

N99-17889

1992

NASA/ASEE SUMMER FACULTY FELLOWSHIP PROGRAM

**MARSHALL SPACE FLIGHT CENTER
THE UNIVERSITY OF ALABAMA**

**ANALYSIS OF FILM COOLING
IN ROCKET NOZZLES**

Prepared By: Keith A. Woodbury, Ph.D., P.E.
Academic Rank: Assistant Professor
Institution and
Department: The University of Alabama
Mechanical Engineering Department
NASA/MSFC:
Office: Combustion Devices Group
Division: Component Development Division
Branch: Turbomachinery and
Combustion Devices Branch
MSFC Colleague: Dave Sparks

Introduction. Computational Fluid Dynamics (CFD) programs are customarily used to compute details of a flow field, such as velocity fields or species concentrations. Generally they are not used to determine the resulting conditions at a solid boundary such as wall shear stress or heat flux. However, determination of this information should be within the capability of a CFD code, as the code supposedly contains appropriate models for these wall conditions. Before such predictions from CFD analyses can be accepted, the credibility of the CFD codes upon which they are based must be established.

This report details the progress made in constructing a CFD model to predict the heat transfer to the wall in a film cooled rocket nozzle. Specifically, the objective of this work is to use the NASA code FDNS to predict the heat transfer which will occur during the upcoming hot-firing of the Pratt & Whitney 40K subscale nozzle (1Q93). Toward this end, an $M = 3$ wall jet is considered, and the resulting heat transfer to the wall is computed. These values are compared against experimental data available in Reference [1]. Also, FDNS's ability to compute heat flux in a reacting flow will be determined by comparing the code's predictions against calorimeter data from the hot firing of a 40K combustor. The process of modeling the flow of combusting gases through the Pratt & Whitney 40K subscale combustor and nozzle is outlined.

What follows in this report is a brief description of the FDNS code, with special emphasis on how it handles solid wall boundary conditions. The test cases and some FDNS solution are presented next, along with comparison to experimental data. The process of modeling the flow through a chamber and a nozzle using the FDNS code will also be outlined.

FDNS. The computer code name, FDNS, stands for **F**inite **D**ifference **N**avier-**S**tokes. The code, written by SECA, Inc. in 1988 [2], was completely rewritten by ESI, Inc. in 1992. It is a pressure-based finite-difference solver. The code implements artificial viscosity in order to capture shocks in high-speed flows. It solves the continuity, u -, v -, w -momentum, energy, k - ϵ , and specie conservation equations. The k - ϵ turbulence models available in the code are both the "standard" and "extended" versions. Chemistry capability is provided by finite-rate chemical reactions.

Implementation of solid wall boundary conditions in FDNS is by use of *wall functions*. Wall functions are analytically motivated and empirically formulated relationships which are designed to enforce the no-slip and no temperature jump boundary conditions in a turbulent flow. A previous investigation [3] revealed that the earlier version of the code, FDNS2D, employed wall functions which grossly underestimated the heat flux to the wall in compressible flows. However, in the current formulation of the code, the wall function for the energy equation has a form

$$q_w = (h_w - h_p - Pr_t(u_p - u_w)^2/2)(\tau_w/u_p) \quad (1)$$

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 $Pr_t = 0.90$.

Note that this wall function is similar to the Reynold's Analogy model proposed in Reference [3]. That function follows from the definition of the heat transfer coefficient for a compressible boundary layer (Shapiro [4], page 1100)

$$q_w = h(T_{aw} - T_w) \quad (2)$$

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

$$T_{aw} = T_\infty + RU_\infty^2/2/c_p \quad (3)$$

which defines the *recovery factor*, R . ($R \approx 0.89$ for air.) Then, with the *Reynolds Analogy* (as suggested by Shapiro ([4], page 1100), and verified experimentally by Holden ([5], Figure 12a), expressed as

$$\frac{C_f}{2} = \frac{\tau_w}{\rho U_\infty^2} \approx C_H = \frac{h}{c_p \rho U_\infty} \quad (4)$$

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

$$q_w = \frac{\tau_w c_p}{U_\infty} (T_\infty - T_w) + \frac{\tau_w}{2} U_\infty R. \quad (5)$$

Or,

$$q_w = \frac{\tau_w}{U_\infty} (h_\infty + R \frac{U_\infty^2}{2} - h_w) \quad (6)$$

where here h is the *enthalpy*, not the heat transfer coefficient. Comparing Equation 1 with Equation 6, 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 $60 < y^+ < 700$.

Test Cases. Two test cases are used in the present effort to gauge the usefulness of the FDNS code for predicting wall heat flux. The first is specific for an injected film, and the second is more appropriate for a reacting flow.

The test case for film injection is a flow of Mach 6.4 air over a bank of Mach 3.0 Helium wall jets. The helium is injected parallel to the wall. The specific case being studied is "Run 45" from a set of data collected at Calspan and published by Michael Holden [1]. To model his wind tunnel condition, the wall was treated as two isothermal plates; the entry region (up to the wall jets) was taken as 550 R, and the second section (after the wall jets) was taken as 535 R.

Computational meshes were generated using the GENIE++ program (developed by Mississippi State University) on RS/6000 workstations at the University of Alabama. The meshes are coarse, with 121 nodes in the lengthwise direction (x -direction) covering a distance of 4.333 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. Several meshes were used with differing distances to the first node away from the wall (y_p) which resulted in different values of y^+ .

The results from the executions with three different grids are shown in Figure 1 along with Holden's experimental data. The wide variation in results with different grids is evident. The serious disagreement between computed results and the experimental data led to discussions with MSFC personnel in ED32 and ultimately with the author of the code Y. S. Chen of ESI, Inc. This prompted Dr. Chen to examine the code, and he determined that *there was in fact a bug in the code*. The version used up to that point was taken from the Convex computer in ED32 known as tyrell.msfc.nasa.gov on June 6, 1992, from the directory /u/te/ychen/fdnsy. Dr. Chen issued a corrected version of the code on August 3, 1992 from the EADS system.

Preliminary results from the most recent version of the code are shown in Figure 2. Note that agreement over the larger portion of the entrance plate is generally good. However, questions remain about the leading edge results ($-30 < x < -28$) and in the slot region ($0 < x < 16$). It is believed that the discrepancy in the slot region is due to improper gridding ($y^+ < 60$) and this problem is still being worked.

The second test case, involving a combustor flow through a chamber, is based on experimental data gathered in September 1990 on a Pratt & Whitney 40K chamber [6]. Incidentally, the chamber used in that sequence of tests is the same as the one which will be used on the subscale nozzle firing 1Q93. The test case chosen is designated 027C, and had a O/F ratio of 6.00 and a chamber pressure of 1775 lb_f/in². The data gathered during that test is shown in Figure 3.

FDNS results for this test case are not yet available. However, the next section describes the modelling procedure that is being used to analyze this flow.

Modelling Combustor/Nozzle Flows. Ideally, the flow from the injector face to the nozzle

exit should be modelled, including the effect of atomization and mixing of the liquid propellents. However, this degree of computational complexity is expensive, and the FDNS code does not have this ability. An alternative is to remove the injector face-plate from the model and instead of liquid propellant, "inject" the equivalent products of combustion. These products of combustion are obtained from the NASA ODE deck, which determines the products of combustion assuming equilibrium reactions at the specified chamber pressure. Also, the gas velocity is obtained from the ODE deck, which is based on isentropic flow through the chamber/throat.

From the ODE deck, the following results are obtained. The first column corresponds to the test conditions from September 1990 [6], while the second and third columns are the core and wall conditions corresponding to the planned firing of the 40K nozzle in 1Q93.

O/F	6.00	7.5	4.0
P_c (PSIA)	1775	1750	1750
T (R)	6422	6629	5392
ρ (slug/ft ³)	0.01055	0.01164	0.00918
γ	1.1442	1.1336	1.1929
Mach No.	0.203	0.203	0.203
U (ft/sec)	1055.8	993.4	1147.3
H ₂ O (α_1)	0.6723	0.7213	0.4970
O ₂ (α_2)	0.0032	0.0223	0.0000
H ₂ (α_3)	0.2483	0.1327	0.4894
O (α_4)	0.0030	0.0105	0.0000
H (α_5)	0.0313	0.0294	0.0100
OH (α_6)	0.0418	0.0835	0.0035

In the analysis of the combined chamber and nozzle, the chamber will be analyzed separately from the nozzle. This is because the gross details only are required in the chamber, but more detail, including modelling of the film injection region, are required for the nozzle. The table conditions are used as inlet conditions to the chamber. The resulting exit conditions from the chamber section will be used in a more detailed analysis of the nozzle.

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

- FDNS code from June 1992 contained defects which rendered it useless for modeling thermal effects near solid boundaries.
- The most recent FDNS shows promise for predicting q_w for high speed boundary layer flows.
- The most recent FDNS still must be proven for predicting q_w for film injections.

References

- [1] 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.
- [2] 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.
- [3] 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.

- [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.
- [6] C. Dexter, "Private communication," August 1992.

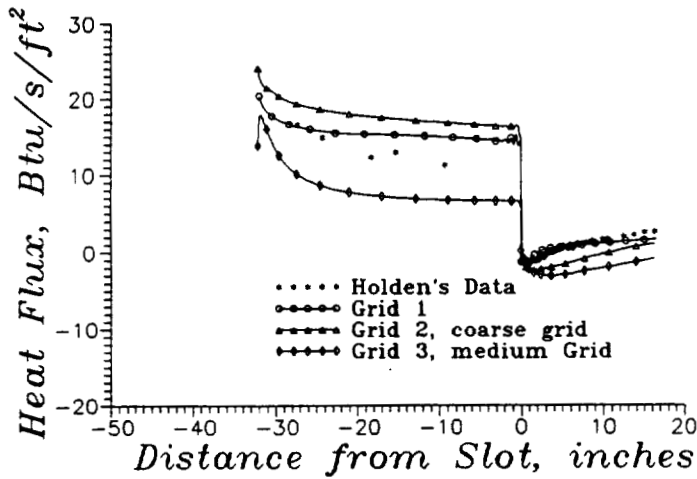
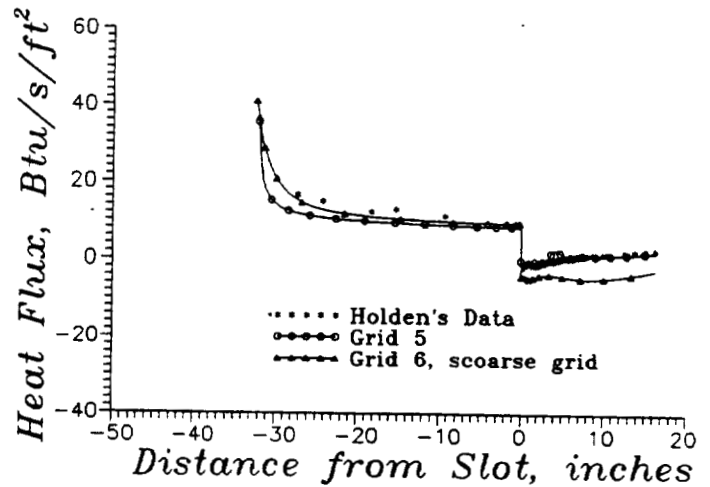


Figure 1: Results from Older FDNS
Version for Different Grids

Figure 2: Preliminary Results from
Newest FDNS Version



P242-027 HEAT FLUX

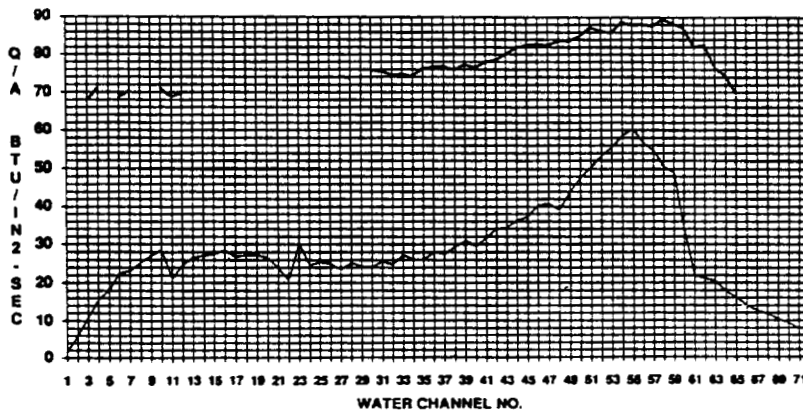


Figure 3: Experimental Data for
40K Calorimeter Chamber