N89-12890

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 Corp.

The main objective of the NASA-sponsored Aerothermal Modeling Program, Phase II--Element A, is to develop an improved numerical scheme for predicting combustor flow fields. This effort consists of the following three technical tasks. Task 1 has been completed and Task 2 is in its final stage.

Task 1--NUMERICAL METHOD SELECTION

Task 1 involved the selection and evaluation of various candidate numerical techniques. The criteria for evaluation included accuracy, stability, boundedness, and computational efficiency. These schemes were used to solve a number of simple test problems. On the basis of these preliminary results, the following three schemes were chosen for detailed evaluation:

(i) flux-spline techniques
(ii) CONDIF
(iii) bounded flux-spline

To make the solution algorithm more efficient, it was decided to evaluate the performance of a fully coupled approach in which the continuity and momentum equations are solved directly, rather than sequentially as in SIMPLE or SIMPLER (ref. 1).

Task 2--TECHNIQUE EVALUATION

Task 2, currently in progress, involves an in-depth evaluation of the selected numerical schemes. The numerical accuracy and computational efficiency were judged using the test cases that have either analytical solutions, fine-grid numerical solutions, or experimental results. The following three classes of test problems were investigated:

(i) convection-diffusion (scalar transport)
(ii) laminar flows
(iii) turbulent flows

The results for each of these groups are summarized in the following paragraphs.

---

*Work done under NASA Contract NAS3-24350.
†University of Minnesota.
‡Analytical and Computational Research, Inc.

Convection-Diffusion

The test problems for convection-diffusion included (a) the transport of a step change in a scalar in a uniform velocity field, and (b) recirculating flow in a cavity with prescribed source for temperature. For both these problems, the higher-order schemes (flux-spline and CONDIF) give more accurate results than the hybrid scheme over the entire range of Peclet numbers (see Figures 1 through 4). For the transport of the step change in the scalar, the flux-spline schemes exhibit undershoots and overshoots.

Laminar Flows

Two selected laminar flow test cases are: (a) driven cavity and (b) flow over a backward facing step.

The driven cavity problem was solved using a 22 x 22 uniform grid and the results were compared with a 82 x 82 hybrid solution. The higher-order schemes give much better accuracy and show substantial advantage over the hybrid scheme (see Figures 5 and 6). Computations for the flow over a backward facing step were made at two Reynolds numbers (Re = 100 and 715) and the results were compared with the experimental data (ref. 2).

At Re = 100, there is negligible false diffusion and the results of all schemes compare well with experiments. At Re = 715, the higher-order schemes predict a longer reattachment length, compared to the hybrid scheme, indicating smaller numerical diffusion. There is, however, disagreement between the numerical and experimental results. These deviations are probably due to the presence of three-dimensional effects in the experiments (ref. 2).

Turbulent Flows

The selected schemes were used to compute the Stanford Conference test case 0421 (flow over a backward facing step) (ref. 3). The computed reattachment lengths from various schemes are listed in Table I. In these computations, plug flow was assumed at the inlet. It is seen that the flux-spline scheme approaches a grid-independent solution with fewer grid points than the hybrid scheme. The improvement shown by the flux-spline technique, however, is not as large as in laminar flow cases. A similar trend in the results was noticed when the experimentally measured velocity profile was specified at the inlet.

Performance of the Coupled Solution Approach

The efficiency of a numerical technique based on the primitive variables depends to a great extent on the manner in which the velocity-pressure coupling is treated. The iterative methods (e.g., SIMPLE, SIMPLER) derive an equation for pressure and solve the momentum and pressure equations in a sequential manner. The convergence of such an approach is found to be slow. An alternative to this sequential approach is the direct solution of the whole set of continuity and momentum equations (ref. 4 and 5). This study evaluates the performance of a direct or coupled approach in conjunction with a flux-spline scheme for convection-diffusion.

In the present approach, the discretized continuity and momentum equations are treated as simultaneous equations and solved using the Yale Sparse Matrix Algorithm (YSMP) (ref. 6). The nonlinearities in the equations are handled using the successive substitution technique. In turbulent flow computations, the solution of the flow equations (using YSMP) is followed by the solution of the equations for the turbulence quantities (k and ϵ). The equations for k and ϵ are solved sequentially in a decoupled manner using a line-by-line tridiagonal matrix algorithm (TDMA). The sequence of calculations is as follows: (1) the continuity and momentum equations are solved using YSMP, (2) with the given velocity field, the k and ϵ equations are solved to provide a new viscosity field for the momentum equations. This procedure is repeated until convergence.

Preliminary Results

The details of the test problems selected for the evaluation of the coupled solution approach are given in Table II. The number of iterations required for convergence and the execution times for the coupled solver are compared with those for the SIMPLER approach in Table III. The results indicate that the direct solver gives a speed-up factor of about three for laminar flows and five for turbulent flows. Further evaluation on a finer grid is under progress.

TASK 3

The convection-diffusion scheme with superior performance in Task 2 and the direct solver will be incorporated in the NASA 3-D elliptic code (COM3D). A test case will be run to assess the accuracy and computational efficiency of the selected scheme/algorithm for three-dimensional situations.

REFERENCES

1. Patankar, S. V., Numerical Heat Transfer and Fluid Flow, Hemisphere, 1980.
2. Armaly, B. F., Durst, F., Pereira, J. C., and Schonung, B., "Experimental and Theoretical Investigation of Backward Facing Step Flow," Journal of Fluid Mechanics, Vol 127, 1983, pp 473-496.
3. Kline, S. J., Cantwell, B. J., and Lilley, G. M. (eds), The 1980-81 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows, Stanford University.
4. Vanka, S. P., "Block Implicit Calculation of Steady Turbulent Recirculating Flows," International Journal of Heat and Mass Transfer, Vol 28, 1985, pp 2093-2103.
5. Braaten, M., Development and Evaluation of Iterative and Direct Methods for the Solution of the Equations Governing Recirculating Flows, Ph.D Thesis, University of Minnesota, May 1985.
6. Eisenstat, M. C., Gursky, M. C., Schultz, M. H., and Sherman, A. H., "Yale Sparse Matrix Package, II, The Nonsymmetric Codes," Research Report No. 114, Yale University, Department of Computer Science, 1977.

Table I.
Calculated reattachment lengths [x_R/h] (plug flow at the step).

Grid	Hybrid	CONDIF	Flux-spline
32 x 32	4.4	4.2	4.6
40 x 40	5.0	4.5	5.3
57 x 57	5.2	---	5.3

Table II.
Laminar flow test cases.

Case No.	Flow	Reynolds number	Grid (uniform)
1	Driven cavity	400	22 x 22
2	Driven cavity	1000	22 x 22
3	Sudden expansion	400	22 x 12
4	Sudden expansion	400	22 x 22
	Turbulent flow test cases		
5	Backward facing step	5.6×10^5	22 x 22

Table III.
No. of iterations required and execution times.

Case No.	No. of iterations		Execution times (seconds)	
	SIMPLER	Direct	SIMPLER	Direct
1	62	17	18	6
2	84	30	24	8
3	106	47	16	5
4	122	48	35	10
5	800	39	408	79

IBM 3084

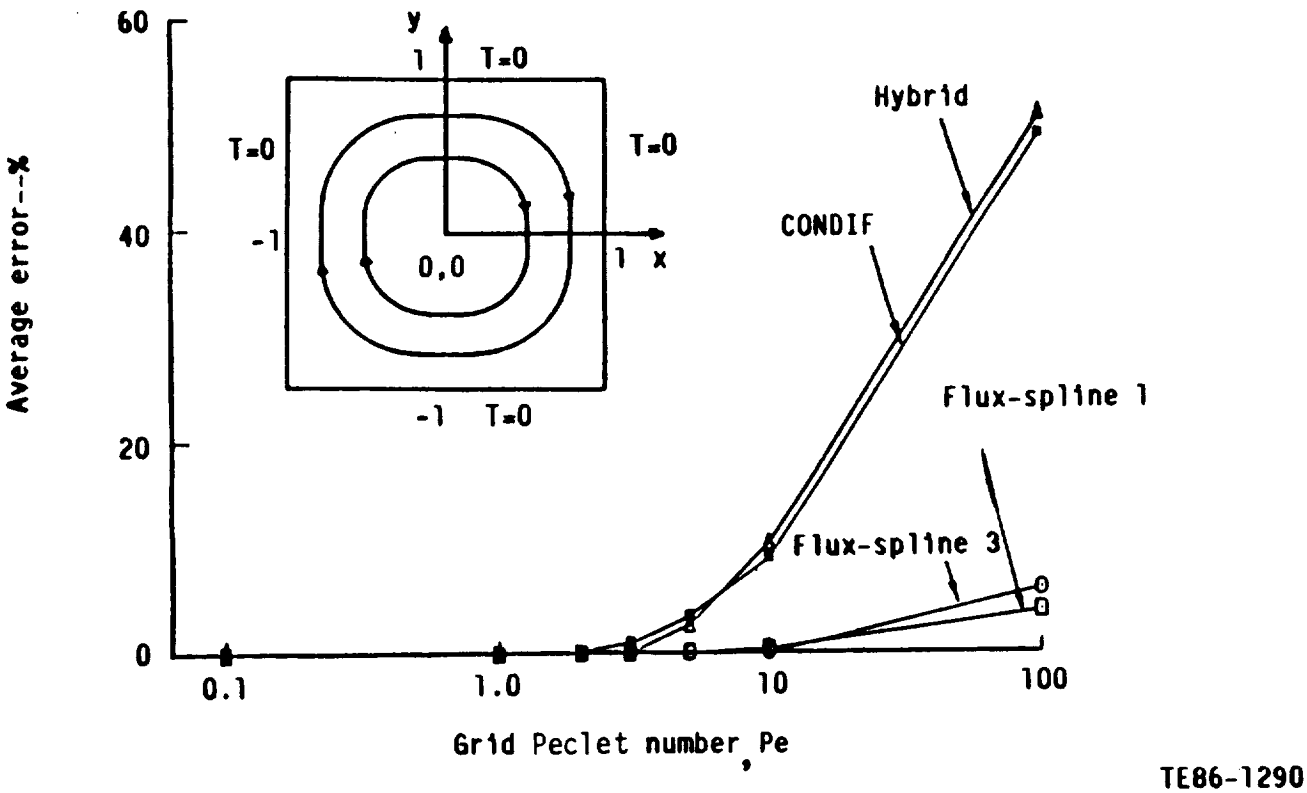


Figure 1. Comparison of the average error in the temperature calculated using different schemes. Recirculating flow with prescribed source for temperature.

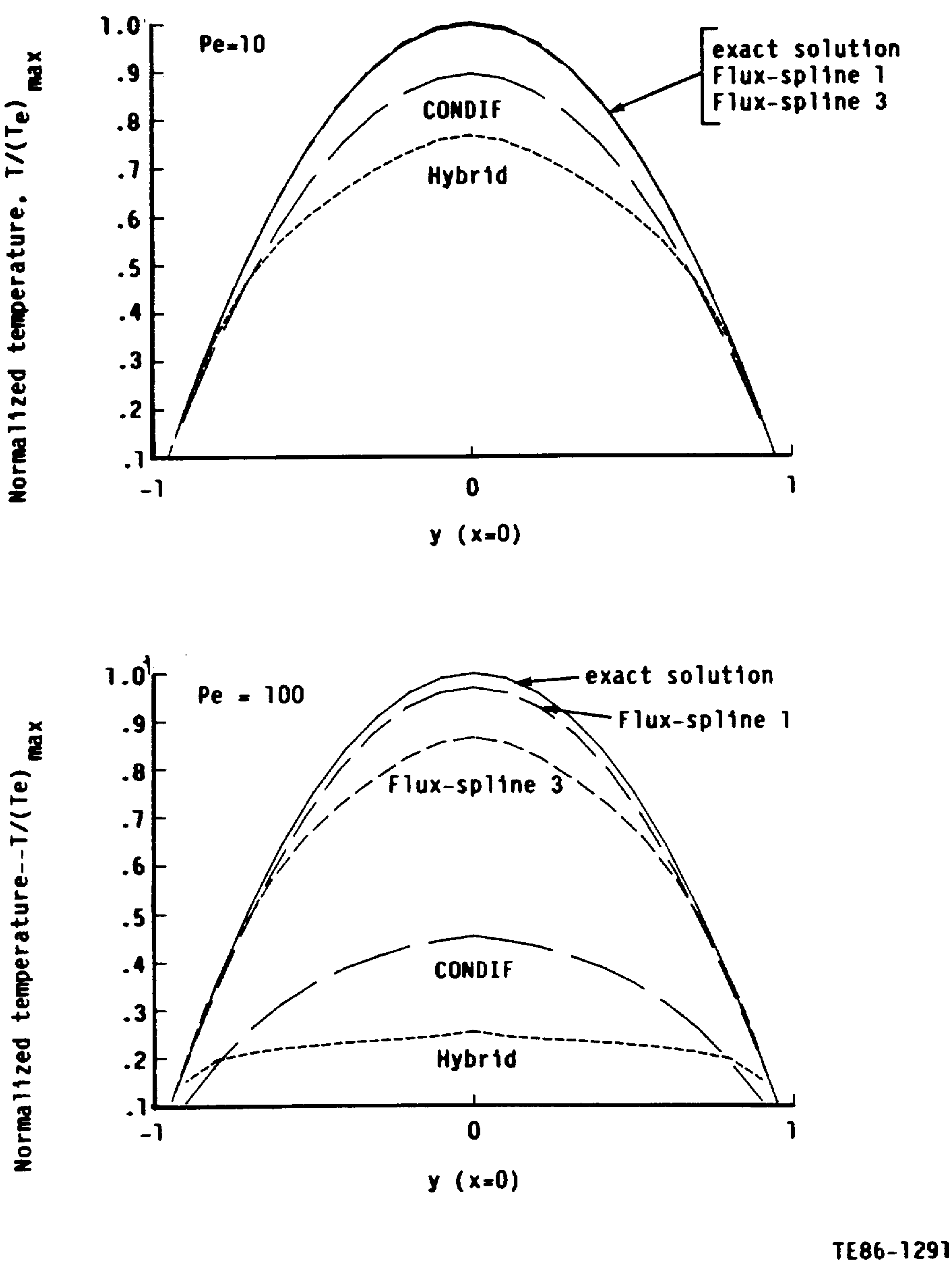
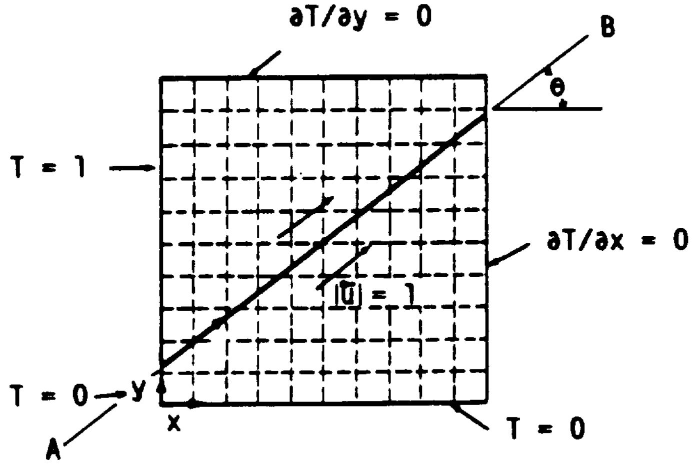
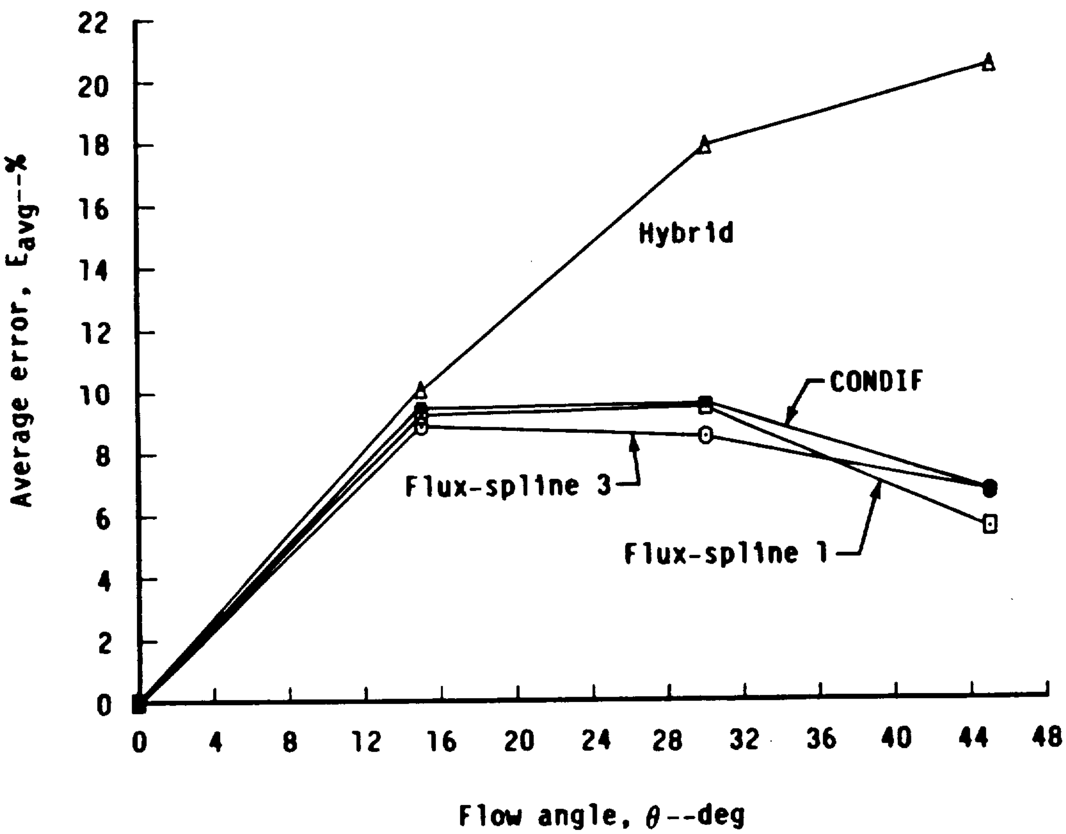


Figure 2. Comparison of the temperature profiles calculated using different schemes, with the exact solution. Recirculating flow with prescribed temperature source and velocity.



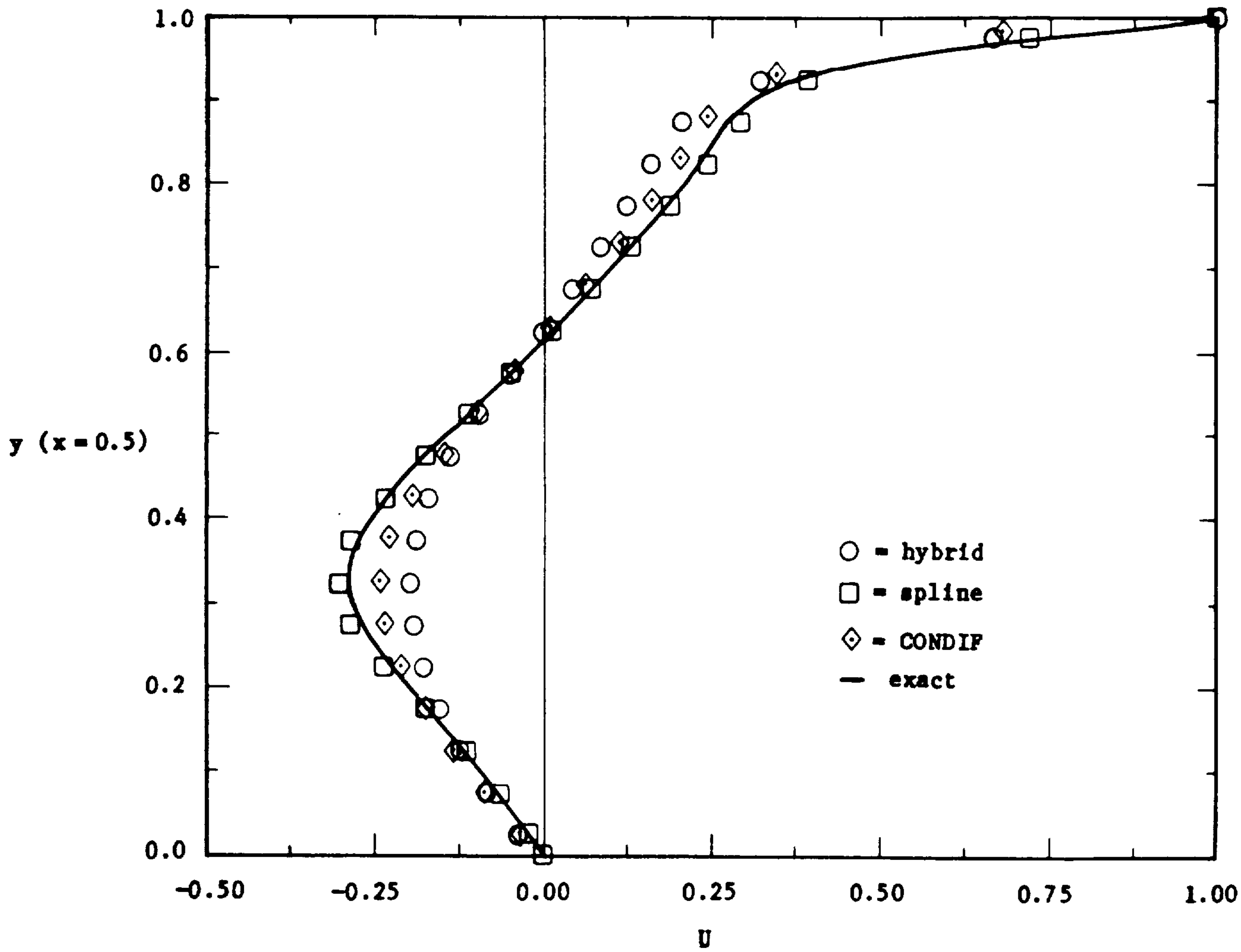
TE85-4217

Figure 3. Transport of a step change in temperature in a uniform flow field.



TE86-1292

Figure 4. Comparison of the average error in the temperature calculated from the different schemes.



TE86-3511

Figure 5. Normalized U-velocity at x = 0.5.

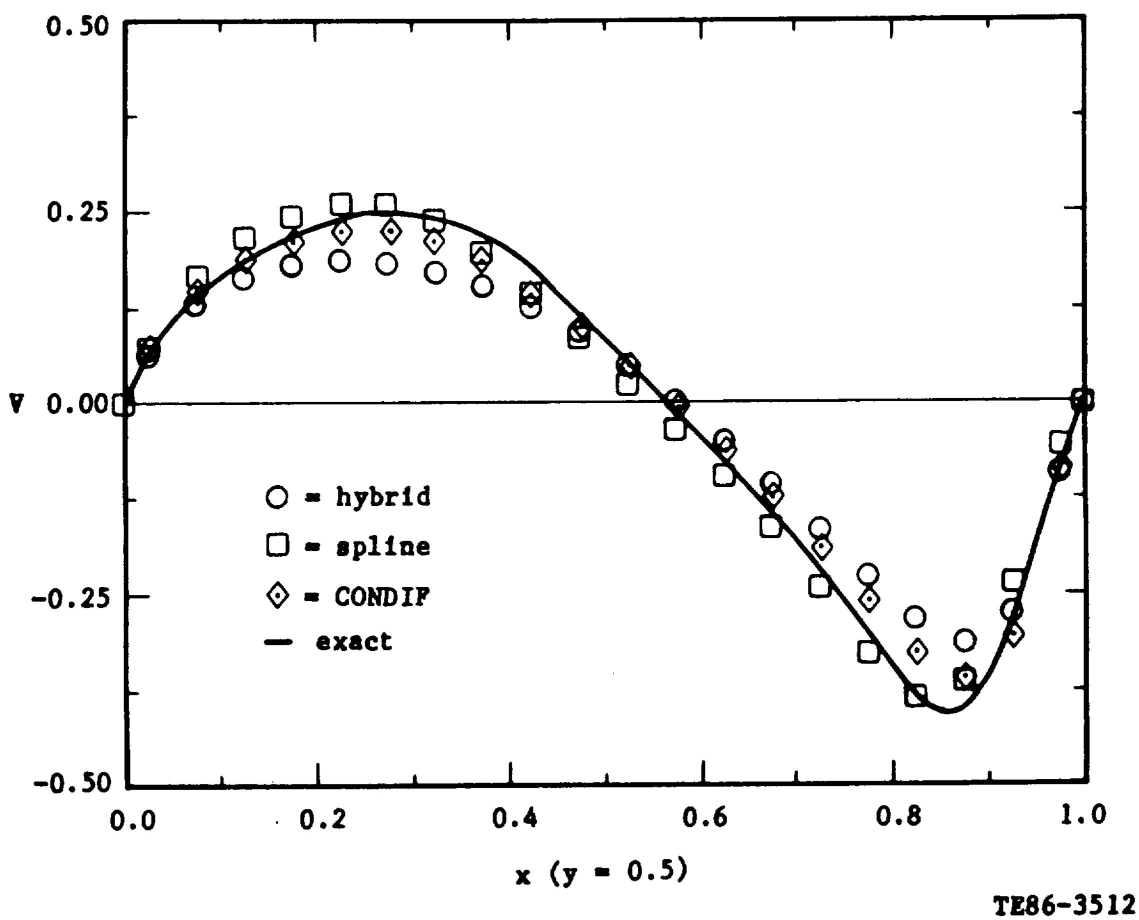


Figure 6. Normalized V-velocity at y = 0.5.