A Modified Wall Matching Treatment to Account for Local Solid to Fluid Thermal Coupling

Jeffery A. White
NASA Langley Research Center
Hampton, VA.

ABSTRACT

The wall-matching methodology of Wilcox is modified to include a solid-wall, thermal-conduction model. This coupled fluid-thermal-structure model is derived assuming that the wall thermal-structure behavior is locally one-dimensional and that structural deformations, due to thermally induced stresses, are not significant. The one-dimensional coupled fluid-thermal-structure model is derived such that the wall temperature is removed as an independent boundary condition variable. The one-dimensional coupled fluid-thermal-structure model is also derived for the general case of an arbitrary mixture of thermally prefect gases and a wall of arbitrary thickness and conductivity by using a compressible, streamwise-pressure-gradient-corrected, wall-matching function and Fourier's law of heat conduction. The resulting model was implemented in the VULCAN CFD code as a new boundary condition type. VULCAN was then used to simulate a two-dimensional Mach 6 wind tunnel facility nozzle flow to demonstrate/validate the one-dimensional coupled fluid-thermal-structure model. The nozzle internal-wall surface temperature and heat transfer distributions computed using the one-dimensional coupled fluid-thermal-structure model are compared to wall temperature and heat transfer distributions from an iterative multi-dimensional analysis obtained by coupling the VULCAN CFD code and the MSC/NASTRAN-thermal code. The one-dimensional coupled fluid-thermal-structure model analysis is shown to be very robust and in excellent agreement with the multi-dimensional iteratively coupled analysis. It is also shown that the one-dimensional analysis can be used as an initial guess for the multi-dimensional iteratively coupled analysis.

INTRODUCTION

The Hyper-X[(1)] and Advanced Space Transportation Programs (ASTP)[(2)] are currently investigating several airbreathing engine designs. The rocket based combined cycle (RBCC) and turbine based combined cycle (TBCC) engines of the ASTP and the Hyper-X program engine all incorporate scramjets as a component. These scramjet flowpaths are required to operate during the high Mach number (and high total enthalpy) portion of the vehicle flight trajectory. The operation of a scramjet engine at high total enthalpy levels causes the engine flowpath to be exposed to very high static temperatures, thereby presenting an extremely challenging thermal-structure design environment. At high total enthalpies, the near wall gas temperatures in the inlet and isolator exceed 2000 Kelvin and the combustor walls, fuel injector surfaces, flame holding devices as well as the nozzle walls are routinely exposed to gas temperatures in excess of 3000 Kelvin. The survival of structures utilizing available materials that are exposed to such high temperatures requires designs that are actively cooled. Furthermore, overall propulsion system thermodynamic-cycle efficiency and limitations on vehicle volume fraction constrain the vehicle design so as to require the use of the scramjet fuel as the structure coolant.

The use of actively cooled structures encourages the development of analysis tools that provide accurate prediction of the structure heat loads and cooling requirements. However, engineering design tools for the analysis of the thermal environment have been predominately composed of either lower order methods, that attempt to approximately couple the fluid and thermal analysis, or higher order methods, that exclude coupling effects and use worst case scenarios to bracket the expected heat loads. Recent work performed as a part of the development of the National Propulsion Simulation System (NPSS) by Suresh[(3)] *et al* demonstrated a high-fidelity steady-state, weakly-coupled fluid-thermal-structural solution methodology for scramjet flowpaths. This methodology iteratively coupled a CFD flow solver, Overflow[4], to a commercially available thermal-structural solver, ANSYS[5], with the assumption that the pressure and thermally induced deformations of the structure were negligible. The simulation that Suresh *et al* performed was an axi-symmetric analysis of the inlet of a GTX[3] engine module operating at Mach 4. Their fluid analysis began with an initial fluid-structure interface (wall) static temperature of 277.77 Kelvin and their coupled solution ultimately converged to a wall temperature distribution that varied between 290 Kelvin and 370 Kelvin. Suresh *et al* observed that the coupling process converged relatively quickly (< 10 coupling iterations). However, their initial wall temperature guess was relatively close to the final temperature, which raises the concern that the number of coupling iterations could be unacceptable, when the initial guess is poor. Since the coupling process requires multiple time consuming (and expensive) multi-dimensional fluid and thermal structural analyses, it is desirable to develop methodologies that provide a better initial wall temperature boundary condition. It is also desirable to develop a methodology that would reduce the envelope where the use of an iterative, multi-dimensional, coupled fluid-thermal-structure analysis would be required.

An approach that provides a better wall-temperature initial condition, and in some circumstances obviates the need for an iterative multi-dimensional coupled analysis, is to embed a one-dimensional thermal-structure analysis within the wall boundary condition used in the CFD analysis tool. The CFD code VULCAN[6], developed at NASA Langley Research Center specifically to analyze scramjet inlet, isolator, combustor and nozzle flows was chosen as the model development and demonstration platform due to its extensive use in scramjet flow analysis. VULCAN employs several techniques to reduce the total number of grid points that are required. One such technique is the use of the wall-matching function or “wall function” method of Wilcox[7,8]. This wall-matching function method assumes that the first cell center off the wall is in the log law layer and that the wall shear stress is obtained by solving a modified form of the law of the wall equation. Wilcox’s method also utilizes a one-dimensional form of the energy equation to solve for the wall heat flux or to specify the wall temperature. The one-dimensional form of the near-wall energy equation can be exploited to provide a method through which the wall thermal structure analysis may be introduced.

RESULTS AND DISCUSSION

The wall matching method originally introduced by Wilcox[7] and later revised[8] begins with a modified statement of the compressible law of the wall that includes streamwise pressure gradient effects,

$$u_1^* = u_\tau \left[\log_e\left(u_\tau y_1 / \nu_w\right) + \kappa B + c_1 \phi\left(u_\tau y_1 / \nu_w\right)\right] / \kappa \,, \tag{1}$$

where the streamwise pressure gradient term is defined as,

$$\phi = \nu_w \left({dp}/{dx} \right)_w \Big/ \left(\rho_w u_\tau^3\right) \,, \tag{2}$$

and the wall friction velocity is,

$$u_\tau = \sqrt{\tau_w / \rho_w} \,, \tag{3}$$

where $\rho_w, \nu_w,$ $(dp/dx)_w$ and τ_w are the wall density, kinematic viscosity, pressure gradient, and shear stress magnitude, respectively. The constants have the following values: c_1 = -1.13, κ = 0.41 and B = 5. In addition, because the model is to be implemented in VULCAN (a cell centered code), the variables in equation (1) with a subscript of 1 refer to values taken from the first cell center adjacent to the wall.

Therefore, y_1 is the distance from the wall to the first cell center adjacent to the wall and u_1^* is the effective velocity tangent to the wall, in the first cell center adjacent to the wall. Wilcox[7] presents u_1^* as,

$$u_1^* = \frac{1}{a}\left[\sin^{-1}\left(\frac{2a^2u_1 - b}{\sqrt{b^2+4a^2}}\right) + \sin^{-1}\left(\frac{b}{\sqrt{b^2+4a^2}}\right)\right] \quad , \tag{4}$$

with

$$a^2 = \mathrm{Pr}_t \Big/ \left(\overline{C_{p\,w}}\,T_w\right) \quad , \tag{5}$$

and

$$b = -a^2\left(\frac{q_w}{\tau_w}\right) \quad , \tag{6}$$

where $\mathrm{Pr}_t, q_w, \overline{C_{p\,w}}$ and T_w are the turbulent Prandtl number, wall heat flux, effective specific heat at constant pressure (evaluated at the wall temperature) and the wall temperature, respectively, and u_1 is the velocity tangent to the wall, in the first cell center adjacent to the wall.

Wilcox[7] presents the one-dimensional energy equation for the wall energy balance (generalized here for a frozen mixture of thermally prefect gases) as,

$$\overline{C_{p1}}T_1 - \overline{C_{p\,w}}T_w = -\mathrm{Pr}_t\left[\left(\frac{q_w}{\tau_w}\right)u_1 + \frac{1}{2}u_1^2\right] \quad , \tag{7}$$

where T_1 is the static temperature in the first cell center adjacent to the wall and $\overline{C_{p_1}}$ is the effective specific heat at constant pressure (evaluated at the static temperature in the first cell center adjacent to the wall). Equation (7) is then solved to yield an expression for T_w, (which reduces to the adiabatic wall temperature if q_w is set to 0),

$$T_w = \left\{\overline{C_{p1}}T_1 + \mathrm{Pr}_t\left[\left(\frac{q_w}{\tau_w}\right)u_1 + \frac{1}{2}u_1^2\right]\right\} \Big/ \overline{C_{p\,w}} \quad , \tag{8}$$

or to yield an expression for q_w, (which yields the wall viscous energy flux for a specified value of T_w),

$$q_w = \tau_w\left[\frac{\left(\overline{C_{p_w}}T_w - \overline{C_{p_1}}T_1\right)}{\left(\mathrm{Pr}_t\, u_1\right)} - \frac{1}{2}u_1\right] \quad . \tag{9}$$

MODIFICATION OF THE WALL MATCHING ENERGY EQUATION

Fourier's one-dimensional law of heat conduction[9] in differential form is,

$$q_k = -k\frac{dT}{dy} \quad , \tag{10}$$

and is written for a planar wall, in discreet form as,

$$q_k = -\frac{k_w}{l_w}\left(T_w - T_e\right) \quad , \tag{11}$$

where k_w is the thermal conductivity of the wall material, l_w is the wall thickness and T_e is the wall external temperature as shown in Figure 1.

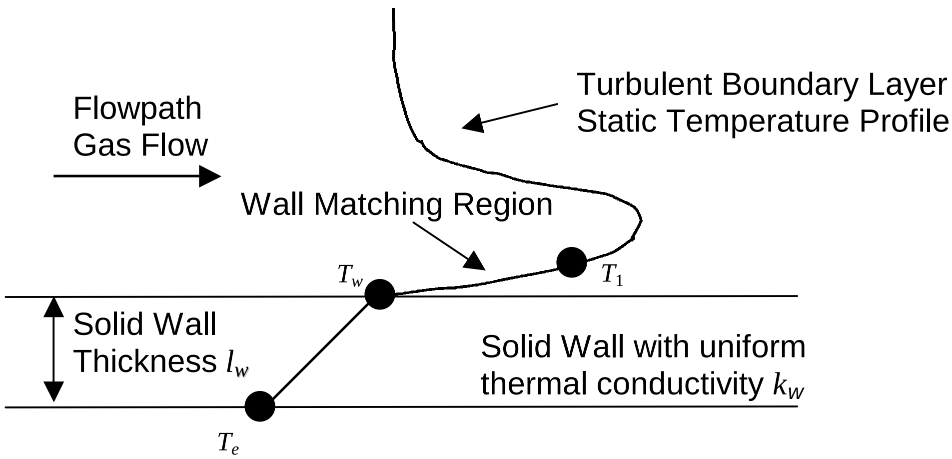

Figure 1. - One-dimensional conduction through a plane wall with forced convection.

Recognizing that, at steady state, the conduction heat transfer through the wall, q_k, balances with the convection heat transfer from the gas to the wall, q_w, yields,

$$q_w = q_k = -\frac{k_w}{l_w}\left(T_w - T_e\right) \quad , \tag{12}$$

which is solved for T_w to give,

$$T_w = T_e - \frac{l_w}{k_w} q_w \, . \tag{13}$$

Substitution of equation (13) into equation (8) results in an equation that no longer requires that the flowpath gas-side wall temperature be known. The resulting equation can then be simplified to give,

$$q_w^c = \left[\frac{\left(\overline{C_{p_w}} T_e - \overline{C_{p_1}} T_1\right)}{\Pr_t} - \frac{1}{2} u_1^2 \right] \Bigg/ \left[\frac{\overline{C_{p_w}}}{\Pr_t} \left(\frac{l_w}{k_w} \right) + \frac{u_1}{\tau_w} \right]. \tag{14}$$

Equation (14) implicitly couples the wall conduction and gas-side wall convection processes and is used instead of equation (9) when imposing the wall-matching function viscous-energy-flux boundary condition. Note that the superscript c has been added to indicate that the energy flux is now coupled to the wall conduction. The coupled wall-matching function process is then imposed on the gas-side flow solution through the following steps:

1) An initial guess is created for the wall friction velocity, u_τ.
2) Equations (2), (4), (5) and (6) are used to construct the components of equation (1) and are iteratively solved, to obtain u_τ, using Newton's method.
3) Equation (3) is used to compute the magnitude of the wall shear stress, τ_w.
4) Equation (14) is used to compute the wall viscous energy flux due to coupled conduction and convection, q_w^c.
5) The wall shear stress magnitude, τ_w, is used in the viscous momentum flux routines to impose the wall momentum flux.
6) The wall energy flux due to coupled conduction and convection, q_w^c, is used in the energy equation viscous flux routine to impose the wall energy flux.
7) Equation (13) is then used to provide the wall temperature for flow solution post-processing and visualization purposes.

PROOF OF CONCEPT TEST CASE

In order to test the wall-matching function coupling procedure, a relatively simple geometry that creates large, spatially varying, wall heat transfer loads is required. This requirement is satisfied by solving the flow through a two-dimensional convergent-divergent nozzle operating with high values of inflow total enthalpy and cooled walls. This nozzle flow solution is computed using two different approaches and the wall temperature and heat transfer distributions are compared. The first approach uses the one-dimensional coupled wall-matching function procedure implemented in the VULCAN code. The second approach uses a multi-dimensional coupled analysis similar to the one described by Suresh *et al.* However, in this instance, the VULCAN code, using an uncoupled wall-matching function method with a specified wall temperature distribution boundary condition, is iteratively coupled with the MSC/NASTRAN-thermal[10] analysis code.

The test case selected was a two-dimensional nozzle design under consideration as a candidate to replace the existing three-dimensional Langley Arc-Heated Scramjet Test Facility[11] nozzle. The nozzle was computed using a single block two-dimensional grid consisting of 417 points in the i-direction and 65 points in the j-direction. As shown in Figure 2, only one half of the nozzle height was modeled. The inflow boundary, located at the i-minimum boundary of the grid, was modeled as a subsonic inflow boundary by specifying the nozzle stagnation conditions. The outflow boundary, located at the i-maximum boundary of the grid, was modeled as an extrapolation boundary. The j-minimum boundary was modeled as a symmetry plane and the no-slip wall, located at the j-maximum boundary of the grid, was modeled using either a one-dimensional coupled wall-matching function or a specified temperature distribution wall-matching function.

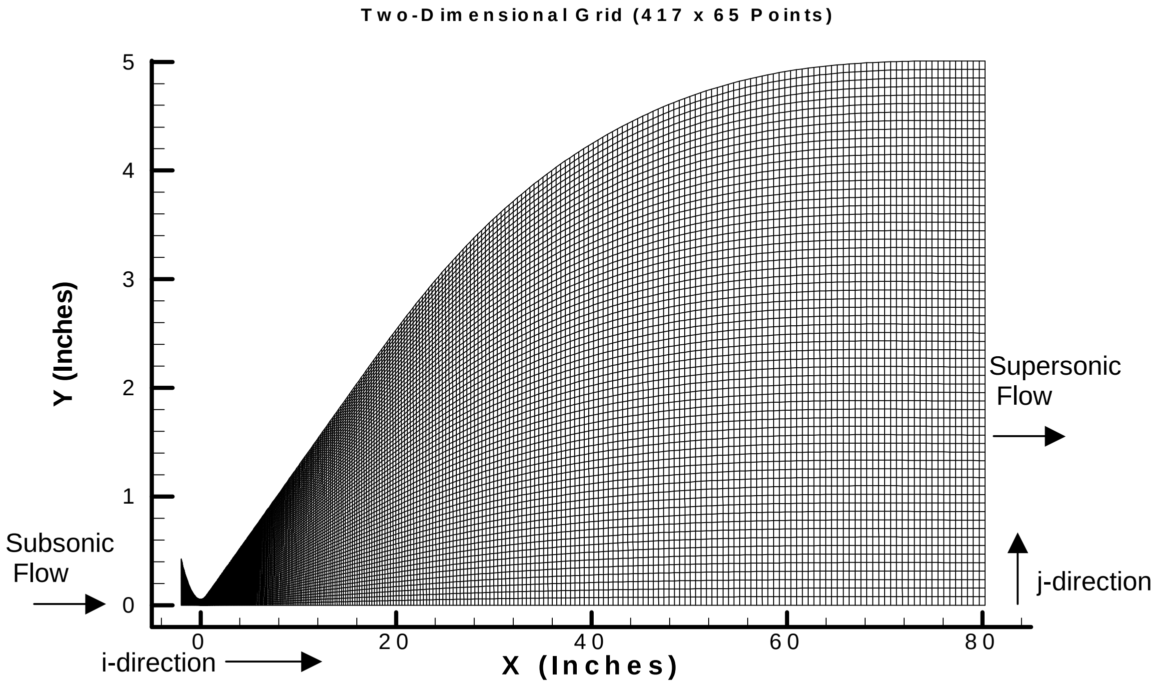

Figure 2. - Two-dimensional computational grid.

The inflow boundary condition was modeled as a mixture of four chemical species and the flow chemistry was assumed to be chemically frozen throughout the expansion process. The tunnel plenum conditions were used to specify the inflow pressure and temperature as well as the gas composition and are presented in Tables 1 and 2, respectively.

Stagnation Pressure	35.384 atmospheres
Stagnation Temperature	2159.5 Kelvin

Table 1. - Nozzle plenum/inflow pressure and temperature.

Chemical Species	N_2	O_2	Ar	NO
Mass Fraction	0.7450	0.2188	0.0124	0.0238

Table 2. - Nozzle plenum/inflow gas composition.

The nozzle calculations were run using the two-equation $k-\omega$ turbulence model of Wilcox[8] with the flow assumed to be fully turbulent. A typical nozzle Mach contour plot resulting from these analyses, as presented in Figure 3, shows a well-designed nozzle flow with uniform outflow Mach number and an exit boundary layer thickness that is approximately 32 % of the nozzle exit height. The wall

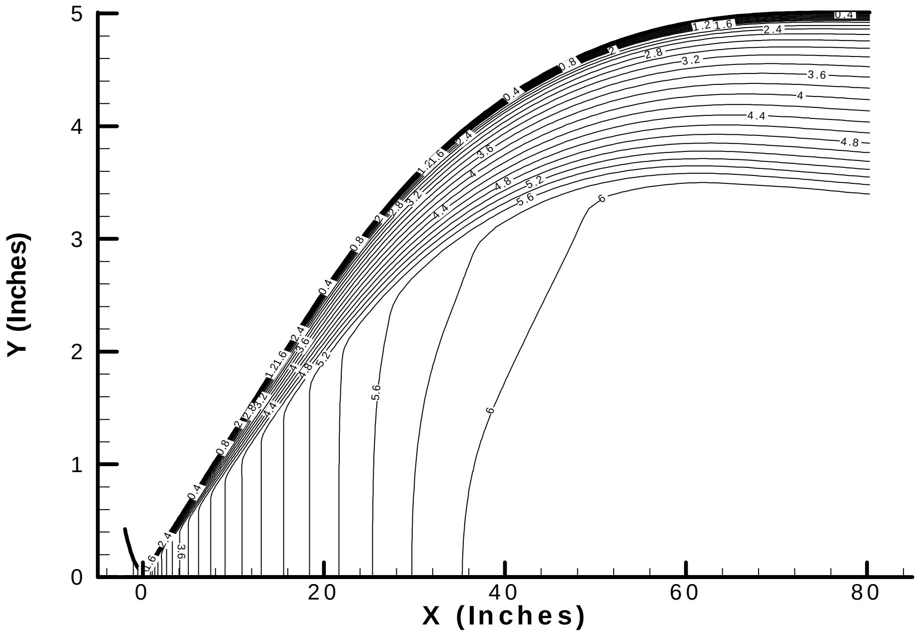

Figure 3. - Typical nozzle Mach number contours.

thermal analysis began by simulating the nozzle flow using a constant wall temperature of 500 K. Figure 4 presents the computed wall heat transfer distribution. The computed heat transfer

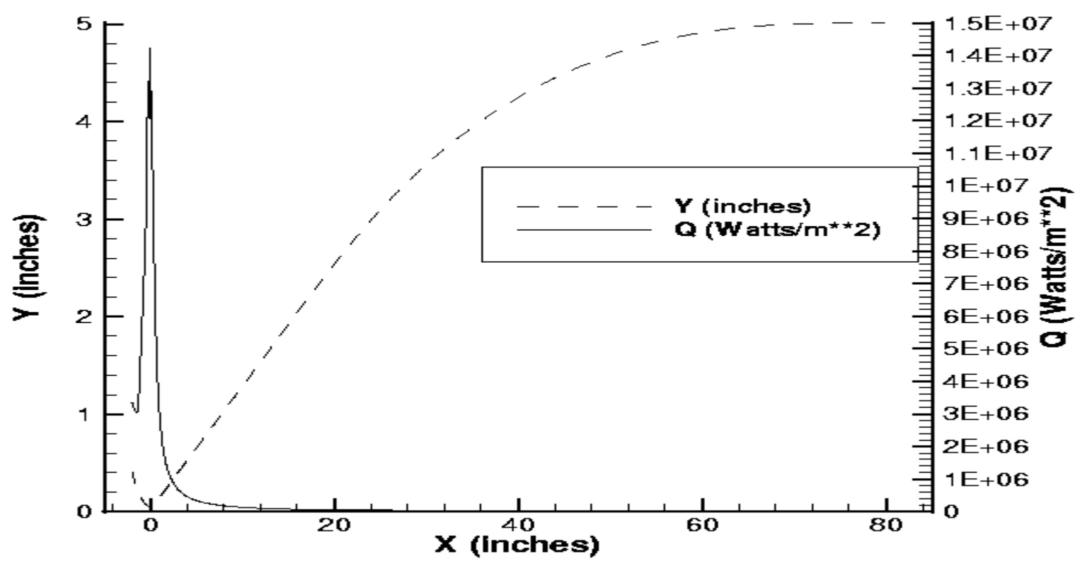

Figure 4. - Computed nozzle wall heat transfer (Q) distribution for the isothermal wall case.

reaches a peak of 1.5×10^7 Watts per meter squared at the nozzle throat. This heat transfer distribution was converted into an input file usable by MSC/NASTRAN-thermal and then a multi-dimensional (in this case 2-D) thermal analysis was performed to obtain the temperature distribution on the nozzle fluid-side wall. For simplicity, the nozzle wall was modeled as a constant thickness wall of 0.25 inches (6.35 mm.), with a uniform thermal conductivity of 14.4 Watts/Meter-Kelvin (a value typical of mild steel), and a constant external temperature of 300 Kelvin. Once the multi-dimensional heat transfer problem was solved, the fluid-side wall temperature distribution was output from MSC/NASTRAN-thermal and converted into a VULCAN profile file[12]. The resulting temperature distribution illustrated a difficulty with using a poor initial temperature guess in the flow solver. The computed wall temperature was found to be significantly greater than the flow stagnation temperature. Ultimately, under-relaxing the temperature distribution obtained from the wall thermal analysis code during the initial iterations of the multi-dimensional coupling process alleviated this undesirable behavior. However, this under-relaxation increased the number of iterations required to converge the coupled analysis.

Alternatively, the nozzle flow was computed using the one-dimensional coupled wall-matching

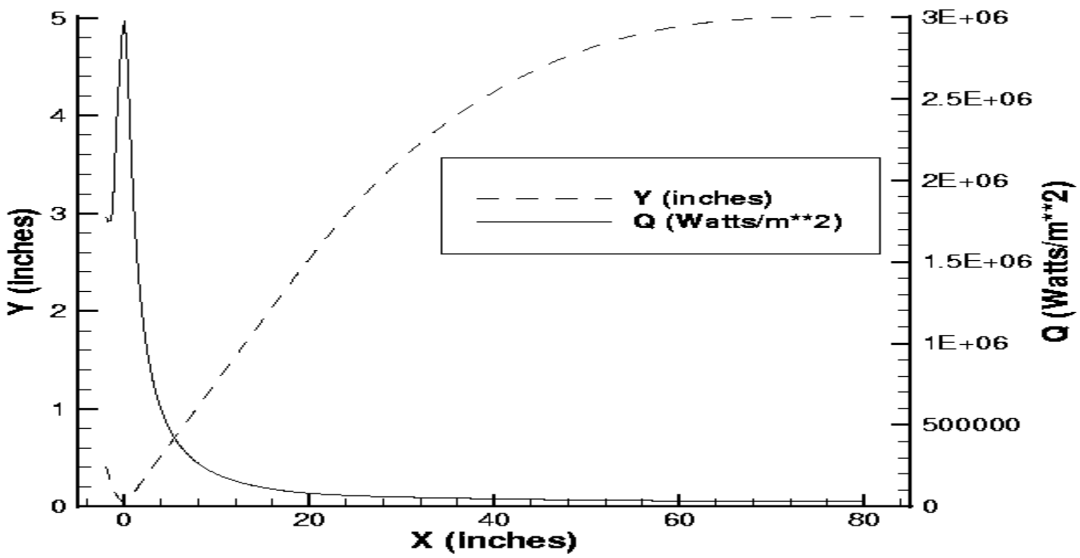

Figure 5. - Computed nozzle wall heat transfer (Q) distribution for the one-dimensional coupled wall matching function model.

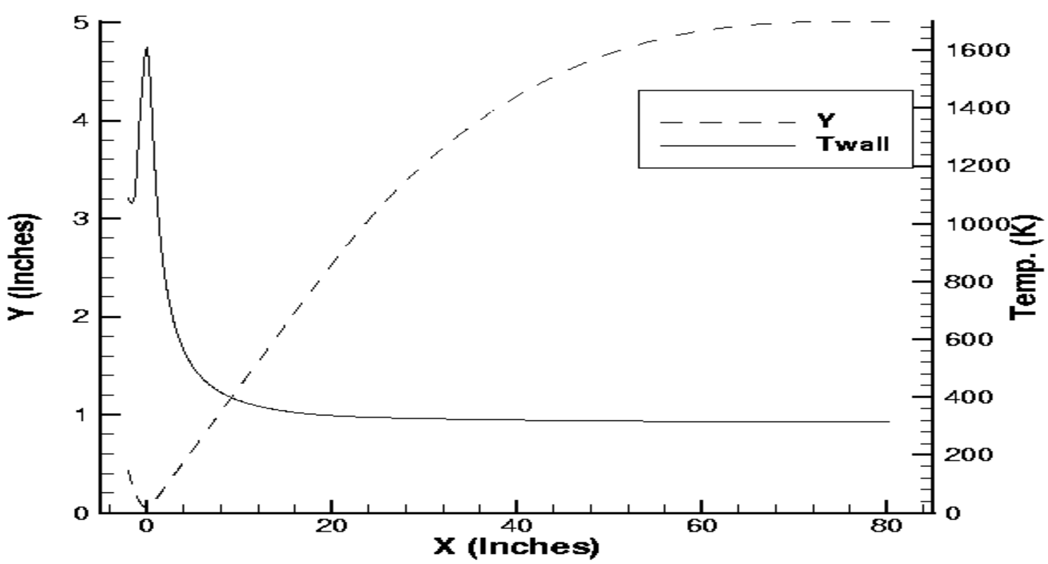

Figure 6. - Computed nozzle wall temperature distribution for the one-dimensional coupled wall matching function model.

function model given by equation (14). Figures 5 and 6 present the computed wall heat transfer and wall temperature distributions, respectively. Figure 5 shows that the nozzle throat computed wall heat transfer is approximately one fifth of the level developed by the iso-thermal wall temperature case. The reason for this is determined through inspection of the computed wall temperature distribution. Figure 6 shows that the computed wall temperature distribution reaches a maximum value of approximately 1600 Kelvin, which is approximately 5 times the assumed wall temperature used to perform the iso-thermal wall case. Note that the wall temperature for the majority of the nozzle length was computed to be approximately 314 Kelvin, which is physically consistent with the assumed external surface temperature of 300 Kelvin.

The computed wall heat transfer distribution obtained from the one-dimensional coupled wall-matching function model was then converted into an input file usable by MSC/NASTRAN-thermal and an iterative multi-dimensional analysis was initiated to study the feasibility of using the coupled one-

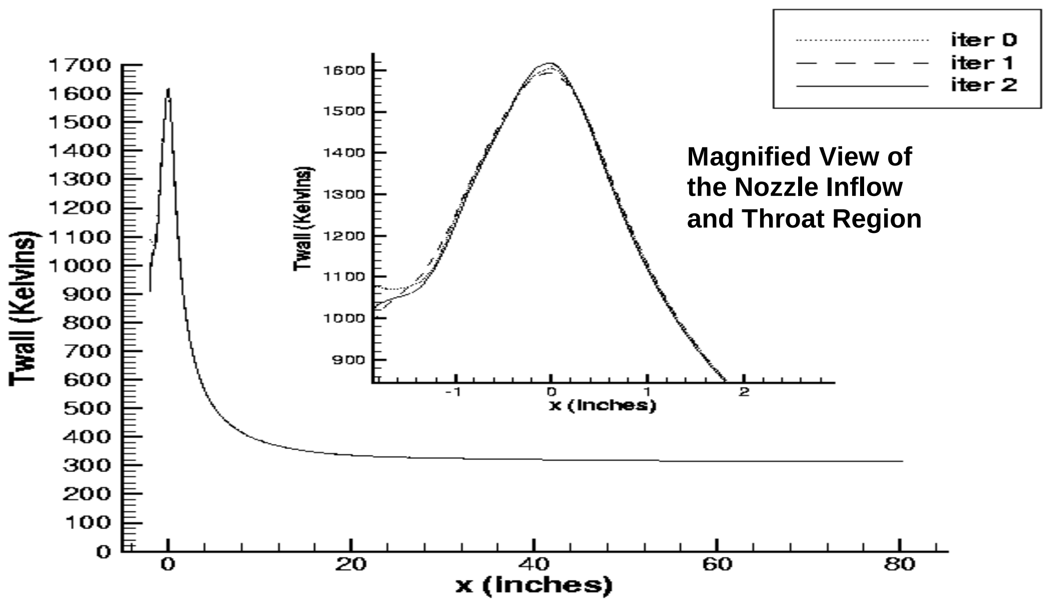

Figure 7. - Multi-dimensional iterative method wall temperature distribution evolution.

dimensional method as an initial guess for the multi-dimensional iterative method. The evolution of the wall temperature distribution during the iterative process is presented in Figure 7 for the initial one-dimensional coupled wall-matching function model method as well as 2 subsequent multi-dimensional coupled method iterations. The computed temperature distribution after iteration 2 can be seen to be very similar to the distribution computed using the one-dimensional coupled wall matching function model (iteration 0). Examination of a magnified view of the nozzle inflow and throat region shows that the differences between the iterations were less than 20 Kelvin at the nozzle throat, on the order of 80 Kelvin at the nozzle inflow, and less 5 Kelvin for the remainder of the nozzle. The differences between the one-dimensional and multi-dimensional methods at the nozzle inflow and throat are attributable to multi-dimensional effects. At the nozzle inflow the additional surface area of the leading edge of the wall that is in contact with the 300 Kelvin external temperature pulls more energy from the wall. This effect is completely ignored by the one-dimensional method. A similar observation can be made of the throat region of the nozzle. The throat axial temperature gradients on the fluid-side wall (and thus in the wall material) are large and cause significant conduction to take place. Again this effect is missed due to the nature of the one-dimensional coupled wall- matching boundary condition. However, the overall agreement between iteration 0 and iteration 2 is excellent over the majority of the nozzle wall, demonstrating the utility of the one-dimensional method as an initial guess. This observation is reinforced by the heat transfer distribution after iteration 2, presented in Figure 8. When Figure 8 is compared

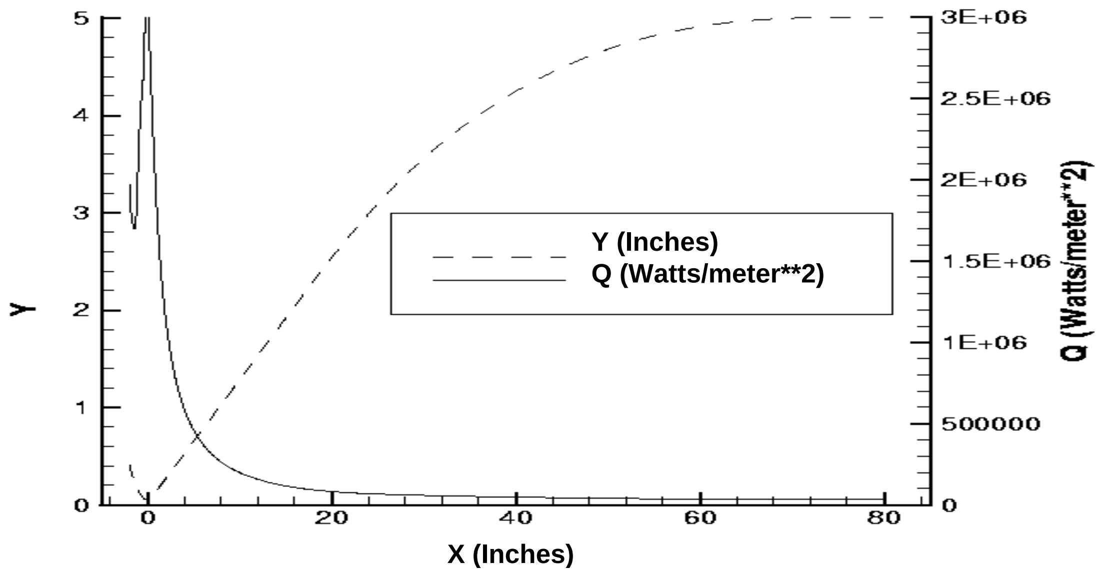

Figure 8. - Multi-dimensional iterative method wall heat transfer (Q) distribution after iteration 2.

with Figure 5 the coupled one-dimensional wall matching model wall heat transfer distribution is seen to also be in close agreement with the iteratively coupled model wall heat transfer.

SUMMARY AND CONCLUSIONS

The wall matching methodology of Wilcox was modified to include a one-dimensional solid-wall thermal-conduction model. This coupled fluid-thermal-structure model was derived assuming that the wall thermal-structural behavior is locally one-dimensional and that structural deformations due to thermally induced stresses were not significant. The resulting model was implemented in the CFD code VULCAN as a new boundary condition type such that the wall thickness and thermal conductivity are either held constant or varied over the boundary surface. VULCAN was used to simulate a two-dimensional Mach 6 wind-tunnel facility-nozzle flow. The predicted nozzle internal-wall surface temperature and heat transfer distributions were compared to the wall temperature and heat transfer distributions computed using a multi-dimensional analysis performed by iteratively coupling the VULCAN CFD code with the MSC/NASTRAN-thermal code in a manner similar to the approach of Suresh, *et al*. The one-dimensional analysis was found to be very robust and in excellent agreement with the multi-dimensional iteratively coupled analysis. It was also found that the one-dimensional analysis provides a much better initial guess for the multi-dimensional iteratively coupled analysis than the traditional iso-thermal guess, thereby reducing the number of iterations required to converge the multi-dimensional analysis.

AKNOWLEDGEMENTS

The author would like to thank Dr. Richard Gaffney of the Hypersonic Airbreathing Propulsion Branch of the NASA Langley Research Center for providing the grid and inflow conditions for the two-dimensional nozzle from his Arc Heated Scramjet Test Facility nozzle redesign study. Special thanks are due to Dr. Kim Bey of the Metals and Thermal Structures Branch of the NASA Langley Research Center for providing the MSC/NASTRAN-thermal analysis results used as part of the multi-dimensional iterative coupling analysis.

REFERENCES

1) Freeman, D.C., Reubush, D.E., McClinton, C.R., Rausch, V.L. and Crawford, J. L., "The NASA Hyper-X Program", 48th International Astronautical Conference, October 1997.
2) Cockrell, C.E. Jr., Capriotti, D.P., Glass, D.E., Rock, K.E. and Rogers, R.C., " Ground Testing and Technology Development Requirements for Dual-Mode Scramjet Flowpath Development," 38th JANNAF CS/APS/PSHS/MSS Joint Meeting, April 2002.
3) Suresh, A., Liou, M.-S. and DeBonis, J.R., "Multidisiplinary Analysis of a Supersonic Engine Inlet," FEDSM2001-18137, Proceedings of FEDSM'01, ASME, May, 2001.
4) Buning, P.G., Jesperson, D.C., Pulliam, T.H., Klopfer, W.M., Chan, W.M., Slotnick, J.P., Krist, S.E., and Remeze, K.J., "Overflow Users Manual," Version, 1.8j, February 1999.
5) Analysis Systems, Inc., "ANSYS Users Manual," Version 5.6, 1999.
6) White, J. A. and Morrison, J. H., "A Pseudo-Temporal Multi-Grid Relaxation Scheme for Solving the Parabolized Navier-Stokes Equations," AIAA Paper No. 99-3360, June 1999.
7) Wilcox, D.C., "Wall Matching, A Rational Alternative to Wall Functions," AIAA 89-0611, January 1989.
8) Wilcox, D.C., <u>Turbulence Modeling for CFD</u>, 2nd Edition, DCW Industries, Inc. 1998.
9) Kreith, F. and Black, W.Z., <u>Basic Heat Transfer</u>, Harper & Roe, Publishers, 1980.
10) The MacNeal-Schwendler Corporation, "Getting Started With MSC/NASTRAN," 2nd Edition, October 1996.
11) Guy, R.W., Rogers, R.C., Puster, R.L., Rock, K.E. and Diskin, G.S., "The NASA Langley Scramjet Test Complex," AIAA Paper No. 96-3243, 32nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference, July 1996.
12) White, J.A., "VULCAN Users Manual," Version 4.3, March 2002.