N92-15902

1991

NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA IN HUNTSVILLE

Assessment of the NASA Code FDNS2D for Computation of Film Cooling Effectiveness

Prepared By:	Keith A. Woodbury, Ph.D., P.E.
Academic Rank:	Assistant Professor
Institution:	The University of Alabama Mechanical Engineering Deparment
NASA/MSFC:	
Office:	Combustion Devices Group
Division:	Component Development Division
Branch:	Turbomachinery and Combustion Devices
MSFC Colleague:	Dave Sparks
Contract No.:	NGT-01-008-021 University of Alabama in Hunsville

Introduction. The role of Computational Fluid Dynamics (CFD) programs is usually one of analysis. Generally they are not used in the design phase of a project. There has been a concerted effort at MSFC to integrate CFD codes into the design phase of Combustion Devices, specifically, in the design of the STME nozzle. Before the results of such analyses can be accepted, the credibility of the CFD codes upon which they are based must be established.

This report details the effort to assess the capability of the NASA code FDNS2D to compute the heat transfer to a solid bounding surface. Specifically, high-speed flow over a flat plate is considered, and the resulting wall shear stress, and heat transfer are computed. These values are compared against analytical results (for wall shear stress) and experimental data (for heat transfer).

What follows in this report is a brief description of the FDNS2D code, with special emphasis on how it handles solid wall boundary conditions. The flow conditions and the FDNS solution are presented next, along with comparison to analytical and experimental data. Some intermediate observations are then made, followed by a recommendation for adoption of an alternate method for computing the wall heat flux. Some conclusions are made to close out the report.

FDNS. The computer code name, FDNS, stands for *F*inite *D*ifference *N*avier–*S*tokes. The code, written by SECA, Inc. in 1988 [1], is a pressure–based finite-difference solver. The code implements artificial viscosity in order to capture shocks in high-speed flows.

The version used in this effort is two-dimensional, hence the name FDNS2D. It solves the continuity, u-, v- momentum, energy, k-ε, and specie conservation equations. The k-ε turbulence models available in the code are both the "standard" and "extended" versions. Additionally, chemistry capability is provided by either equilibrium or finite-rate chemical reactions.

The implementation of solid wall boundary conditions in FDNS is by use of wall functions [2]. The particular implementation in the code is assumed valid whenever the dimensionless distance $y^+ = \rho y_p C_\mu^{1/4} k^{1/2}/\mu$ is greater than 11.63. If y^+ is less than this value, then a laminar expression is used. It is believed that this "patching" of the laminar expression for near wall grid points is highly inaccurate. Thus, to expect reasonable solutions from the FDNS2D code, wall functions must be employed, and this means that the computational mesh must be chosen so that the y^+ parameter is greater than 11.63.

Test Case. The test case used in this investigation is a simple flow over a flat plate. The fluid flowing is air, which approaches the plate with a Mach number $M = 6.42$ and a static temperature of $T = 258$ R. The specific case being studied is "Run 4" from a set of data collected at Calspan and published by Michael Holden [3]. To model his wind tunnel condition, the plate was treated as isothermal at a temperature of 540 R.

The computer mesh was generated using the GENIE3D program on the IRIS workstations in the CFD branch (ED32) at MSFC. The mesh was coarse, with 121 equally spaced nodes in the lentghwise direction (x-direction) covering a distance of 3.75 feet, and 41 nodes in the cross-stream direction (y-direction) over a range of 0.5 feet. The mesh in the y-direction was graded using a hyperbolic tangent stretching scheme, with the node closest to the wall at a distance of 3.75E-5 feet. After solution of the problem, it was found that this y_p distance resulted in y^+ values in the range $23 < y^+ < 31$. This ensures that wall functions were utilized by the FDNS code.

The FDNS code produces an output file (the restart file, FORTRAN unit 9) which contains a table of all the solution variables at the nodal points. From these tabular values, and by making use of the wall functions as implemented in the program, the values of the wall shear stress, τ_w, and the wall heat flux, q_w, can be determined.

To judge the quality of these computations, comparison is made against the analytical solution of van Driest (from Shapiro [4]) for the wall shear stress, and against the experimental data of Holden [3] for wall heat flux. A plot of the friction coefficient $C_f = \tau_w/\frac{1}{2}\rho U_\infty^2$ versus distance along the wall is shown in Figure 1. As can be seen from that figure, the agreement between the FDNS predictions and the van Driest solutions is good. A plot of the wall heat transfer q_w versus distance along the wall is shown in Figure 2. The open circles, denoting the heat transfer predictions from the wall functions as implemented in the FDNS code, are seen to fall well below the experimental data of Holden.

Observations. Although the wall functions do an excellent job of modelling the wall shear stress for flow over a flat plate with high Mach numbers, the corresponding computations for wall heat flux are grossly in error. Therefore, the heat fluxes utilized in the FDNS code for boundary conditions are inaccurate. What this means, of course, is that the resulting temperature profiles in the fluid must be in error.

Is there hope? Given the poor result of the heat flux predictions from the FDNS code, and the fact that it is exactly these values that are needed in order to ultimately assess the effectiveness of film cooling in rocket nozzles, an alternative approach to computing the wall heat flux is desired. One method might be to scour the literature and find another representation for the wall function for the energy equation (there are many), but a simpler approach is suggested here. The method to be used is based on a Reynolds analogy.

For a compressible boundary layer (Shapiro [4], page 1100)

$$q_w = h(T_{aw} - T_w) \tag{1}$$

where T_{aw} is the *adiabatic wall temperature*, and T_w is the actual wall temperature. The adiabatic wall temperature is given by (Shapiro [4], page 1099)

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

which defines the *recovery factor*, R. ($R \approx 0.89$ for air.) The Reynolds Analogy, as suggested by Shapiro ([4], page 1100), and verified experimentally by Holden ([5],

Figure 12a), may be expressed as

$$\frac{C_f}{2} = \frac{\tau_w}{\rho U_\infty^2} \approx C_H = \frac{h}{c_p \rho U_\infty}. \tag{3}$$

By combining these relations, the heat transfer may be inferred based on the wall friction as

$$q_w = \frac{\tau_w c_p}{U_\infty}(T_\infty - T_w) + \frac{\tau_w}{2} U_\infty R. \tag{4}$$

This equation was used to reprocess the results from the previous execution of the FDNS code. The resulting values are plotted in Figure 2 as the open squares. As can be seen from the figure, the agreement with the experimental data from Holden is vastly improved over the strict application of the wall function.

Conclusions. The following conclusions can be drawn from this investigation:

- FDNS2D using wall functions does a good job of predicting τ_w for high speed boundary layer flows.
- FDNS2D using wall functions does a poor job of predicting q_w for high speed boundary layer flows.
- The Reynolds Analogy may be employed to obtain reasonable estimates of the heat fluxes based on the FDNS2D output.
- If FDNS2D is expected to give reasonable values for the temperature field in the fluid, modification of the existing wall function boundary condition will be necessary.

References

[1] Y. Chen, R. Farmer, and J. A. Freeman, "The use of variational principles in improving computational fluid dynamics methodology," Tech. Rep. SECA-TR-90-05, SECA, May 1990. prepared for NASA/MSFC, Contract NAS8-37408.

[2] 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.

[3] 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.

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

[5] 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.

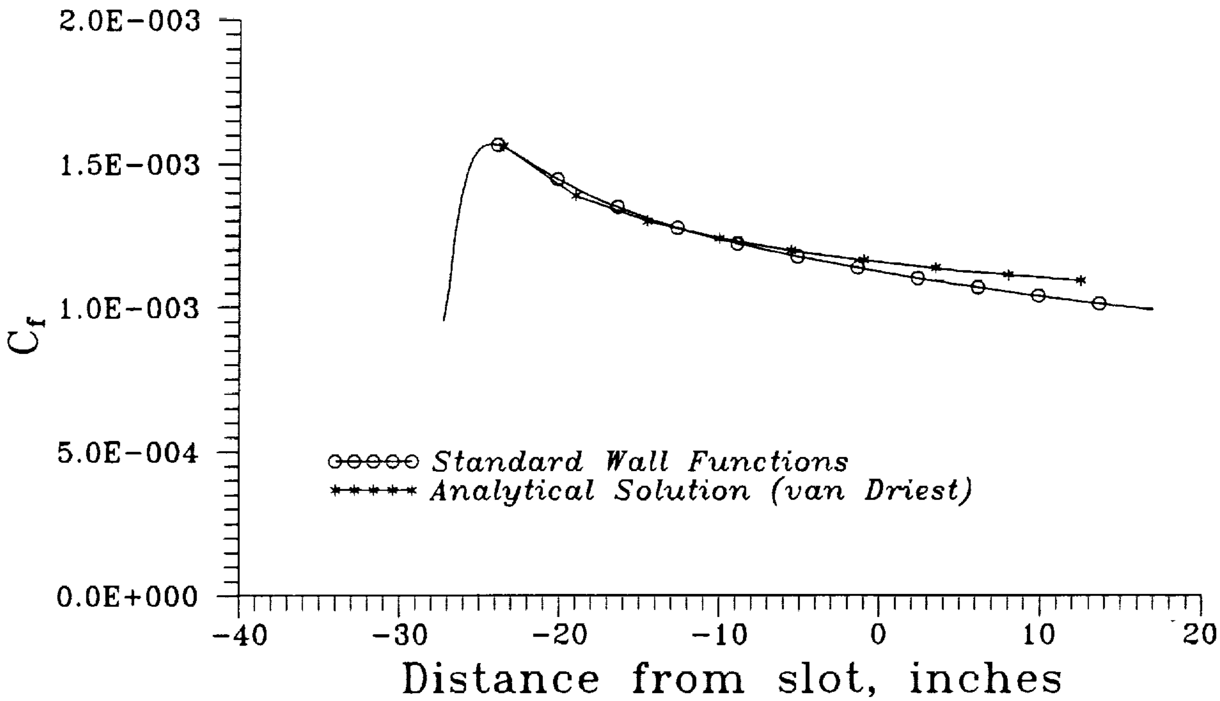

Figure 1: Surface Shear Stress Comparison

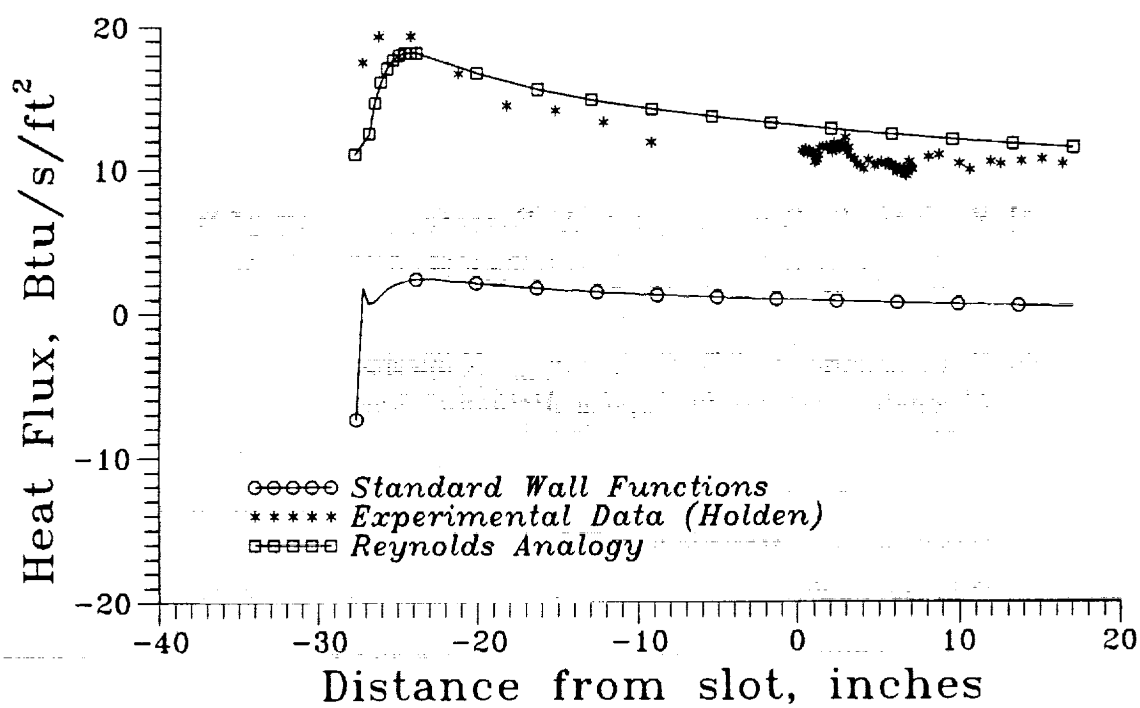

Figure 2: Surface Heat Flux Comparison

☆ U.S. GOVERNMENT PRINTING OFFICE 1991 — 631–060/40173