N89-17303

AEROTHERMAL MODELING PROGRAM - PHASE II ELEMENT A: IMPROVED NUMERICAL METHODS FOR TURBULENT VISCOUS RECIRCULATING FLOWS

K.C. Karki, H.C. Mongia, S.V. Patankar*, and A.K. Runchal**
Allison Gas Turbine Division
General Motors Corporation
Indianapolis, Indiana

The objective of the NASA-sponsored Aerothermal Modeling Program, Phase II--Element A, is to develop improved numerical schemes for predicting combustor flow field. The effort consists of three technical tasks. Tasks 1 and 2 have been completed. Task 3 is in progress.

TASK 1--NUMERICAL METHOD SELECTION

Task 1 involved the evaluation of various candidate numerical schemes and selection of the promising schemes for detailed assessment under Task 2. The criteria for evaluation included accuracy, computational efficiency, stability, and ease of extension to multidimensions. The candidate schemes were assessed against a variety of simple one- and two-dimensional problems. These results led to the selection of the following schemes for further evaluation:

- o flux-spline schemes (linear and cubic)
- o Controlled Numerical Diffusion with Internal Feedback (CONDIF)

To improve the computational efficiency, a direct inversion technique was also selected for further testing. In this approach, the continuity and momentum equations are solved directly, rather than sequentially.

TASK 2--TECHNIQUE EVALUATION

Task 2 involved an in-depth evaluation of the numerical schemes selected in Task 1. The accuracy was judged by solving test problems for which reference solutions are available. The test cases included problems of scalar transport, laminar flows, and turbulent flows. These results indicated superior performance of the improved schemes. From scalar transport problems it was seen that the cubic flux-spline results were more accurate than those from the linear flux-spline. However, the cubic spline involved much more computational and programming effort and was not considered for fluid flow calculations.

For all the test problems, the linear flux-spline results were more accurate than the CONDIF results. The flux-spline scheme exhibited mild oscillations in the regions of steep gradient. However, it was felt that the presence of physical diffusion would tend to diminish these oscillations.

* University of Minnesota
**Analytic and Computational Research, Inc.

To improve the computational efficiency, the flux-spline (linear) was combined with a direct inversion technique using the Yale Sparse Matrix Package (YSMP) [ref 1]. Use of such a technique resulted in a factor of 2 to 3 reduction in the computational effort compared to the sequential solvers. A summary of the Task 2 effort is presented in references 2 and 3.

TASK 3-3D COMPUTATIONAL EVALUATION

Task 3, currently in progress, involves the incorporation of the flux-spline scheme and direct solution strategy in a computer program for 3D flows.

Due to the large storage requirement for the LU factorization, it is not possible to invert the continuity and momentum equations for the entire 3D field. Consequently, a plane-by-plane solution strategy was devised in which the cross stream (in-plane) velocities and pressure are solved in a coupled manner and the axial velocity is solved decoupled. However, the axial momentum and continuity equations are satisfied simultaneously. Such a procedure used in conjunction with the power-law scheme [ref 4] for convection-diffusion was found to be fast convergent and robust. Work is continuing on the use of the flux-spline scheme.

To demonstrate the accuracy of the flux-spline scheme for 3D flows, results are presented for the following two test cases:

- o radial heat conduction in a rotating hollow sphere
- o shear-driven laminar flow in a cubic cavity

Radial Heat Conduction in a Rotating Hollow Sphere

The problem is shown schematically in figure 1. A hollow sphere with its center located at the origin of a fixed Cartesian coordinate system rotates about the x-axis with a constant angular velocity $\omega = \omega \vec{e}_x$. The radius of the the inner surface is r_1, and is maintained at a uniform temperature T_1; r_2 is radius of the outer surface, which is at temperature T_2. For the case considered here, the radius ratio (r_2/r_1) is taken as 2.

With uniform properties under steady state, the temperature distribution is given by:

$$\theta = \frac{T - T_2}{T_1 - T_2} = \frac{2}{(r/r_1)} - 1 \qquad (1)$$

This problem, which is one-dimensional in the radial direction, appears three-dimensional if formulated in a Cartesian coordinate system. The calculation domain selected is shown as R in figure 1. The calculation domain is assumed to be fixed in space, so that the material within R has a steady velocity field given by:

$$\vec{V} = (\omega \vec{e}_x) X (x \vec{e}_x + y \vec{e}_y + z \vec{e}_z) \qquad (2)$$

The exact temperature distribution in Cartesian coordinates is obtained by transforming equation (1) to Cartesian coordinates:

$$\theta = \frac{2\, r_1}{\sqrt{x^2 + y^2 + z^2}} - 1 \qquad (3)$$

A uniform 11 x 11 x 11 grid was used to discretize the computational domain. Results were obtained for a range of Peclet numbers ($Pe = \rho\omega r_1^2/\Gamma$) and compared with the power-law scheme. Table I shows the error at the center point of the domain. The error has been defined as:

$$\varepsilon = \frac{|T_{computed} - T_{exact}|}{(T_{max} - T_{min})_{exact}} \times 100$$

Shear-Driven Laminar Flow in a Cubic Cavity

The flow situation under consideration is shown in figure 2. Due to symmetry considerations, the computational domain extended only half cavity width in the lateral (z) direction.

The flow Reynolds number is 400 and a uniform 22 x 22 x 12 (x, y, z) grid is employed for computations. The present results have been compared with the solution of Ku et al. [ref 5] obtained using a pseudospectral method (25 x 25 x 13 mode). This solution has been designated as "REFERENCE" in subsequent figures.

Figure 3 shows the velocity profiles of the u-component on the vertical centerline and the v-component on the horizontal centerline of the plane Z = 0.5. It is seen that for the same number of grid points the flux-spline solution is more accurate than the (lower-order) power-law solution.

Computations for turbulent flows are in progress.

REFERENCES

1. Eisenstat, M. C., Gursky, M. C., Shultz, M. H. and Sherman, A. H.; Yale Sparse Matrix Package, II, The Nonsymmetric Codes, Research Report No. 114, Yale University Department of Computer Sciences, 1977.
2. Patankar, S. V., Karki, K. C. and Mongia, H. C.; Development and Evaluation of Improved Numerical Schemes for Recirculating Flows, AIAA-87-0061.
3. Runchal, A. K., Anand, M. S. and Mongia H. C.; An Unconditionally-Stable Central Differencing Scheme for High Reynolds Number Flows, AIAA-87-0060.
4. Patankar, S. V.; Numerical Heat Transfer and Fluid Flow, Hemisphere, 1980.
5. Ku, H. C., Hirsch, R. S., and Taylor, T. D.; A Pseudospectral Method for Solution of the Three-Dimensional Incompressible Navier-Stokes Equations, Journal of Computational Physics, Vol 70, 1987, pp 439-462.

Table I.—ERROR AT THE CENTER POINT OF THE COMPUTATIONAL DOMAIN

	Pe = 1	10	100	1000
Error (ϵ) power-law	4.095×10^{-3}	1.439×10^{-2}	2.922×10^{-1}	2.756×10^{-1}
Error (ϵ) flux-spline	1.365×10^{-3}	4.445×10^{-3}	1.609×10^{-4}	1.177×10^{-2}

RADIAL HEAT CONDITIONS IN A ROTATING HOLLOW SPHERE: (a) PROBLEM SCHEMATIC; (b) DOMAIN DISCRETIZATION PATTERN ($x_1 = y_1 = z_1 = r_1/3$; $x_2 = y_2 = z_2 = r_2/3$).

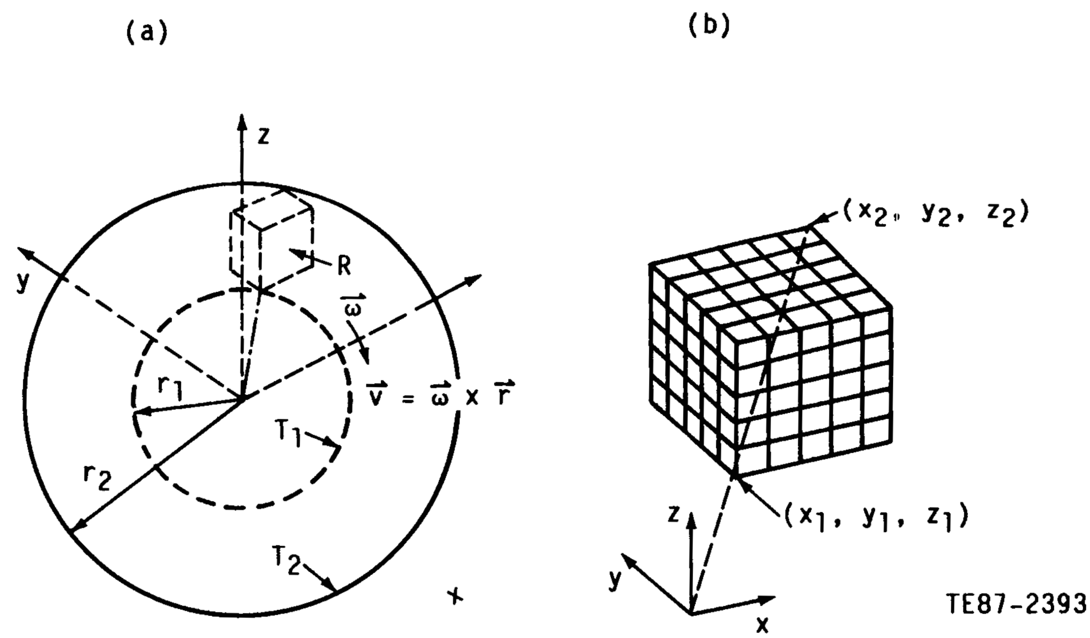

Figure 1

3D CAVITY FLOW CONFIGURATION

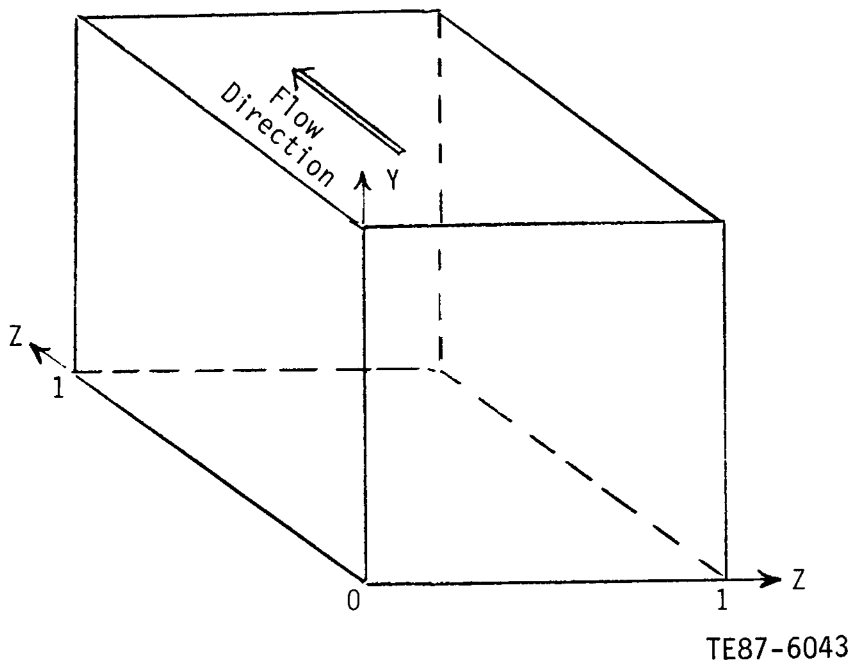

Figure 2

CUBIC CAVITY VELOCITY PROFILES FOR RE=400 ON (a) VERTICAL CENTERLINE;
(b) HORIZONTAL CENTERLINE

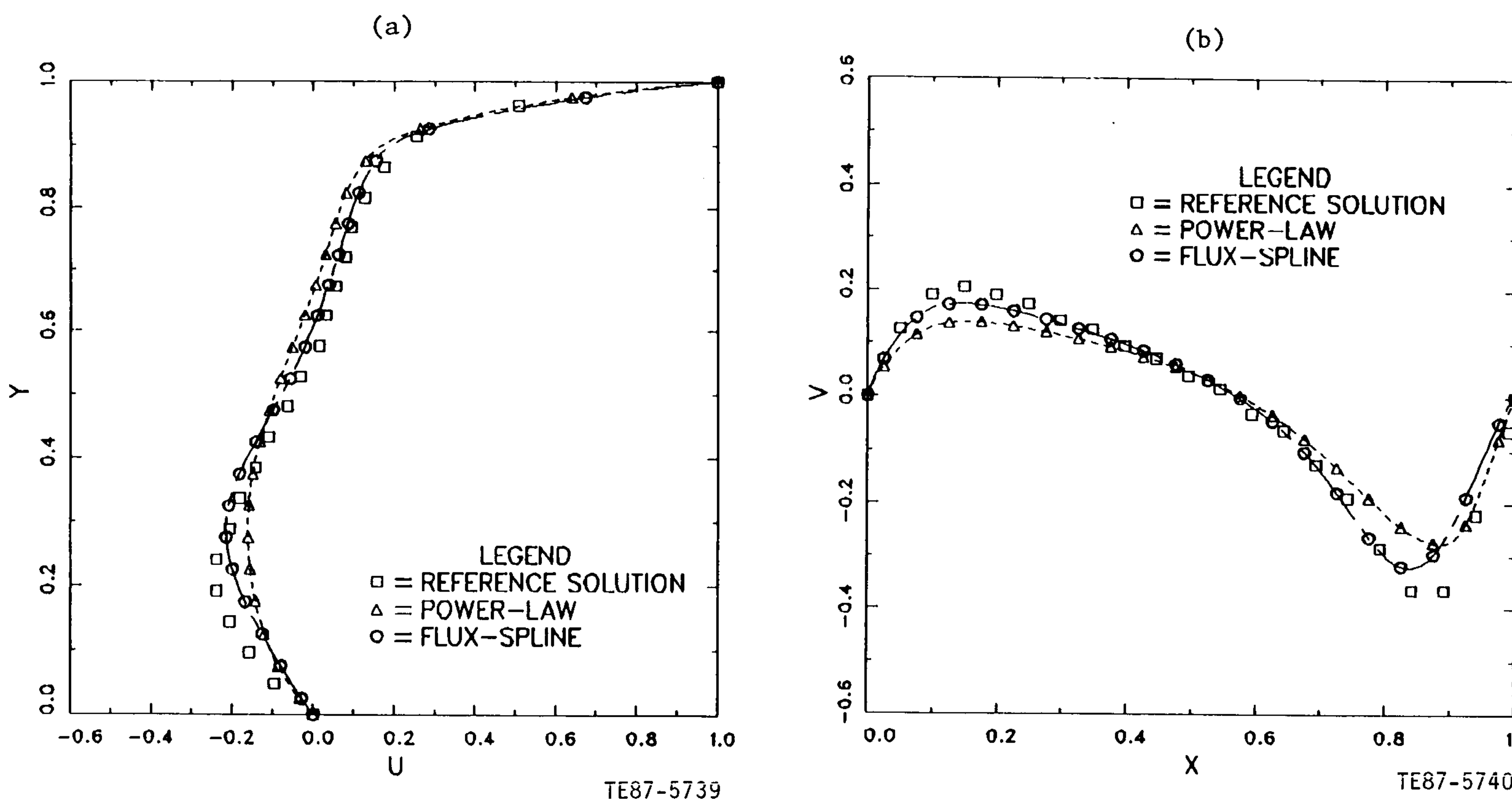

Figure 3