GRANT
IN-20-CR
142865
P- 12

NAG8-212

FNAS/Summer Faculty Fellowship Research Continuation Program

Task #3 Entitled

Analysis of Film Cooling in Rocket Nozzle

SEMI - ANNUAL REPORT

January 17, 1992 - October 17, 1992

Submitted to

National Aeronautics and Space Administration
George C. Marshall Space Flight Center
Marshall Space Flight Center, AL 35812

Prepared by

Dr. Keith Woodbury
The University of Alabama

Submitted by

Dr. Gerald R. Karr
Principal Investigator
The University of Alabama in Huntsville
Huntsville, Alabama 35899

November 20, 1992

Analysis of Film Cooling in Rocket Nozzles : Progress Report

Keith A. Woodbury
Department of Mechanical Engineering
The University of Alabama
Tuscaloosa, Alabama 35487

October 16, 1992

Abstract

This report summarizes the progress to date on the NASA contract #NAG8-212, Task No. 3. The overall project consists of three tasks, and Task 1 and Task 2 are now complete.

Task 1 involved the modification of the wall functions in the code FDNS to use a Reynolds Analogy-based method. This task was completed in August, 1992.

Task 2 involved the verification of the code against experimentally available data. The data chosen for comparision was from an experiment involving the injection of helium from a wall jet. Results obtained in completing this task also show the sensitivity of the FDNS code to unknown conditions at the injection slot. This task was completed in September, 1992.

Background. Analysis of film cooling in rocket nozzles by computational fluid dynamics (CFD) computer codes is desirable for two reasons. First, it allows prediction of resulting flow fields within the rocket nozzle, in particular the interaction of the coolant boundary layer with the main flow. This facilitates evaluation of potential cooling configurations with regard to total thrust, etc., before construction and testing of any prototype. Secondly, CFD simulation of film cooling allows for assessment of the effectiveness of the proposed cooling in limiting nozzle wall temperature rises. This latter objective is the focus of the current work.

A NASA code is available for the analysis of CFD processes. The FDNS (Finite Difference Navier Stokes) code was commissioned by MSFC and was authored by SECA, Inc. in 1990. Briefly, the FDNS code uses a central differencing scheme, coupled with artificial damping to capture shock waves, to solve for the heat, mass, and momentum conservation within an arbitrary geometrical domain. The code uses either a "standard" or "extended" k-ϵ turbulence model with an implementation of Launder and Spalding-like [1]wall functions for modelling of solid wall boundaries. Furthermore, the code allows for either equilibrium or finite-rate chemical reactions.

A major re-write of the code was performed over 1991-92 by Dr. Y. S. Chen, now of Engineering Sciences Incorporated (ESI). The resulting code is streamlined, has 3-D capability, but is limited to finite-rate chemical reactions. This code also has three turbulence models: standard k-ϵ, "extended" k-ϵ, and a low Reynolds number k-ϵ.

During the summer of 1991, Keith Woodbury of The University of Alabama performed computations using the NASA code FDNS for high-speed flow of air over an isothermal flat plate . The focus of his analysis was on the computed heat flux from the wall. The results showed that the FDNS code predicted heat fluxes about an order of magnitude lower than those measured under similar conditions in a shock tunnel . The explanation for the discrepancy is two-fold. First, the k-ϵ turbulence model used in FDNS does not account for the retarded velocity of the fluid in the near-wall region. Secondly, the particular form of the wall function used as a boundary condition for the energy equation does not adequately account for the effect of viscous heating in the near-wall region.

Project Plan. The desired objective is to use the FDNS code to predict wall heat fluxes or wall temperatures in rocket nozzles. As prior work [2] has revealed that the FDNS code is deficient in the thermal modeling of boundary conditions, the first step is to correct these deficiencies in the FDNS code. Next, these changes must be tested against available data. Finally, the code will be used to model film cooling of a particular rocket nozzle. Table 1 summarizes the tasks to be completed under this project.

The modifications to the FDNS code will be in the handling of the thermal boundary condition at the solid wall. The goal is to introduce as few changes as possible into the FDNS code, but enough to bring predictions from FDNS in line with available data. Previous work [2] demonstated that a simplistic Reynolds' Analogy brought the FDNS code predictions for wall heat flux into reasonable

Task One.	Modify the boundary wall functions in the FDNS code to include either an implementation of either a Reynolds Analogy-based method or the Jones-Whitelaw wall function. This task addresses the code's deficiency in modeling the viscous heating near the wall.
Task Two.	Calibrate the FDNS code against published experimental data. Specifically, the code will be used to compute the helium film cooling from a wall jet.
Task Three.	Use the modified code to compute the flow of hot gases through a nozzle. For this case, the nozzle geometry currently planned for the 40K subscale nozzle test is to be used. The gas composition will be frozen, i.e., non-reacting, and the film coolant used will be ambient hydrogen.

Table 1: Tasks to be completed under project

agreement with data for the case of flow over an isothermal plate. Such a modification will be introduced in the wall functions in the FDNS code, and it will be determined if this alteration is adequate in Task 2. If not, an alternate form of the wall functions (due to Jones and Whitelaw) has been reported to yield good estimates for the wall jet problem [3] and this will be implemented and verified in Task 2.

Verification of the FDNS code modifications will be accomplished by comparing the code predictions to the experimental data of Holden [4]. The basis for comparison will be the predicted wall heat flux and the wall static pressure. Specifically, Holden's case number 45 will be considered. Case 45 is for supersonic injection of Helium coolant ($T_0 = 530\ R$, $M = 3$) parallel and into the flow of air at the nominal conditions $T_0 = 2200\ R$ and $M_\infty = 6.4$ via a wall jet.

The code will ultimately be used to compute the flow through a rocket nozzle, with supersonic film coolant injection. The geometry of the nozzle, gas composition, and coolant injection scheme to be used in the computation will be that of the 40K Subscale Nozzle. This information was disseminated at the CFD Consortium in Propulsion Technology meeting of August 1, 1991.

Project Progress. Task 1 of the project was completed in August. The current version of the code was obtained from Dr. Y. S. Chen of ESI on August 3, 1992. This version contained a heat flux wall function similar to the one recommended by Woodbury [2]. This function was modified to make it conform to the Reynolds-Analogy desired for this project.

The current formulation of the code, the wall function for the energy equation has a form

$$q_w = (h_w - h_p - \mathrm{Pr}_t(u_p - u_w)^2/2)(\tau_w/u_p) \tag{1}$$

where h_w and h_p are the enthalpies of the wall and the adjacent point away from the wall, respectively; u_w and u_p are the velocities, τ_w is the wall shear stress, and Pr_t is the turbulent Prandtl number, taken to be $\mathrm{Pr}_t = 0.90$.

Note that this wall function is similar to the Reynold's Analogy model proposed in Reference [2]. That function follows from the definition of the heat transfer coefficient, h_{conv} for a compressible boundary layer (Shapiro [5], page 1100)

$$q_w = h_{conv}(T_{aw} - T_w)$$

where T_{aw} is the *adiabatic wall temperature*, and T_w is the actual wall temperature. If the adiabatic wall temperature (given by Shapiro [5], page 1099) is

$$T_{aw} = T_\infty + RU_\infty^2/2/c_p$$

which defines the *recovery factor*, R. ($R \approx 0.89$ for air.) Then, with the *Reynolds Analogy* (as suggested by Shapiro ([5], page 1100), and verified experimentally by Holden ([6], Figure 12a), expressed as

$$\frac{C_f}{2} = \frac{\tau_w}{\rho U_\infty^2} \approx C_H = \frac{h_{conv}}{c_p \rho U_\infty}.$$

the heat transfer may be inferred based on the wall friction as

$$q_w = \frac{\tau_w c_p}{U_\infty}(T_\infty - T_w) + \frac{\tau_w}{2}U_\infty R.$$

Or,

$$q_w = \frac{\tau_w}{U_\infty}(h_\infty + R\frac{U_\infty^2}{2} - h_w) \tag{2}$$

where here h is the *enthalpy*, not the heat transfer coefficient. Comparing Equation 1 with Equation 2, and recognizing that Pr_t is numerically equal to R, it can be seen that the expressions are substantially the same.

The wall functions are implemented using a dimensionless distance y^+. This distance is defined in terms of the resulting shear stress at the wall as $y^+ =$

$y\sqrt{\tau_w/\rho}/\nu$. The wall functions implemented in this version are claimed to be accurate over a range of $60 < y^+ < 700$.

Task 2 was completed in September, 1992. This task involved using the FDNS code to predict the heat flux from a $M = 3$ Helium wall jet. The actual case is documented in the experimental work of Holden [4].

In Holden's report, specific information about the actual profile conditions (velocity and temperature) at the jet injection point were not available. This led to a parametric study in the present investigation to determine the effects of various assumptions about these conditions.

This effort is made to study the effects of inlet boundary conditions of the injection on the wall heat transfer downstream of the injection slot. Results that follow are all for test condition "Run 45" one of the test cases in Holden's report[4]. Computations are carried out for a grid containing 121 by 41 mesh points. Grid spacing has been adjusted to ensure convergent solutions and desired dimensionless normal distance y^+ within the range of $60 < y^+ < 700$, as is suggested by the author of the code, Dr. Y. S. Chen.

In all cases, turbulence quantities k and ϵ are assumed to be uniform at the exit of the injection slot, and are given by

$$k = 0.001 U_{ref}^2 = \text{Constant}$$

$$\epsilon = \frac{C_\mu (k)^{3/2}}{0.03 X_{ref}} = \text{Constant}$$

Fig. 1 shows the effects of the inlet temperature profile on the heat transfer downstream of the slot. In the figure, Holden's data are compared to computed results from FDNS for both a constant inlet temperature and a turbulent inlet temperature profile. In the computed results, the velocity profile at the inlet was taken as uniform. The turbulent inlet temperature profile was obtained from a contour map of computed results for analysis of the injection nozzle alone. These injection nozzle compuatations were performed by Dr. Y. S. Chen[7]. This profile was approximated by curve fit as

$$T(y) = 0.321 T_{ref} \left[\frac{y}{h/2}\right]^{-0.3831}$$

This figure shows that the effect of temperature profile on the predicted wall heat flux is limited to a distance of 2 inches (about 30 – 35 times the slot height) from the slot. In this region, Chen's profile predicts a higher heat flux than the experimental result.

Fig. 2 incorporates Chen's results for temperature and velocity at the injection nozzle. The result, denoted 8.28 in the figure, underpredicts the heat flux over most of the flow region.

Fig. 3 show the effect of the laminar versus turbulent velocity profiles on the downstream wall heat flux. For these calculations, the inlet temperature profile was assumed uniform. In the figure, the results corresponding to the turbulent velocity profile are denoted as 8.18, and those for the laminar assumption as 8.25. The turbulent profile again was assumed as the 1/7 power law, and a simple parabolic assumption was made for the laminar profile:

$$U(y) = \frac{3}{2}(4967.77)\left[\frac{y}{h/2}\right]^2$$

The laminar profile results in a very strong decrease, then an increase, in heat flux over a short distance. This confirms that the assumption of a laminar velocity profile at the slot inlet is clearly unreasonable.

Fig. 4 shows the effect of varying the inlet velocity profile. In this figure, both computations use Chen's temperature profile, but one (denoted 8.16) uses uniform velocity profile, while the other (denoted 8.18) uses an approximate turbulent profile (the 1/7 power law):

$$U(y) = 4967.77\left[\frac{y}{h/2}\right]^{1/7}$$

It can be seen from this figure that the turbulent velocity profile does not result in a better prediction than the uniform one.

Summary. The project is proceeding according to schedule. Tasks 1 and 2 are complete, and Task 3 is in process. The following observations can be made regarding the FDNS code predictions obtained to date:

1. The Reynolds Analogy-based wall function gives reasonable, but not accurate, estimates of the wall heat flux downstream of a wall jet.
2. The predictions obtained depend on the velocity and temperature profiles of the flow at the injection. However, uniform profiles give as good agreement as any other assumption (turbulent, or laminar). Of course, actual inlet profiles will produce more accurate results.

3. The inlet velocity profile affects wall heat flux much more than the temperature profile does.

References

[1] B. E. Launder and D. B. Spalding, "Numerical computation of turbulent flows," *Computer Methods in Applied Mechanics and Engineering*, vol. 3, pp. 269–289, 1974.

[2] K. A. Woodbury, "Assessment of nasa code fDNS2D for computation of film cooling effectiveness," Tech. Rep. ZZ, NASA/ASEE SFFP, August 1991. prepared for NASA/MSFC, Contract NGT-01-008-021.

[3] C. P. Chen, "Benchmark problem: status report," August 1 1991. Presented at CFD Consortium Meeting, NASA/MSFC.

[4] M. S. Holden, "A data base of experimental studies of shock wave/wall jet interaction in hypersonic flow," Tech. Rep., Calspan-UB Research Center, Buffalo, NY, April 1990.

[5] A. H. Shapiro, *The Dynamics and Thermodynamics of Compressible Flow*. Brown, 1954.

[6] M. S. Holden, "Shock wave-turbulent boundary layer interaction in hypersonic flow," Tech. Rep. 77-45, AIAA, January 24-26 1977. presented at AIAA 15th Aerospace Sciences Meeting.

[7] Y. S. Chen, "FDNS user's short course," July 1992. held at ESI, Huntsville, AL.

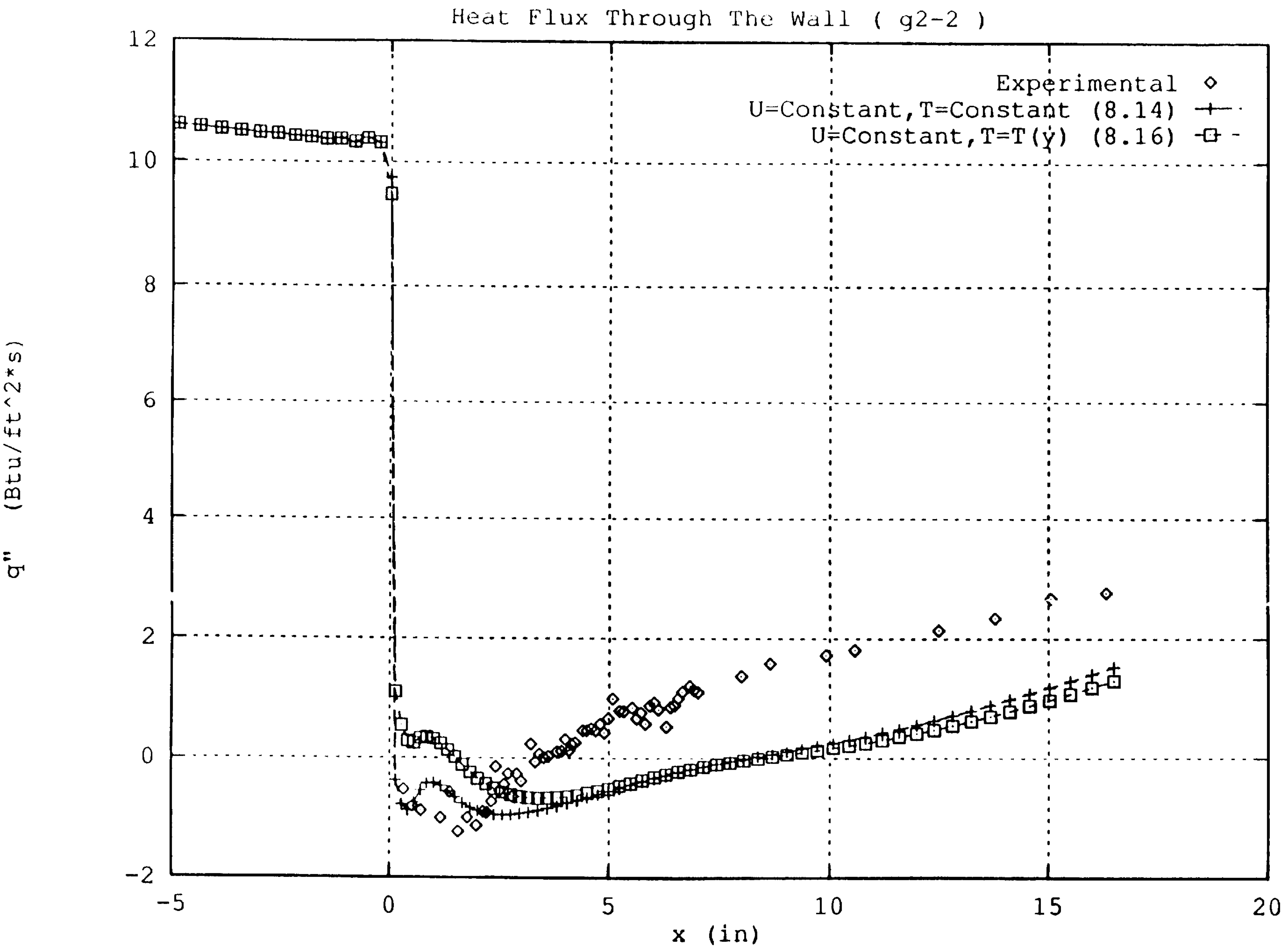

Figure 1: Effect Turbulent Inlet Temperature Velocity Profile

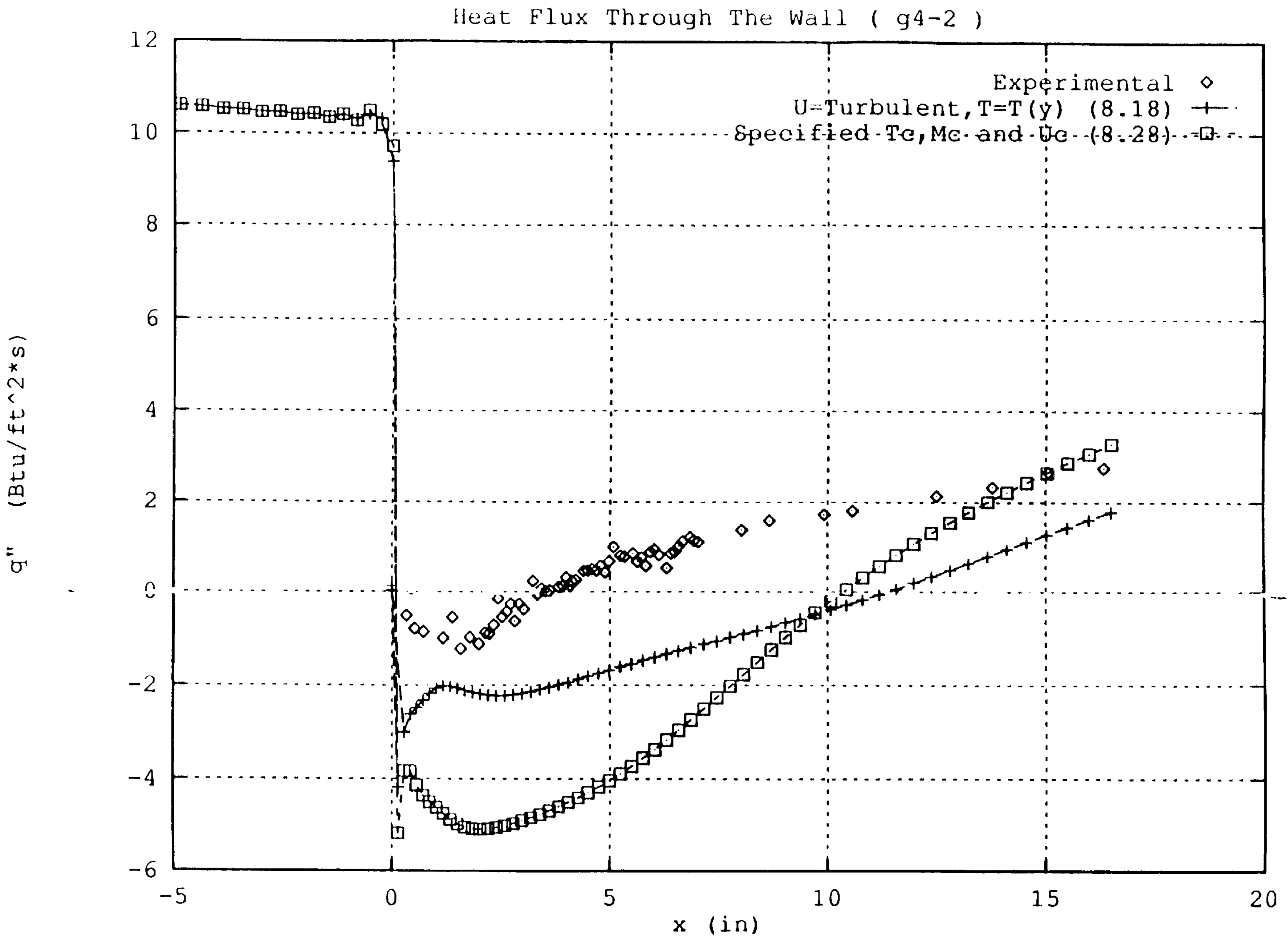

Figure 2: Effect of Computed Inlet Temperature and Velocity Profiles

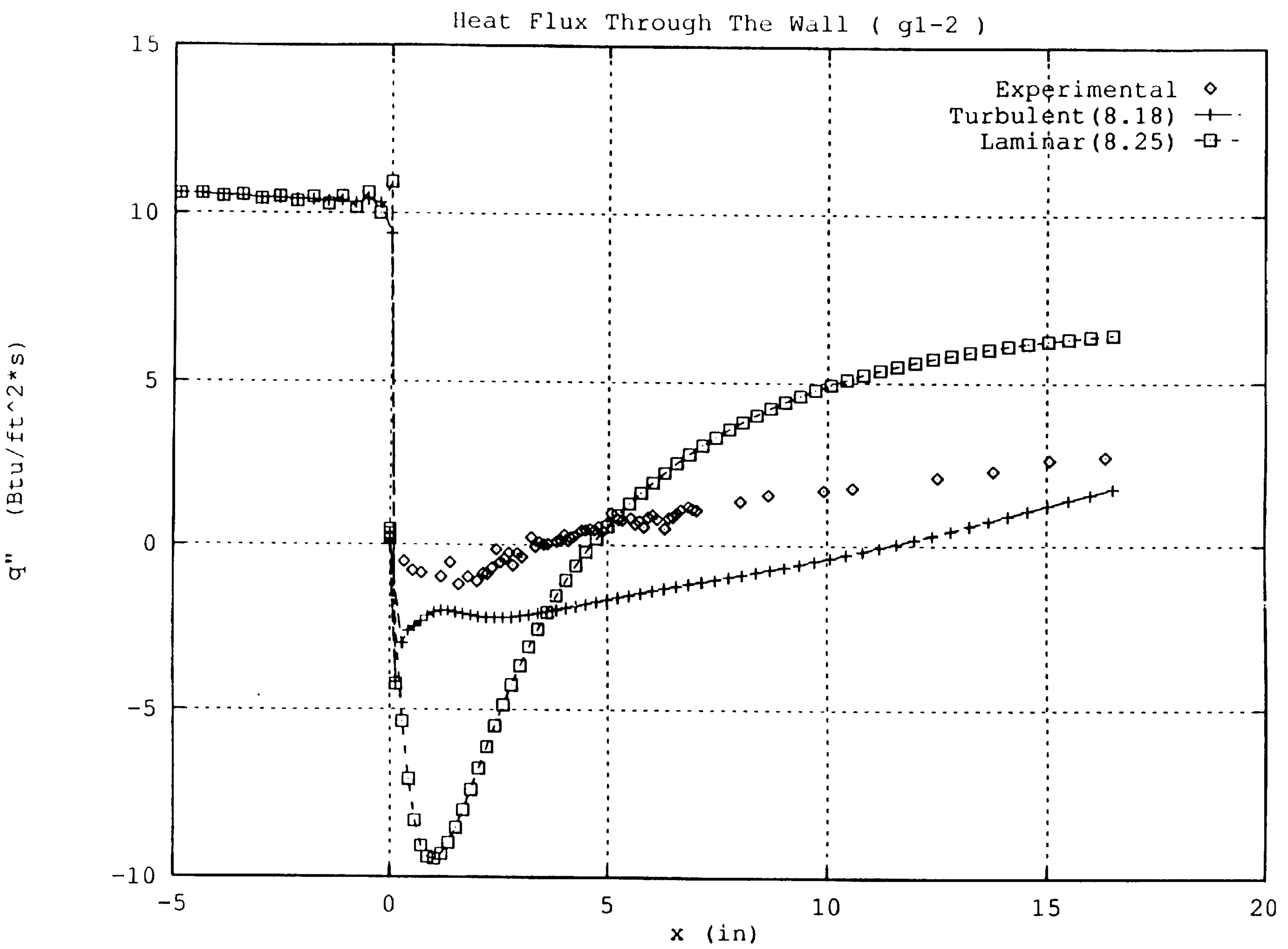

Figure 3: Laminar versus Turbulent Velocity Profile Effects

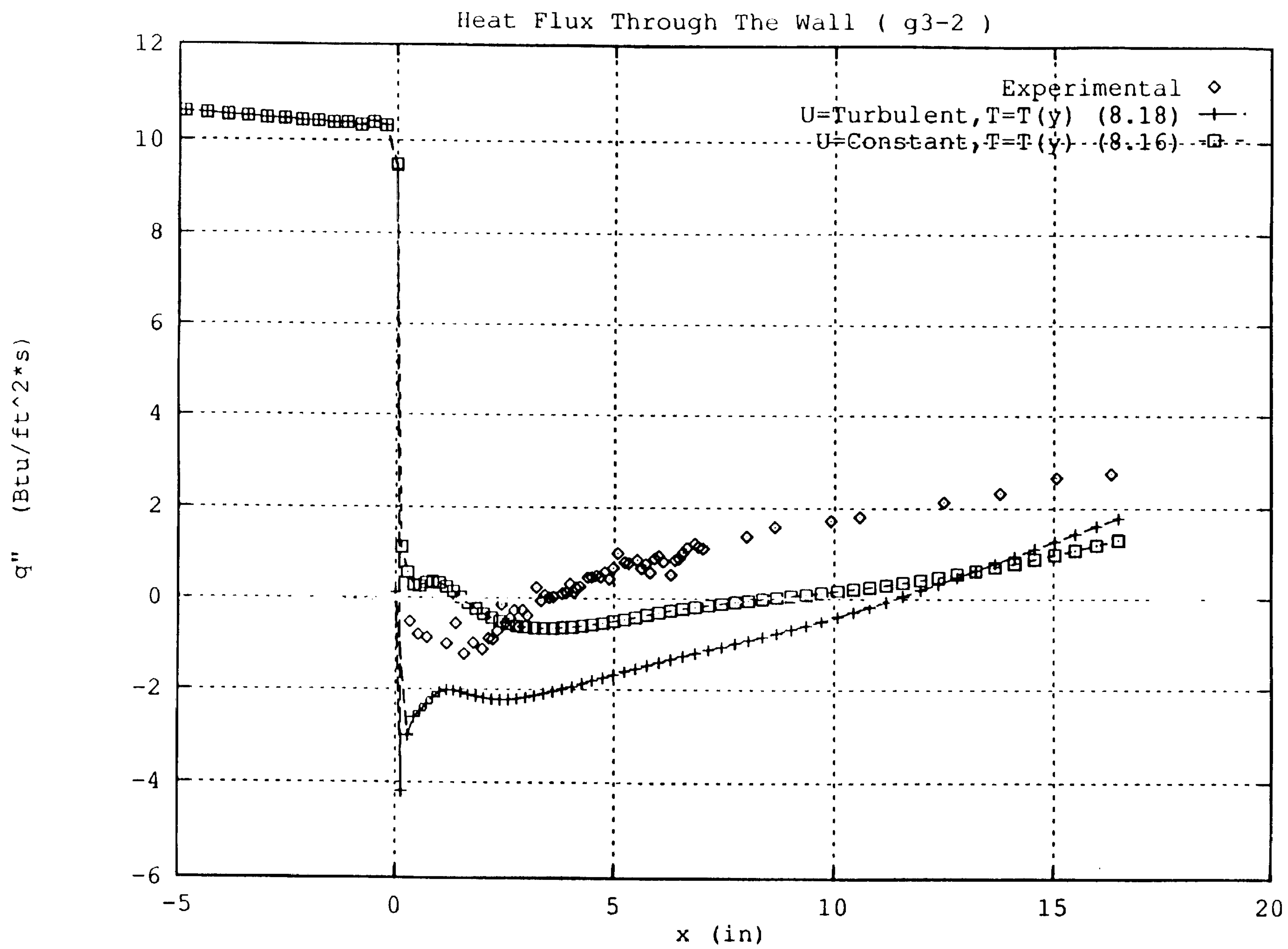

Figure 4: Effect of Tubulent Inlet Velocity Profile