Effect of Transverse Hydrogen Injection on Cooling of Hypersonic Vehicle Surfaces

D.E. Musielak
University of Texas - Arlington
Arlington, Texas

Abstract

A study was made of a scramjet inlet to determine the effects of injecting hydrogen fuel on convective heat transfer. This work is part of a research effort to simulate transverse gas injection into supersonic airstreams, using a computer code that solves the Navier-Stokes equations, with both laminar and turbulent flow schemes, fully coupled with a multi-species, multi-equation kinetics reaction model. Slot fuel injection is modeled at various hydrogen jet conditions and different injection angles. Results from this study suggest that premixing fuel and air at the inlet of a scramjet engine, if properly controlled, can reduce the heat transfer to the inlet surfaces. Other effects of transverse fuel injection such as ignition potential and flow separation were also investigated.

List of Symbols

C_h	wall heat transfer coefficient
d	slot size
h	height of computational domain
H_2	hydrogen fuel
L	length of computational domain
m	mass flow
P	gas pressure
Pr	Prandtl Number
q	heat flux
T	gas temperature
U	streamwise velocity
u,v	cartesian velocity components
x,y	cartesian coordinates
δ	boundary layer thickness
κ	coef. of thermal conductivity
ρ	gas density
ϕ_{H2}	equivalence ratio for cooling

Subscripts

∞	inlet air conditions
a	air
aw	adiabatic wall
f	fuel
w	wall

1. Introduction

An inherent characteristic of hypersonic vehicles is the extreme thermal loads which result from the very large freestream stagnation enthalpies. These thermal loads can heat known materials beyond their allowable working temperatures, leading to the need for wall cooling. Wall heat transfer will take place by conduction from the boundary layer gas. This convective heat transfer, or aerodynamic heating, is absorbed by the coolant that flows on the cooled side of the wall. In hypersonic vehicles, the coolant is usually fuel that is later injected into the engine's main flow.

For thermal boundary layers, preliminary estimates of convective heat transfer can be made using a relationship between the heat flux and the product of a conductance and an enthalpy difference. This relationship suggests that materials with the highest wall temperature should be used to minimize the convective heat transfer. However, even the most advanced materials have maximum working temperatures that are much lower than the adiabatic wall temperature encountered in hypersonics. For example, for vehicles flying at a Mach number of 10, adiabatic wall temperatures on the order of 6,000 R (3333 K) are expected[1].

In the 1960's, a high-altitude scramjet engine with premixing of fuel and air at the inlet diffuser, was first proposed[2-5]. Based on experiments and calculations, results indicated that injecting fuel at the inlet could be used to mix fuel with the freestream air prior to entering the combustion zone. The main advantages of premixing were expected to include reduction in combustor length, reduction in skin friction drag, and cooling of the inlet diffuser walls. Furthermore, researchers suggested using streamwise inlet injection, stating that premixing fuel and air would lead to better control of fuel distribution, by selecting the location of the injectors at an upstream point in the inlet of the engine[3-4]. Streamwise injection is desirable because of its potential for low total pressure losses. However, this injection technique is capable of only low fuel penetration and, therefore, is low in overall mixing efficiency. Conversely, with transverse injection one can achieve high fuel penetration and higher mixing efficiency, at the expense of large pressure losses and higher internal drag. One can also argue that streamwise injection would result in lower heat loads since the cold fuel will diffuse closer to the wall. This, however, is the issue that is addressed in this paper.

The current work is part of a research program on supersonic reacting flows, to simulate hydrogen fuel-air injection, mixing, and ignition, for application to a pre-mixed, shock-induced combustion-type hypersonic engine. The main goal of this investigation is to study the effects of transverse hydrogen injection into supersonic airflow, including cooling effects. As reported in Refs. 6 and 7, after examining the interaction of the fuel jet with the freestream flow, it was found that the injected cold fuel tends to cool the boundary layer, therefore affecting the amount of convective heat transfer to the wall of the scramjet inlet. This led us to explore the validity of the claim that pre-mixing fuel in the upstream section of a scramjet can result in cooling. The results obtained in this study also apply to other problems in which control of heat transfer to the walls is important. Because experimental facilities for hypersonic flows are not yet available, a computational modeling approach is the most readily available tool to obtain an accurate prediction of the interaction between the supersonic freestream air flow and the injected fuel.

The computer code selected for this study solves the Navier-Stokes equations, using both laminar and turbulent flow schemes, coupled with a 7-species, 7-reaction hydrogen-air kinetics model. The code was adapted to compute mainly transverse gas injection in supersonic air streams, to simulate the thermophysical processes at the inlet section of an airbreathing hypersonic engine.

2. Approach and Analysis

In order to simulate the interaction of the transverse fuel jet with the freestream airflow, a simple flat plate with a slot-wall injector was chosen as representative of the scramjet inlet. Figure 1 illustrates this physical model. The plate is 0.3048 m long, and the injector is located at x = 0.10 m. The transverse dimension of the computational domain is h = 0.0762 m.

To simulate the finite-rate chemically reacting flow for the above model, a new code, developed by J. White[8] under contract with NASA Langley Research Center, was adapted. The Langley Algorithm for Research in Chemical Kinetics, LARCK, has built-in a number of diffusion, reaction, and flow models which makes the program easily tailored to many applications. For this study, both laminar and turbulent flow models are used.

The equations governing laminar flow of a multi-species gas mixture are the Navier-Stokes equations augmented with species continuity equations. These equations are solved by LARCK assuming that the gas mixture is described by the perfect gas law. The heat flux vector, q_j, is modeled with Fourier's law,

$$q_j = - \kappa\ \partial T/\partial x_j + \rho \sum f_n v_{nj} h_n$$

where κ is the coefficient of thermal conductivity, f_n is the mass fraction, v_{nj} is the diffusion velocity of species *n* in the x_j direction, and h_n is the enthalpy of species *n*. The coefficient of viscosity for each species in the mixture is calculated using Sutherland's law, and the coefficient of thermal conductivity is computed using the Prandtl number relationship, $\kappa = \mu\ C_p/Pr$.

The equations governing turbulent mean flow are the Reynolds-averaged Navier-Stokes equations. LARCK has the option of using either one-equation or two-equation turbulence models. For this investigation, Menter's k-ω turbulence model provided the best results. Thus, all reported data are based on this model. For a complete description of this turbulence model, please consult References 9 and 10.

The flow and boundary conditions chosen for this investigation are shown in the following Table 1.

Let us consider a simple model of the scramjet inlet as a square domain of side h and length L (Fig. 1). Hydrogen fuel is injected through a slot of diameter d, on the lower surface, which is at a uniform temperature T_w. The temperature of the fuel at the injection point is T_{H2}.

The total hydrogen mass flow emanating from the slot is given by

$$m_{H2} = \rho_{H2}\, u_{H2}\, h\, d$$

and the air mass flow is

$$m_a = \rho_a\, u_a\, h^2$$

The equivalence ratio for cooling, ϕ_{H2}, is defined as

$$\phi_{H2} = m_{H2}/(f\,m_a) = m\,(d/h)/f$$

where f is the stoichiometric fuel-air ratio ($f = 0.0293$) and m is the propellants mass flux ratio ($m = \rho_{H2}\,u_{H2}\,/\rho_a\,u_a$). For a flow domain of 0.0762 m height,

$$\phi_{H2} = 447.9\ m\ d$$

The fuel is injected at two different density conditions resulting in two values of m. Also, two injector sizes are evaluated; therefore, the equivalence ratio for cooling is varied from 0.559 to 4.396, as shown in Table 2.

3. Results and Discussion

The character of the undisturbed boundary layer is illustrated by the temperature profile in Figure 2. At the slot position ($x = 0.10$ m), the temperature of the gas has a well defined transverse distribution, with peak temperatures of 1846 K and 1886 K for the turbulent and laminar boundary layers, respectively. As shown, the peak temperature values are found at $y/\delta = 0.219$ and 0.286 for turbulent and laminar flow respectively.

Without injection, the total heat transfer for the 30.46 cm long plate is computed as 3.663×10^5 and 2.882×10^5 W/m^2, with turbulent and laminar flow respectively.

Effect of Fuel Injection on Heat Transfer

The wall heat transfer is reduced dramatically by the effect of the hydrogen jet. The magnitude of the calculated heat fluxes with turbulent flow is greater than that for laminar flow cases. Also, the cooling effect of the injected hydrogen is greater if the flow is turbulent.

In general, the best cooling is achieved by injecting tangentially the smallest amount of coolest hydrogen fuel. As shown in Table 2, the heat transfer is reduced almost 57 percent with hydrogen injected parallel to the streamwise direction. Also note that, at the injection angle of 20°, for turbulent flow cases 1a and 3c, the total heat transfer rate is approximately the same, but the heat transfer rate downstream from the injector is fifty percent lower with the smallest slot diameter.

The nature of the undisturbed boundary layer for the turbulent flow in the presence of the cooled wall is characterized by a hot sublayer with temperatures up to 2000 K immediate above the cooled viscous layer, as illustrated by the temperature distribution in Figure 2. With hydrogen injection, the gas temperature is lowered by a few hundred Kelvin due to mixing with the relatively cold hydrogen jet.

Laminar Flow - Isothermal Wall.

The structure of the laminar airflow with transverse hydrogen injection is illustrated in Figure 3. As shown, due to the interaction of the fuel jet with the incoming airflow, the gas temperature near the wall, downstream from the injector, has been cooled considerably. This results in a reduction of the heat flux as illustrated in Figure 4, where the effects of fuel temperature and injection angle are included.

Turbulent Flow - Isothermal Wall.

The heat flux distribution follows a profile similar to that of the wall pressure, as shown in Figures 5 and 6 for streamwise and transverse injection respectively. Flow separation begins where the first increase in wall pressure is observed. The flow separates upstream of the injector due to the interaction of the shock with the boundary layer, and it re-attaches at a point downstream, dependent on the jet angle. Correspondingly, at the point of separation, the heat transfer begins to decrease. The minimum value of the heat flux, q_w, represents the lifting of the boundary layer upstream from the injector.

The greatest reduction of heat transfer occurs when hydrogen is injected tangentially. Choosing the same fuel condition and same injector diameter, the heat transfer to the wall is reduced by 51 percent for 30° injection, and by 57 percent with a 0° jet. The effect of injection angle on the total heat transfer to the wall is illustrated in Figure 7. The distribution of heat flux downstream from the slot for the various injection angles is given in Figure 8.

Adiabatic Wall.

The large stagnation enthalpy of the flow in the boundary layer will heat the wall material beyond its allowable working temperatures. The undisturbed boundary layer (no injection) with turbulent flow is characterized by its large thickness which continues growing along the length of the plate. At the axial location that corresponds to the injection point, the boundary layer thickness is approximately 0.0056 m, and at the exit plane this thickness has increased by a factor of 2.4. The adiabatic wall temperature ranges from 5560 K to 7094 K, with the largest values at the plate's leading edge.

Preliminary estimates of convective heat transfer, for thermal boundary layers with variable properties, are based on an equation that represents the heat flux at the surface, q_w, as the product of a conductance and an enthalpy difference. A heat transfer coefficient, C_h, is defined as the ratio $q_w/[\rho u c_p(T_{aw}-T_w)]$. Two computations are needed to determine C_h, i.e., an isothermal wall case and an adiabatic wall case. For the isothermal case the wall temperature is set to 290 K, and the local heat transfer $q_w(x)$ is computed. For the adiabatic case, zero wall temperature gradient, $\partial T/\partial y = 0$, is used as the boundary condition, and the local adiabatic wall temperature $T_{aw}(x)$ is computed. Results of this calculation are illustrated in Figure 9.

The nature of the laminar boundary layer flow in the presence of an adiabatic wall is largely characterized by the hot sublayer with temperature up to 7000 K at the leading edge of the plate. Case 3b was chosen to simulate fuel injection through this laminar boundary layer. The adiabatic wall temperature averages 5996 K along the first 10 cm the plate's length. However, immediately upstream and downstream from the injector, the wall temperature is lowered by several hundred Kelvin due to the effect of the relatively cold hydrogen jet. The average adiabatic wall temperature over the last 20 cm of the plate is 4804 K, reflecting the cooling effect of the hydrogen injection.

Naturally, this flow condition would not be practical due to the extreme temperatures the material walls would experience. It is included here only to help establish the effect of the injected fuel on the heat loads. The computed adiabatic wall temperature is also used in the calculation of the convection heat transfer coefficient or Stanton Number, as explained above.

Pre-Ignition Potential

The potential for pre-ignition at the engine's inlet was the focus of earlier papers (Refs. 6-7). It was reported that, at the conditions of this simulation, ignition of the hydrogen-air mixture at the injection point is not likely. Shock created a sudden static temperature rise in the airstream and the magnitude of the temperature exceeds the ignition temperature of hydrogen. However, the production of water molecules is negligible at the injection point. The maximum mass fraction of H_2O is found at the exit plane of the flow domain. Thermal choking and flow separation were also observed.

4. Conclusions

Injecting cold hydrogen fuel into a supersonic airflow at Mach 6.3 results in cooling of the surfaces, with a reduction of heat transfer rates up to 57 percent. The best cooling is achieved by injecting the smallest amount of hydrogen fuel at the lowest temperature. Injection angle seems to have some effect on the magnitude of heat transferred, however, the lowest value of heat flux was obtained with parallel injection.

This study was intended as a preliminary exploration of the effect of hydrogen injection on the convective heat transfer at the inlet of a hypersonic airbreathing engine.

The results presented herein covered flow conditions where no experimental data are yet available, thus, these computational calculations should be used with some reservation. However, these results provide needed insight into the complex physics characterizing hypersonic chemically reacting flows. These results are also very useful in making comparisons, in determining trends, and in establishing the limitations of the computer code being developed. To validate the CFD code used here, a comparison with limited data reported in the literature is made and reported in a separate paper. This comparison gives the present results the degree of validation required.

5. References

1. Heiser, W.H. and Pratt, D.T., Hypersonic Airbreathing Propulsion, AIAA Education Series, 1994.

2. Rubins, P.M. and Bauer, R.C., "Review of Shock-Induced Supersonic Combustion Research and Hypersonic Applications", J. of Propulsion and Power, Vol. 10, No. 5, Sept.-Oct. 1994.

3. Stull, F.D., "Scramjet Combustion Prospects", Astronautics and Aeronautics Journal, December 1995.

4. Rubins, P.M. and Bauer, R.C., "A Hypersonic Ramjet Analysis with Premixed Fuel Combustion", AIAA Paper 66-648.

5. Rubins, P.M and T.H.M., Cunningham, "Shock-Induced Combustion in a Controlled Area Duct", Paper AIAA 64-84.

6. Gonzalez, D.E., *"Computational Study of Inlet Injection for Pre-Mixed, Shock-Induced Combustion"*. Paper 96-4560, AIAA 7th International Spaceplanes and Hypersonic Systems and Technologies Conference, Norfolk, VA, November 18-22, 1996.

7. Gonzalez, D.E., *"Simulation of the Fluid Dynamic Behaviour of Transverse Hydrogen Injection in a Hypersonic Airstream"* . Paper AIAA 97-2224 presented at the 15th Applied Aerodynamics Conference, Atlanta, GA, June 23-25, 1997.

8. White, J., Private Communication, 1996.

9. Menter, F.R., "*Improved Two-Equation k-w Turbulence Model for Aerodynamic Flows*", NASA TM-103975, 1992.

10. Menter, F.R., "*Assessment of Higher Order Turbulence Models for Complex Two- and Three-Dimensional Flowfields*". NASA TM 103944, 1992.

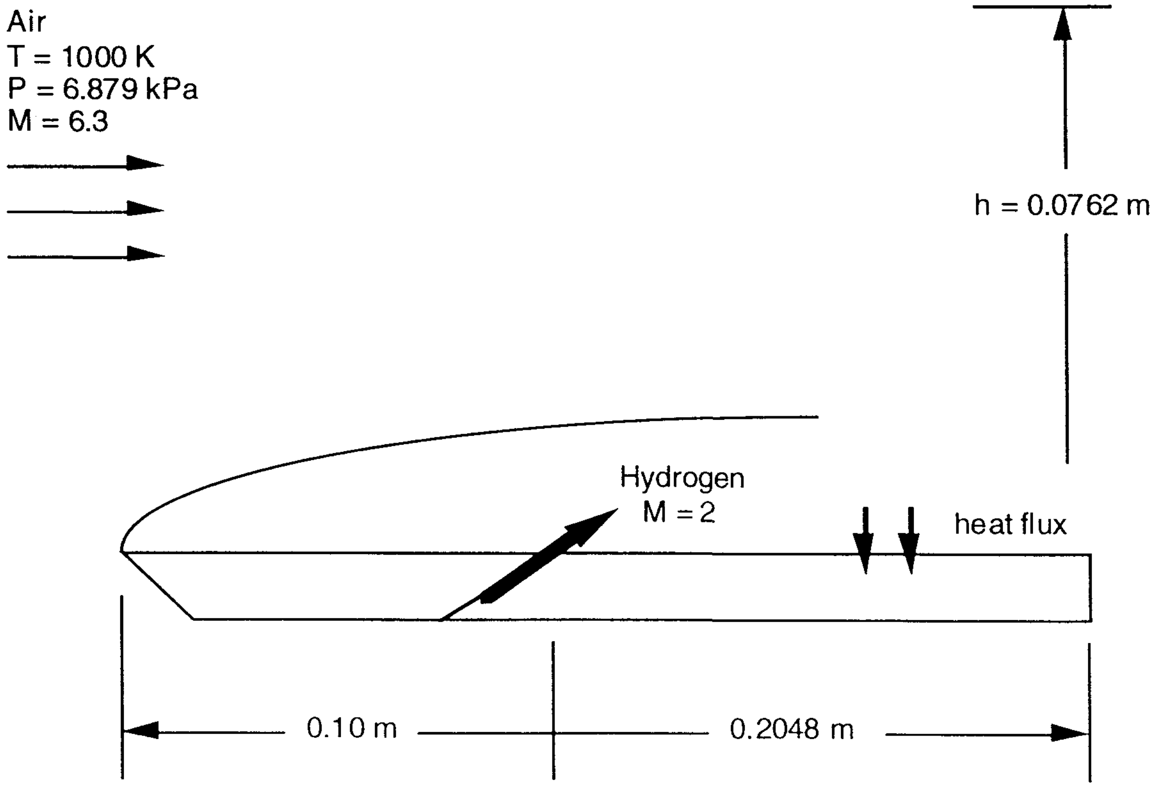

Fig. 1. Schematic of Hypersonic Vehicle Surface with Hydrogen Injection

Table 1
Flow Conditions

Parameter	Air	Hydrogen
Mach Number	6.3	2.0
Static Temperature (K)	1000.0	161.11 - 444.44
Static Pressure (kPa)	6.879	352.70

Boundary Conditions:

Wall Temperature	Isothermal with $T_w = 290$ K, and Adiabatic
Fuel Injection Angle	0°, 4°, 15°, 20°, 30°
Slot Size (cm)	0.0457, 0.1488

Table 2
Effect of Fuel Injection on Heat Transfer Rates

Case	m	ϕ_{H2}	d (cm)	Jet Angle	Q_u	Q_d (MW/m^2)	Q_t	Total Percent Reduction
	TURBULENT FLOW				NO INJECTION		0.3663	- - -
1a	2.73	0.559	0.0457	20	0.1424	0.0311	0.1735	52.63
3a	6.60	1.351	0.0457	4	0.1087	0.0562	0.1649	54.98
3d	6.60	4.396	0.1487	0	0.1087	0.0499	0.1585	56.73
3c	6.60	4.396	0.1487	20	0.1080	0.0633	0.1713	53.24
3f	6.60	4.396	0.1487	30	0.1094	0.0701	0.1796	50.93
	LAMINAR FLOW				NO INJECTION		0.2882	- - -
1c	2.73	0.559	0.0457	20	0.1445	0.0021	0.1463	49.24
3a	6.60	1.351	0.0457	4	0.1447	0.0363	0.18099	37.20
3b	6.60	1.351	0.0457	20	0.1368	0.0379	0.1748	39.35

subscripts: t = total, u = upstream from injector, d = downstream from injector

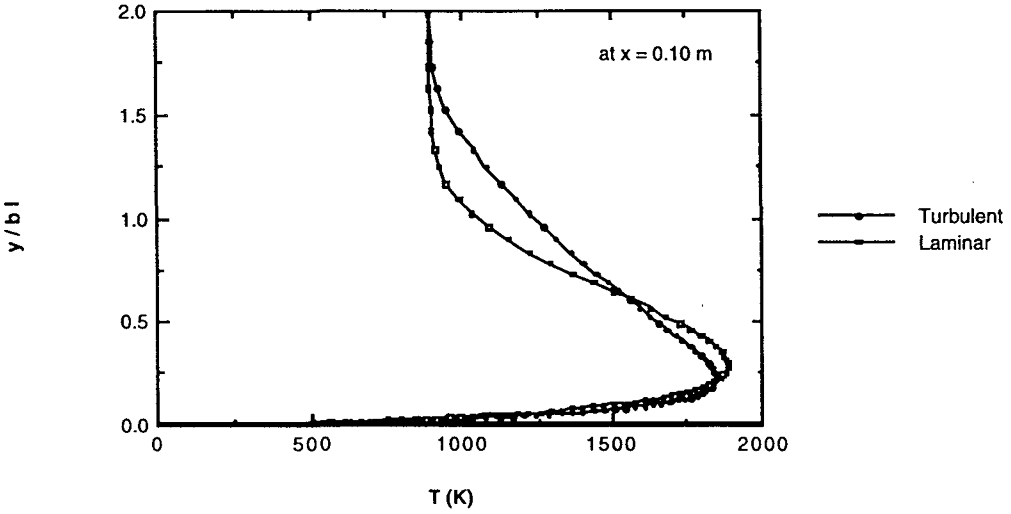

Figure 2. Undisturbed Boundary Layer Temperature Profiles

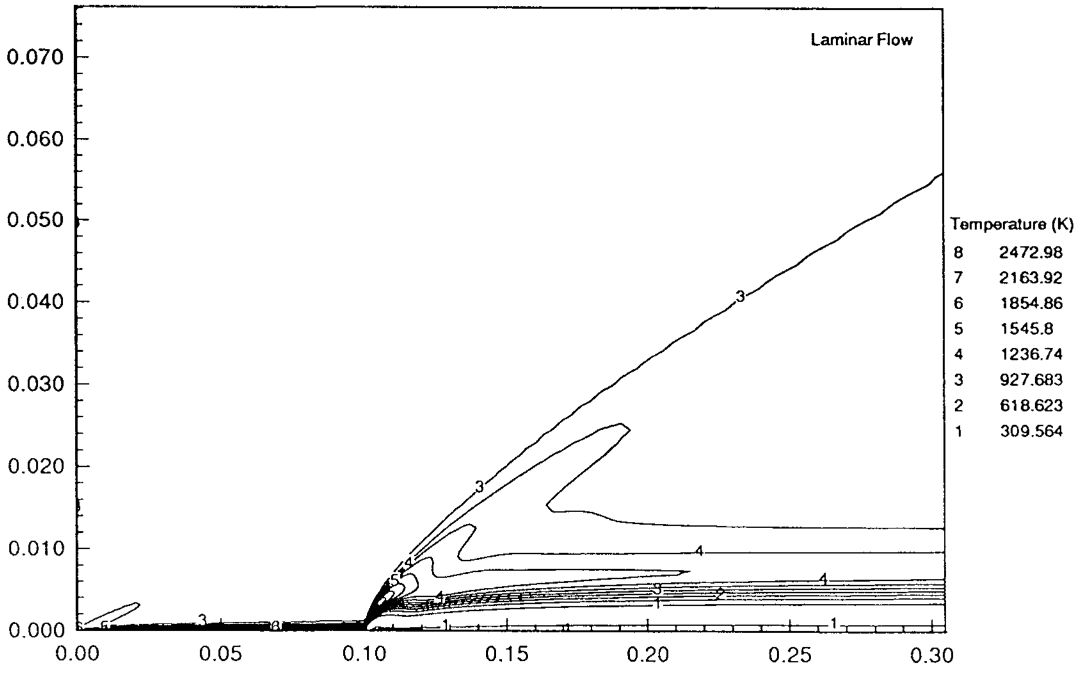

Figure 3. Temperature Field for 20° Fuel Injection - Laminar Flow

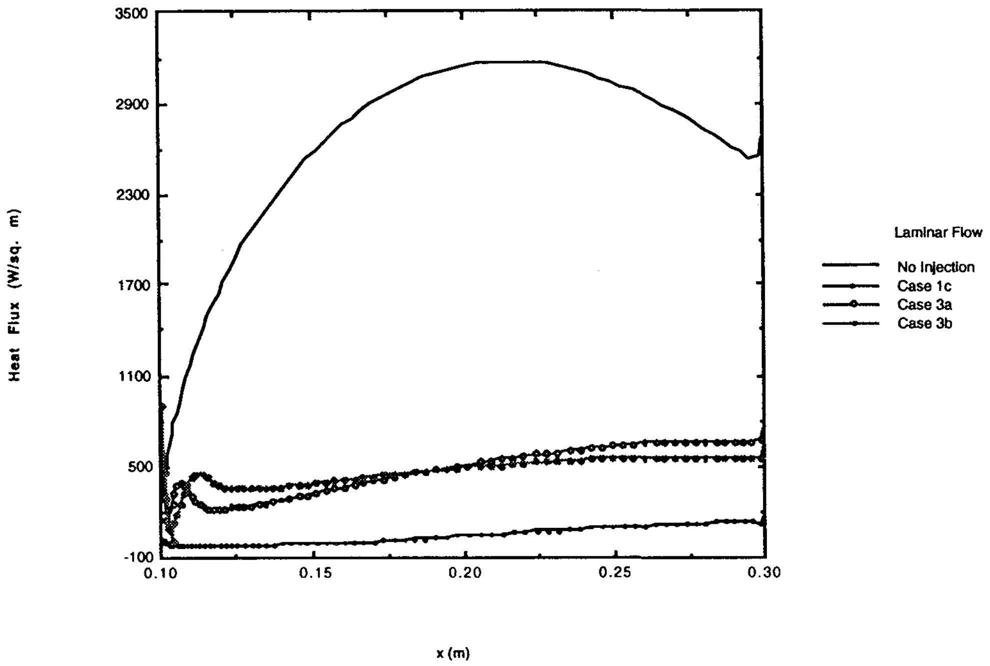

Figure 4. Heat Flux Distribution - Laminar Flow

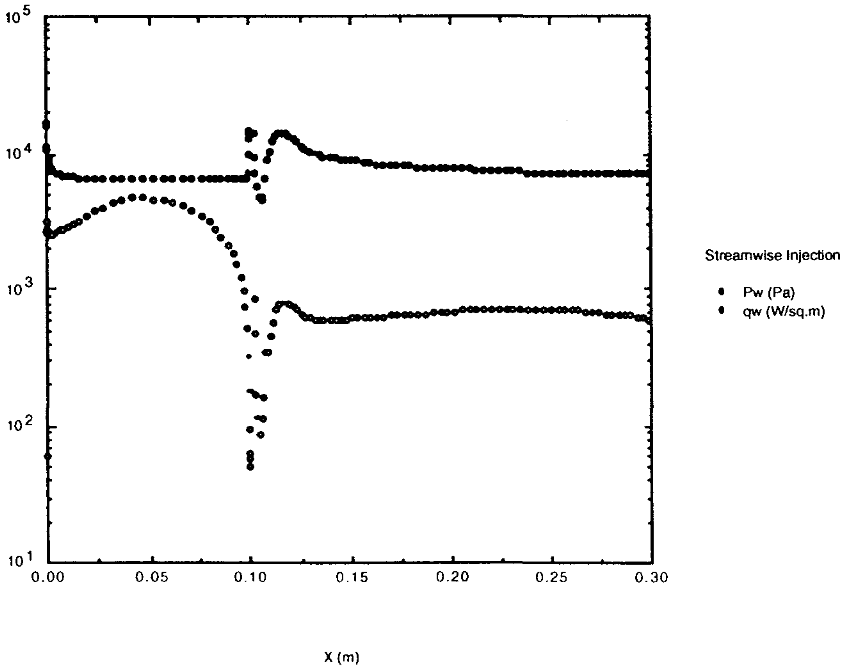

Figure 5. Wall Pressure and Heat Flux Distribution. 0° Injection

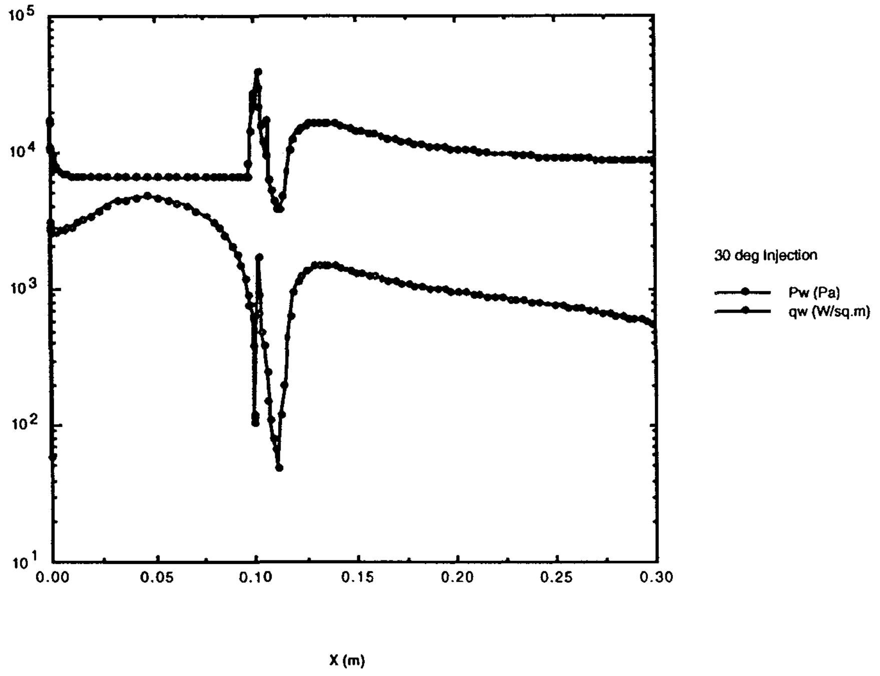

Figure 6. Wall Pressure and Heat Flux Distribution. 30° Injection

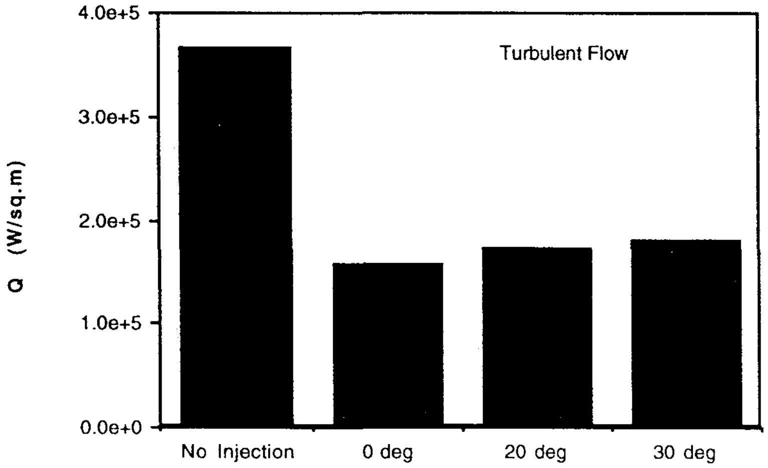

Figure 7. Effect of Injection Angle on Total Heat Load

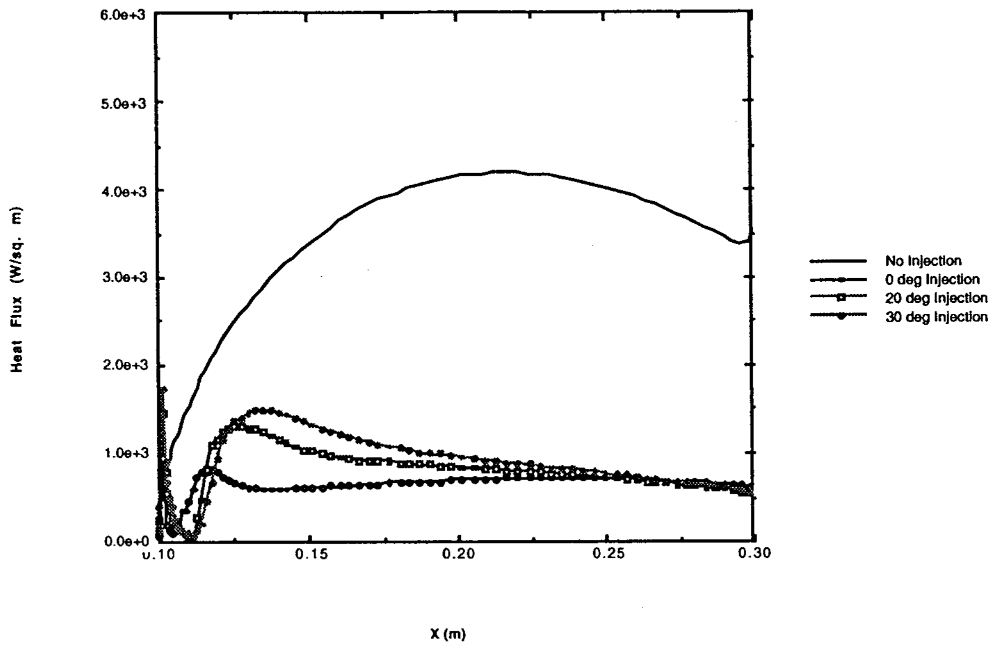

Figure 8. Effect of Injection Angle on Heat Flux Distribution - Turbulent Flow

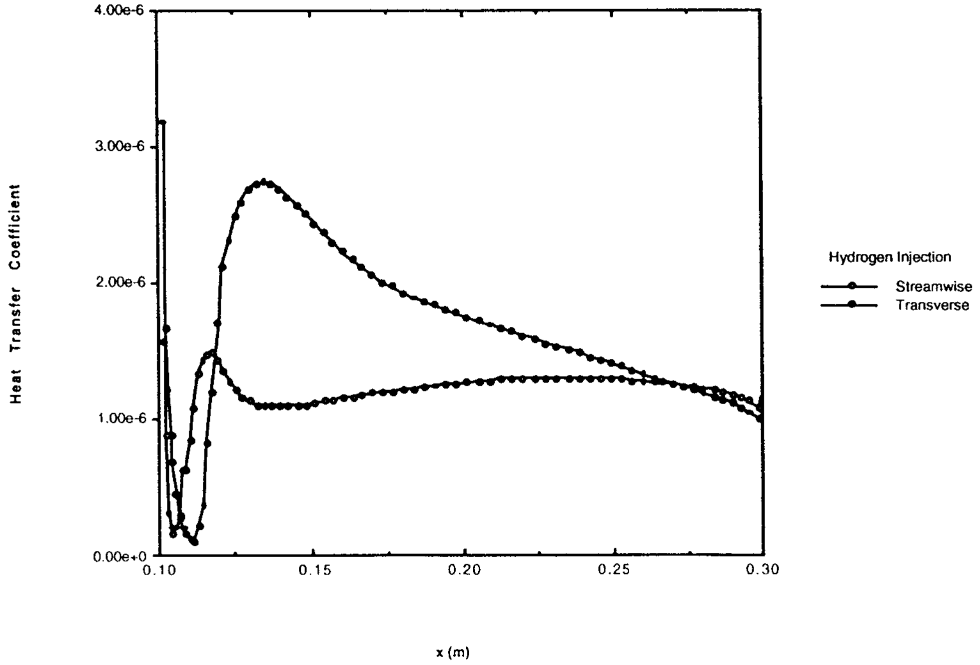

Figure 9. Effect of Hydrogen Injection on Wall Heat Transfer Coefficient