolved with very fine mesh which corresponds to Y^+ of 0.2. A very fine mesh (140×20) is also taken in the solid region of the cylinder. Time step of 1 ms is employed in the solution of RANS equations using Ansys Fluent[8]. The grid independence of the simulation results are presented[18] and are not repeated here. As the Reynolds number in the test section is 1.312 X 10^6 per meter, transitional simulation is performed by employing Menter's 4 equation transition model[19]. The predicted pressure and temperature behind the shock wave are in good agreement with other calculation reported[20] indicating that overall flow features are well captured.

The constancy of heat transfer coefficient for different wall temperature is shown in Fig. 2. It is clearly observed that azimuthal distribution of heat tranfer coefficients collapsed into a single curve for different wall temperatures. The computed cold wall heat flux is compared with the experimental and other numerical results[20] in Fig. 3. There is close agreement for the resullts of isothermal method with experiment. As we proceed away from the stagnation point, the heat flux is reducing exponentially. In the region 10°< θ < 25°, the experimental heat flux rate is showing some oscillatory trend which is not observed in the computed values. The circumferential variation of predicted surface temperature is compared with experimental results and computed results of Zhao[20], *et al.* in Fig 3. The computed heat flux and surface temperature matches well with the experimental results; while the computation of Zhao[20], *et al.* showed higher values in the region 10°< θ < 25°.

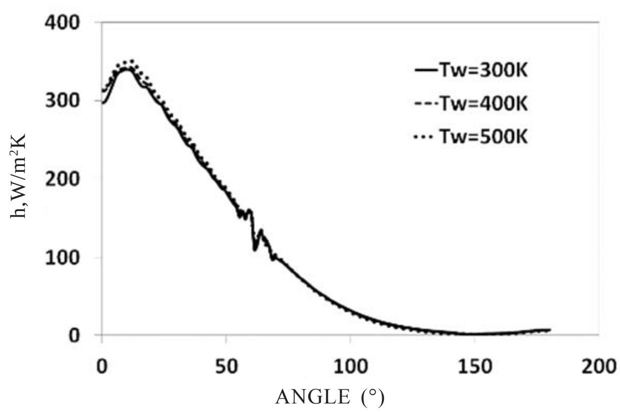


Figure 2. Azimuthal variation of heat transfer coefficients for different wall temperatures.

3.2 Kinetic Heating of Nose Cone of an Aerospace Vehicle from Subsonic to Mach 8

The schematic of the nose cone section of a high speed aerospace vehicle with temperature measurement locations are shown in Fig. 4. Eight Number of temperature sensors were located at X/d=1 and X/d=2 locations. The peak Mach number, altitude, angle of attack and side slip angle are 7 km, 45 km, 12° and 10°, respectively. About 70 points are selected along the trajectory to perform a detailed kinetic heating analysis along the trajectory with detailed 3D RANS calculation with k-ε turbulence model using Ansys Fluent[8]. Axial distribution of heat transfer coefficients with different wall temperature is shown in Fig. 5 to depict constancy of heat transfer coefficients with different wall temperatures.

Figures 6 and 7 show the comparison of predicted wall temperatures with flight measured values for locations T1 to T8. Isothermal method captured the trend of flight measurements for all locations and performed very well for the first axial location (X/d = 1). The method over predicts the surface temperature (about 15 % - 20 %) for second axial location (X/d = 2).

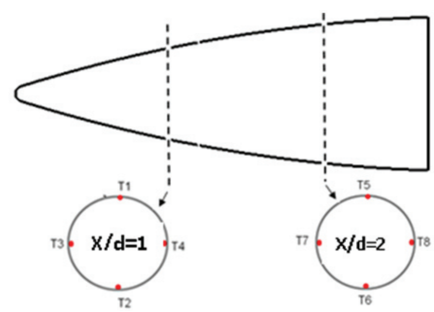


Figure 4. Schematic of nose cone with temperature measurement locations.

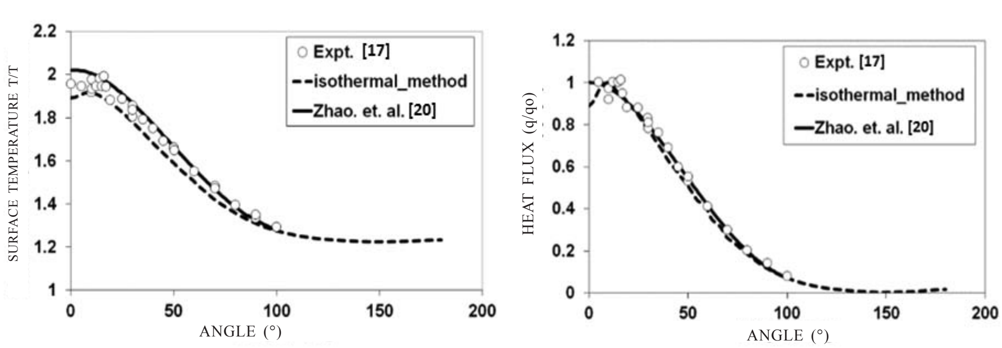


Figure 3. Heat flux and surface temperature variation along cylinder.

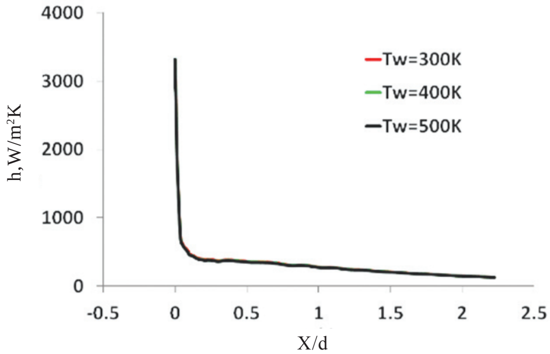


Figure 5. Heat transfer coefficient (h) distribution along axial direction for nose cone.

3.3 Hydrogen Fuelled Scramjet Combustor with Wall Injection

Development of scramjet engine is being pursued form early 1960s for hypersonic air breathing propulsion system for civil and military applications. Hydrogen fuelled scramjet combustor has been investigated experimentally and numerically for the last few decades. Efforts have continued to conduct well instrumented experimentation of scramjet combustor flow field to validate CFD data. The SCHOLAR experiment conducted at NASA Langley's Direct Connect Supersonic Test facilities[21-22] gives detailed measurements of temperatures and species mass fraction at various cross sections as well as wall pressures for hydrogen fuel is injected at 30° angle to Mach 2

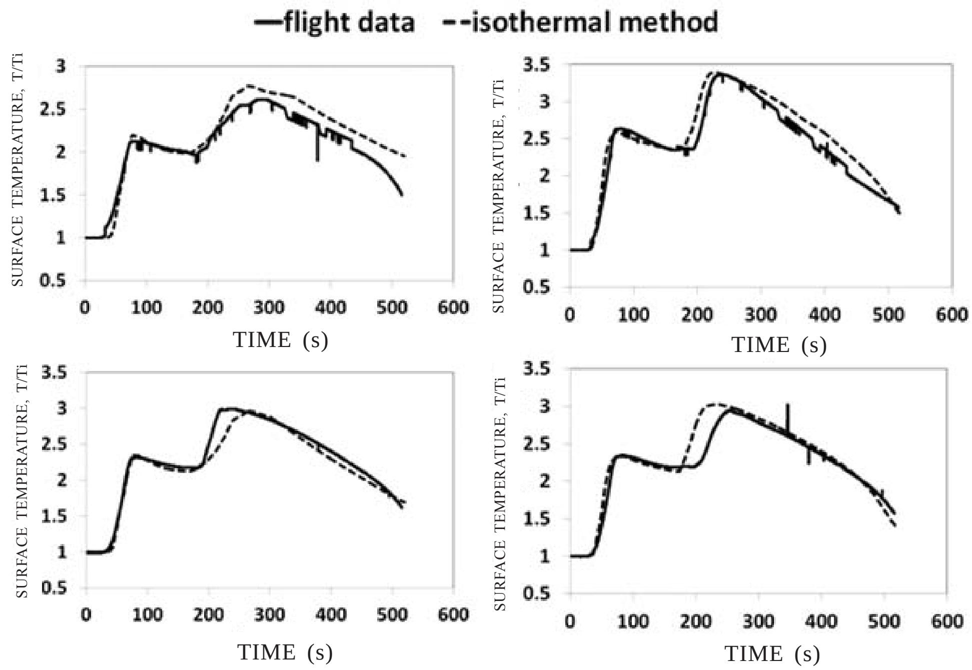


Figure 6. Comparison of wall temperature at x/d = 1 (a) T1, (b) T2 (c) T3 (d) T4.

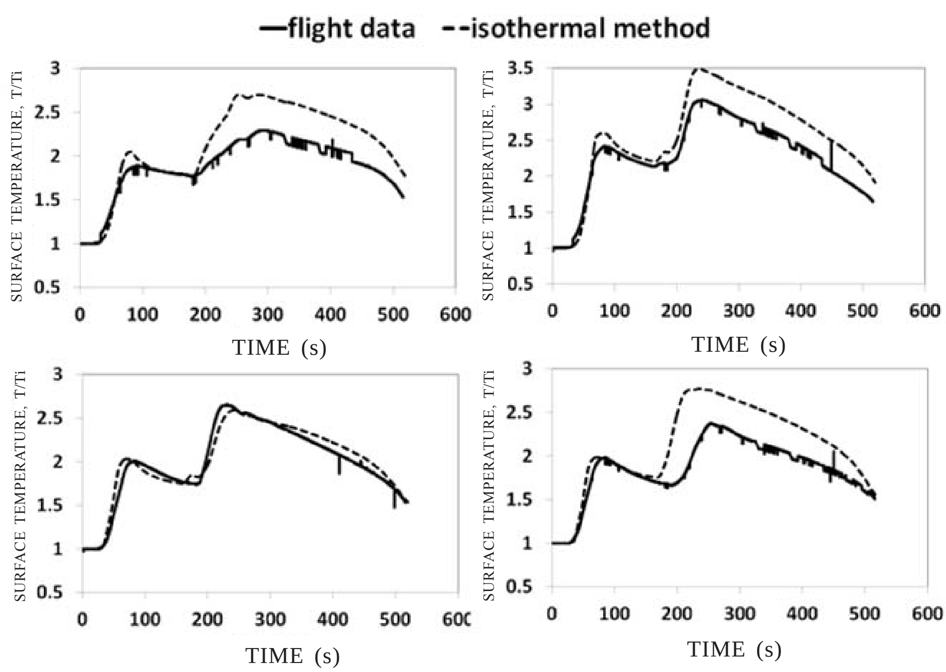


Figure 7. Comparison of wall temperature at x/d = 2 (a) T5, (b) T6 (c) T7 (d) T8.

airstreams with 1200 K temperature in a divergent duct. The schematic of the experimental condition is shown in Fig. 8. The reacting flow field of SCHOLAR experiment was simulated extensively numerically through RANS as well as LES models by many researchers with different chemical kinetics schemes, turbulence models and well reported in the literature[23-27].

Front section of the combustor (containing the isolator and injector) is made of copper while the rear section is made of carbon steel. The thickness of the copper section is 45 mm while the thickness of the carbon steel section is 18 mm. The wall thickness and the conductivity of the materials allow the combustor to run without cooling. Since, the duct is un-cooled, surface temperatures vary greatly during the test. Temperature history is measured in the combustor at two locations (X=197 mm and 426 mm) for the copper section and one location for the carbon steel block (X=978 mm). Three dimensional unsteady heat conduction equation is solved to determine the thermal response of the combustor wall to the hot combusting gas in the duct using CFD generated convective heat flux data with different isothermal conditions (300 K, 500 K, 750 K, 1000 K). The axial distribution of heat transfer coefficient $h = \dot{q} / (T_{aw} - T_w)$ is presented in Fig. 9.We can observe that for different isothermal calculations, heat flux distribution collapse into regular curve when scaled with $(T_{aw} - T_w)$ for the entire domain. Hence, for thermal response calculations, it is sufficient to carryout two different CFD simulations (one adiabatic and one isothermal). For any other isothermal condition heat flux can be estimated by employing $h = \dot{q} / (T_{aw} - T_w)$. The computed wall temperature history at two different locations X = 426 mm (near the end of copper section) and X = 978 mm (middle of carbon steel section) are compared with experimental data in Fig. 10. Computed temperature over predicts (~10%) the experimental value at X = 426 mm and under predicts (~10%) the temperature at X = 978 mm. Whether the temperature dependent material properties can narrow these differences further is not investigated.

3.4 Hydrocarbon Fuelled Scramjet Combustor with Strut Injection

Due to higher energy density and easier handling issue, liquid hydrocarbons fuels are considered for the scramjet engine for volume limited applications in the lower hypersonic (M<8) flight regimes. Atomisation, vaporisation, mixing and slow chemical reactions are some of the major technical and scientific problems in the realisation of liquid hydrocarbon based scramjet engine. Strut based injection systems are being considered to alleviate the problem of slow lateral fuel transport in the air stream. Kerosene fuelled scramjet combustor for an hypersonic airbreathing mission[28] are numerically explored for design optimisation and important reacting flow field results including grid independence studies are presented[29,30]. Typical half scale scramjet combustor is shown in Fig. 11 which has divergence in vehicle upper surface. Kerosene fuel is injected through 220 injection holes (with 0.5 mm diameter for each hole) provided through 10 struts. For doing detailed thermostructural analysis of the combustor wall, number of reacting simulations with equivalence ratio 1.0 is carried out for different wall temperatures. The distribution of average heat transfer coefficient for top wall for three isothermal wall conditions (600 K, 800 K, and 1000 K) are shown in Fig. 12. The constancy of heat transfer coefficients for different wall temperatures are also observed in the simulation. Similar observations are also found for the heat transfer coefficients of other walls. Typical heat flux distribution on all the walls of the combustor is shown is Fig. 13. The computed heat flux is used in transient thermal analysis to select the material and thickness of combustor wall. It was observed that for both high

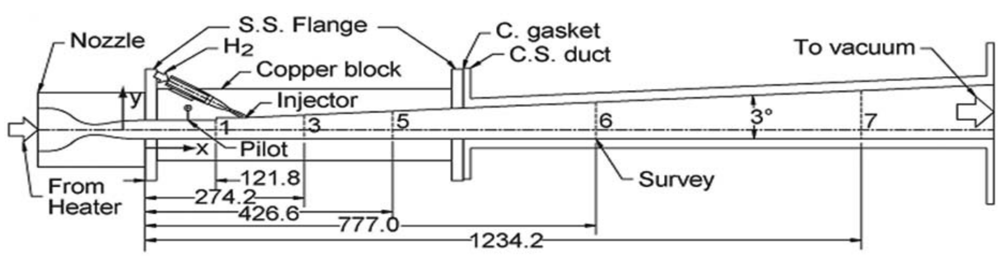


Figure 8. Schematic of Scholar experiment geometry[21].

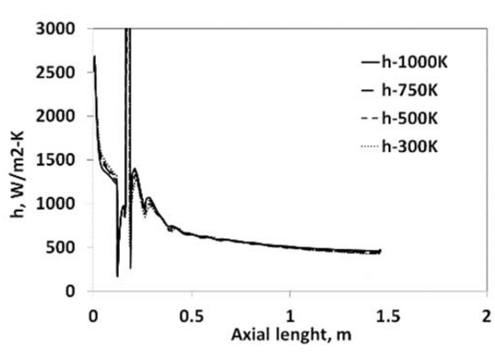


Figure 9. Axial distribution of convective heat transfer coefficient for different wall temperature

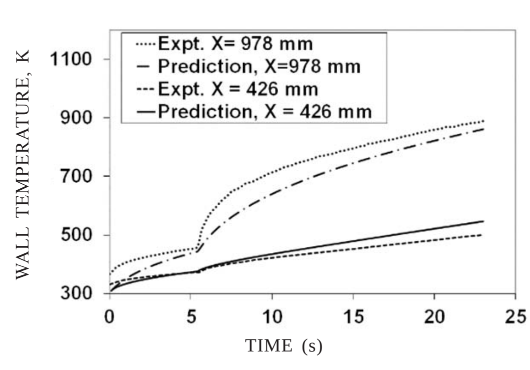


Figure 10. Comparison of wall temperature at locations X = 426 mm and 978 mm.

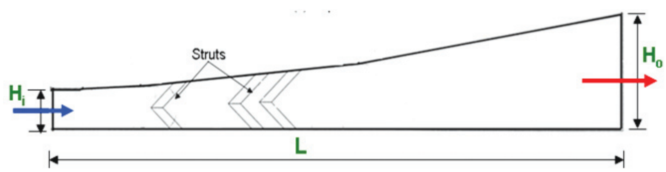


Figure 11. Schematic of the scramjet combustor.

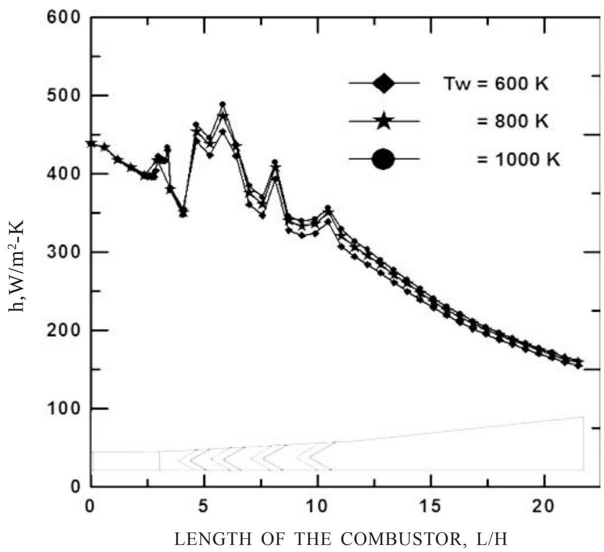


Figure 12. Axial distribution of heat transfer coefficients at upper wall.

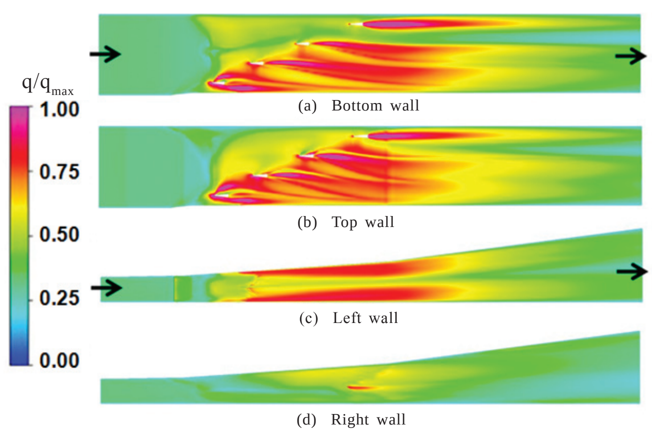


Figure 13. Heat flux distribution at scramjet combustor walls.

speed external and internal flows, the role of gas dynamics is very prominent and the flow is governed more by kinetic energy of the flow and less by sensible enthalpy.

4. CONCLUSION

Detailed aerothermal analysis is performed for high speed vehicle airframe using heat flux data from high fidelity CFD calculations. Reynolds Averaged Navier Stokes (RANS) equations are solved alongwith turbulence and combustion models using using commercial CFD software. It is observed that heat transfer coefficients scales with the difference of adiabatic wall temperature and skin temperature for different geometries and flow conditions. A new 'isothermal method', is proposed to calculate heat flux data for different wall temperature with only two CFD simulations one on adiabatic condition and other on isothermal condition. The proposed methodology is validated for different test cases including high Mach number turbulent flow past a cylinder, kinetic heating of nose-cone of high speed aerospace vehicle in its full trajectory, hydrogen and kerosene fuelled scramjet combustor airframe. The predicted surface temperatures matches well with experimental and flight measured values. The methodology reduces the need to perform number of high fidelity CFD calculation with different isothermal conditions and is being used to perform detailed aerothermal analysis for the airframe of different high speed vehicles for material selection and wall thickness determination. It was observed that for both high speeds external and internal flows, kinetic energy govern the flow dynamics more than the sensible enthalpy.

REFERENCES

1. Eckert, E.R.G. Survey of boundary layer heat transfer at high velocities and high temperatures. WADC Tech. Rep. 59-624, April 1960
2. Van, Driest E. R. Turbulent boundary layer in compressible fluids. *J. Space. Rockets,* 2003, **40**(6), 1012-1028. doi: 10.2514/2.7048
3. Fay, J. A. & Riddell, F.R. Theory of stagnation point heat transfer in dissociated air. *J. Aeronaut. Sci.*, 1958, **25**(2), 73-85. doi: 10.2514/8.7517
4. Manna, P. & Chakraborty, D. Numerical investigation of conjugate heat transfer problems. *J. Aerospace Sci. Technol.*, 2004, **56**(3), 166-175.
5. Zhao, X.; Sun, Z.; Tang, L.; & Zheng, G. Coupled flow-thermal-structural analysis of hypersonic aerodynamically heated cylindrical leading Edge. *Eng. Appl. Comput. Fluid Mech.*, 2011, 5(2), 170-179. doi: 10.1080/19942060.2011.11015361
6. Murty, M.S.R. Chandra; Manna, P. & Chakraborty, D. Conjugate heat transfer analysis in high speed flows. *Proceedings of the Institution of Mechanical Engineers, Pt G*: J. *Aerospace Eng.*, 2013, **227**(10), 1672-1681. doi: 10.1177/0954410012464920
7. Holman, J.P. Heat transfer. Ed. 10, McGraw-Hill Publishers, New York, 2010
8. Fluent 12.1; Theory Manual: 2009.
9. CFX-TASC flow Computation Fluid Dynamics Software. Version 2.11.1, AEA Technology Engineering Software Ltd. 2001.
10. Roe, P.L. Characteristic based schemes for the Euler equations. *Ann. Rev. Fluid Mech.*, 1986, 18, 337-365. doi: 10.1146/annurev.fl.18.010186.002005
11. Launder, B.E. & Spalding, D.B. The numerical computation of turbulent flows. *Comput. Meth. Appl. Mech. Eng.*, 1974, **3**, 269-289. doi: 10.1016/0045-7825(74)90029-2
12. Magnussen, B.F. & Hjertager, B.H. On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion. *In* 16th Symposium on Combustion, The Combustion Institute, 1976, pp. 719-725.
13. Saha, S. & Chakraborty, D. Numerical simulation hypersonic air intake. *Def. Sci. J.*, 2015, **65**(3), 189-195.

doi: 10.14429/dsj.65.8254
14. Dharavath, M. & Chakraborty, D. Numerical simulation of supersonic jet impingement on inclined plate. *Def. Sci. J.*, 2013, **63**(4), 355-362. doi: 10.14429/dsj.63.2545
15. Murty, MSR Chandra; Mishal, R.D. & Chakraborty, D. Numerical simulation of supersonic combustion with parallel injection of hydrogen fuel. *Def. Sci. J.*, 2010, **60**(5), 465-475. doi: 10.14429/dsj.60.57
16. Behera, R. & Chakraborty, D. Numerical simulation of kerosene fueled ramp cavity based scramjet combustor. *J. Aerospace Sci. Technol*., 2006, **58**(2), 104-112.
17. Wieting, AR. Experimental study of shock wave interference heating on a cylindrical leading edge. NASA TM-100484, 1987.
18. Murty, M.S.R. Chandra; Manna, P. & Chakraborty, Debasis. Conjugate heat transfer analysis in high speed flows. *J. Aerospace Eng.*, 2013, **227**(10), 1672-1681. doi: 10.1177/0954410012464920
19. Menter, F.R.; Langtry, R.B.; Likki, S.R.; Suzen, Y.B.; Huang, P.G., & Volker, S. A correlation based transition model using local variables Part-1 Model formulation. ASME-GT2004-53452), 2004.
20. Zhao, X.; Sun, Z.; Tang, L. & Zheng, G. Coupled flow-thermal-structural analysis of hypersonic aerodynamically heated cylindrical leading edge. *Eng. Appl. Comput. Fluid Mech*., 2011, **5**(2), 170-179. doi: 10.1080/19942060.2011.11015361
21. Cutler, A.D.; Diskin, G.S.; Danehy, P.M. & Drummond, J.P. Fundamental mixing and combustion experiments for propelled hypersonic flight. AIAA Paper No., 2002-3879, 2002. doi: 10.2514/6.2002-3879
22. Cutler, A.D.; Danehy, P.M.; O'Byrne, S.; Rodrigues, C.G. & Drummond, J.P. Supersonic combustion experiment for CFD model development and validation. AIAA Paper No. 2004-266, 2004. doi: 10.2514/6.2004-266
23. Rodrigues, C.G. & Cutler, A.D. Computational simulation of a supersonic-combustion benchmark experiment. AIAA Paper No. 2005-4424, 2005. doi: 10.2514/6.2005-4424
24. Keislter, P.G. A Variable turbulent Prandtl and Schmidt number model study for scramjet applications. Department of Mechanical and Aerospace Engineering, North Carolina State University, USA. PhD Thesis
25. Xiao, X.; Hassan, H.A. & Baurle, R.A. Modeling scramjet flows with variable turbulent Prandtl and Schmidt numbers. AIAA Paper No. 2006-0128, 2006. doi: 10.2514/1.26382
26. Chandra Murty, M.S.R. & Chakraborty, D. Numerical simulation of angular injection of Hydrogen fuel in scramjet combustor. *J. Aerospace Eng.*, 2012, **226**(7), 861-872. doi : 10.1177/0954410011414320
27. Chandra Murty, M.S.R. & Chakraborty, D. Effect of injection angle in mixing and combustion characteristics of scramjet combustor. *Int. J. Hypersonics*, 2011, **2**(1-2), 15-27. doi: 10.1260/1759-3107.2.1.15
28. Pannerselvam, S.; Thiagarajan, V.; Ganesh Anavardham, T.K.; Geetha, J.J.; Ramanuchari, V. & Prahlada. Airframe integrated scramjet design and performance analysis. ISABE Paper No. 2005- 1280, 2005.
29. Manna, P.; Dharavath, Malsur; Sinha P.K. & Chakraborty, Debasis. Optimization of a flightworthy scramjet combustor through CFD. *Aerospace Sci. Technol.*, 2013, **27**, 138-146. doi: 10.1016/j.ast.2012.07.005
30. Dharavath, M.; Manna, P.; Sinha P.K. & Chakraborty, D. Numerical analysis of a kerosene fuelled scramjet combustor. *ASME J. Thermal Sci. Eng. Appl.*, 2016, **8**(8), 0111003-1-7. doi: 10.1115/1-4030699

CONTRIBUTORS

Mr A. Bhandarkar obtained his MTech in Mechanical Engineering from IIT, BHU. Presently he is working as a Scientist at the Defence Research and Development Laboratory (DRDL), Hyderabad. His research areas include : Aerodynamic heating, droplet break-up study for high-speed flow, two phase flow analysis, CHT analysis, etc.
In current study, he has carried out grid generation, numerical simulation and analysis of the results for turbulent flow past circular cylinder and kinetic heating of nose cone.

Mr Malsur Dharavath obtained his ME (Aerospace Engineering) from Indian Institute of Science (IISc), Bengaluru and is working as a Scientist at the Defence Research and Development Laboratory (DRDL), Hyderabad. His research areas include Highspeed reacting and non-reacting flows in missile propulsion, scramjet propulsion, free-and confined -supersonic jets etc.
In current study, he has carried out grid generation, numerical simulation and analysis of the results of hydrocarbon fulled scramjet combustor with strut injection.

Mr M.S.R. Chandra Murty obtained his BTech (Mech Engg) from Regional Engineering College (REC), Warangal. He is currently working as Scientist in Computational Combustion Dynamics Division, Defence Research and Development Laboratory (DRDL), Hyderabad. His research interest includes heat transfer and computational fluid dynamics related to aerospace propulsion.
He has carried out grid generation, numerical simulation and analysis of the results of hydrogen fueled scramjet combustor, turbulent flow past circular cylinder and kinetic heating of nose cone.

Dr P. Manna obtained his PhD (Thermal Science and Engineering) from IIT, Kharagpur. Presently he is working as a Scientist in the Directorate of Computational Dynamics, DRDL, Hyderabad. His research interests include: CFD, propulsion, heat transfer, and high-speed reacting flow.
In current study, he has guided for the grid generation, numerical simulation and analysis of the results.

Dr Debasis Chakraborty obtained his PhD in Aerospace Engineering from Indian Institute of Science, Bengaluru. Presently, working as Technology Director, Computational Dynamics Directorate, DRDL, Hyderabad. His research interests are : CFD, aerodynamics, high-speed combustion, and propulsion.
In the current study, he has planned and guided the simulations and prepared the manuscript of the paper.