NASA Technical Memorandum 86863

# Accelerated Convergence for Incompressible Flow Calculations



Geralyn M. Neely and Russell W. Claus
*Lewis Research Center*
*Cleveland, Ohio*


Prepared for the
Twenty-third Aerospace Sciences Meeting
sponsored by the American Institute of Aeronautics and Astronautics
Reno, Nevada, January 14–17, 1985

NASA

NASA Technical Memorandum 86863

# Accelerated Convergence for Incompressible Flow Calculations



Geralyn M. Neely and Russell W. Claus
*Lewis Research Center*
*Cleveland, Ohio*


Prepared for the
Twenty-third Aerospace Sciences Meeting
sponsored by the American Institute of Aeronautics and Astronautics
Reno, Nevada, January 14-17, 1985

NASA

# ACCELERATED CONVERGENCE FOR INCOMPRESSIBLE FLOW CALCULATIONS

Geralyn M. Neely and Russell W. Claus
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135



## Abstract

Two improved algorithms which solve the steady-state Navier-Stokes equations, PISO and SIMPLER, are studied in this paper. Computations were carried out on progressively finer grids for the driven cavity and flow over a backward-facing step. The effects of relaxation factor, number of grid nodes and number of sweeps through the pressure equations are studied to evaluate the performance of the PISO and SIMPLER schemes. Results show that these improved schemes accelerate the convergence rate of the solution generally by a factor of two as compared to the SIMPLE method.

## Introduction

Although currently available combustor models have already proven useful as analytical tools, these codes are not yet cost-effective alternatives to traditional design methods. One area in which combustor codes can be improved is in the execution time required for the solution algorithm. Typically, the method used to solve the steady-state Navier-Stokes equations is an iterative procedure, which, for stability reasons, must be underrelaxed. The widely-used SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) scheme, developed by Patankar and Spalding in 1972, is one such solution algorithm.[1] Although this method has been demonstrated to be quite effective, its convergence rate can be improved. This investigation focuses on two alternate approaches which accelerate the solution to the steady-state Navier-Stokes equations.

A short discussion of the solution algorithms studied, SIMPLE, SIMPLER (SIMPLE Revised),[2] and PISO (Pressure Implicit Split Operator),[3] is included in this report. Results are presented for the driven cavity and flow over a backward-facing step. The SIMPLE scheme is used as a basis for comparison to evaluate the performance of these algorithms.

## Symbol List

| | |
|---|---|
| $C_\mu, C_{\epsilon_1}, C_{\epsilon_2}$ | turbulence model constants |
| $k$ | turbulence kinetic energy |
| $p$ | pressure |
| Re | Reynold's number $= \frac{\rho u h}{\mu}$ |
| $u_i$ | fluctuating velocity about mean |
| $U_i$ | mean velocity |
| $\epsilon$ | turbulence energy dissipation rate |
| $\varphi$ | any of the independent variables |
| $\rho$ | density |
| $\mu_t$ | turbulent viscosity |
| $\sigma_t$ | turbulent Prandtl number |

## Mathematical Model

The TEACH code, developed at Imperial College, is the solution procedure used in this study.[4] This algorithm solves partial differential equations which describe the fluid flow.

### Governing Equations

The governing equations for two-dimensional, steady-state, incompressible, turbulent flow are shown below. The turbulence model employed is the two-equation $k$-$\epsilon$ model developed by Jones and Launder.[5]

#### Continuity

$$\frac{\partial U_i}{\partial x} = 0$$

#### Momentum

$$\underbrace{U_i \frac{\partial U_i}{\partial x_j}}_{\text{Convection}} = \underbrace{-\frac{1}{\rho}\frac{\partial p}{\partial x}}_{\text{Pressure gradient}} + \underbrace{\frac{\partial}{\partial x_i}\left(\frac{\mu}{\rho}\frac{\partial U_j}{\partial x_i} - \overline{u_i u_j}\right)}_{\text{Diffusion}}$$

#### Scalar transport

$$\underbrace{U_i \frac{\partial \varphi}{\partial x_i}}_{\text{Convection}} = \underbrace{\frac{1}{\rho}\left(\frac{\partial}{\partial x_i}\frac{\mu}{\sigma_t}\frac{\partial \varphi}{\partial x_i} - \overline{u_i \varphi}\right)}_{\text{Diffusion}}$$

#### Turbulence model

$$\overline{u_i u_j} = \frac{2}{3}\frac{\delta_{ij}}{\rho}k - \frac{\mu_t}{\rho}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)$$

$$\mu_t = C_\mu \rho \frac{k^2}{\epsilon}$$

$$\overline{u_i \varphi} = -\frac{1}{\rho}\frac{\mu_t}{\sigma_t}\frac{\partial \varphi}{\partial x_i}$$

$$U_i \frac{\partial k}{\partial x_i} = \frac{1}{\rho}\frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) - \overline{u_i u_j}\frac{\partial U_j}{\partial x_i} - \epsilon$$

$$U_i \frac{\partial \epsilon}{\partial x_i} = \frac{1}{\rho}\frac{\partial}{\partial x_i}\left(\frac{\mu_t}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_i}\right) - C_{\epsilon_1}\frac{\epsilon}{k}\overline{u_i u_j} - C_{\epsilon_2}\frac{\epsilon^2}{k}$$

## Solution Algorithms

A staggered grid arrangement is set up to obtain the finite difference approximations.[6] Scalar variables such as pressure and turbulence kinetic energy are stored at mesh points, while velocities are stored at control volume faces. Integration is performed over the control volume surfaces. The partial differential equations are discretized using the hybrid (upwind/central) differencing scheme for the convective terms, and central differencing for the other terms.

## SIMPLE

In the SIMPLE algorithm, a guessed pressure field is inserted into the discretized momentum equations to obtain a velocity field. The pressure field is corrected via an equation which is derived through a combination of the discretized continuity and momentum equations. The velocity field is then updated and is used to solve equations for $k$, $\epsilon$ and $\varphi$. The corrected pressure field is treated as the guessed pressure field and the procedure is repeated until a converged solution is obtained.

The velocity correction equation used in the SIMPLE scheme for $u$ at the point $e$ is the following:

$$A_e u'_e = \underline{\sum_{nb} A_{nb} u'_{nb}} + (p'_p - p'_E) A_e$$

Primes indicate corrections to old values. The underlined term, which represents the influence of corrected pressures on neighboring velocities, is neglected in the SIMPLE algorithm. To include this term would entail solving pressure correction equations for the whole flow field. On the other hand, the converged solution is unchanged by the exclusion of this term since this equation is merely used to correct the velocity. Doing this, however, results in an overestimation of the pressure field, requiring relaxaton of the velocity equations to obtain a converged solution. Relaxation factors less than 1.0 decelerate changes in the velocity from one iteration to the next. The lower the relaxation factor, the more slowly convergence is reached. The SIMPLE scheme necessitates the choice of very low relaxation factors to obtain convergence.

## SIMPLER

The SIMPLER algorithm improves upon the SIMPLE scheme by including the neglected terms in calculating the pressure field. The calculation sequence starts with a guessed velocity field. As in the SIMPLE scheme, an equation for the pressure field which combines the discretized momentum and continuity equations is solved. This pressure field is inserted into the discretized momentum equations to obtain a velocity field. A corrected pressure field is calculated and used to update the velocity field. This velocity field is treated as the guessed velocity field and the iteration procedure continues until convergence is reached.

Because two additional equations are solved, each iteration step through the SIMPLER routine involves more computational time than an iteration step through the SIMPLE procedure. However, higher relaxation factors can be applied in the SIMPLER routine and therefore, convergence is much faster.

## PISO

The PISO scheme also takes into account the terms neglected in the SIMPLE code, but in a different manner. The PISO routine mimics the SIMPLE code up until the end of the first iteration. At this point, the PISO scheme employs an equation containing the neglected terms which corrects the pressure. The velocity is then corrected and updated. Again, this procedure is repeated until the solution converges. For each iteration, the PISO scheme more closely satisfies continuity and momentum than the SIMPLE scheme, due to the additional pressure and velocity corrections. In this manner, the PISO code allows for higher relaxation factors to accelerate convergence.

## Results

The main thrust of this investigation is to analyze methods which speed the convergence of incompressible flow calculations. In order to accomplish this, several factors are considered. The most significant element in the optimization process is the relaxation factor. The choice of a too high relaxation factor may cause divergence; a too low relaxation factor results in a slow, costly calculation. Secondly, the optimum number of sweeps through the pressure equations per velocity iteration is determined. This optimum varies according to the exactness with which each scheme solves the pressure-velocity equations. A third factor to consider is the grid size. A dense mesh may require much longer computational time than a coarse one. All of the above analyses are done for laminar flows. Finally, the "stiffness" imposed by the turbulence model is studied.

The two configurations chosen for this study are the square driven cavity and flow over a backward-facing step. The velocity vectors for a typical driven cavity calculation (Re = 1000) are illustrated (Fig. 1). The top wall of the cavity moves at a constant velocity, u, while the 3 other walls remain motionless. The grid for these calculations is evenly spaced.

Figure 2 shows the geometry employed in the backward-facing step calculations. Computations for this geometry were performed at Reynolds numbers of 1000 (laminar) and 30 000 (turbulent), based on the channel inlet height of 3 h (see Fig. 2). Aspect ratios ($\Delta x/\Delta y$) for these calculations range from 1.0 to 5.0.

## Effect of Relaxation Factor

The importance of applying the optimum relaxation factor for each scheme to obtain the fastest convergence time is illustrated (Fig. 3). Residual error values are $1 \times 10^{-5}$ for the laminar driven cavity and $1 \times 10^{-4}$ for step flow calculations. Residual errors were chosen based on an overall error of 2 percent in the results. The minimum value on each curve represents the fastest convergence time and the corresponding optimum relaxation factor. The vertical lines indicate the relaxation factors at which the calculations become unstable. For the 38 by 38 driven cavity calculations, (Fig. 3(a)), the SIMPLER and PISO

algorithms have higher optimum relaxation factors and therefore, lower convergence times. Note, however, that if a lower relaxation factor, for example 0.5, is chosen, both the SIMPLER and PISO schemes would be slower to converge than the SIMPLE routine.

Similar results can be seen for the 62 by 22 step flow calculations (Fig. 3(b)). For these computations, the PISO code has the lowest convergence time. Note also that with the PISO scheme, one can be quite a bit far from the optimum relaxation factor with minimal effect. The opposite can be said of the SIMPLE algorithm, which has a very short range of relaxation factors which give reasonable results. Although the SIMPLER algorithm has a high optimum relaxation factor as compared to the SIMPLE code, its fastest convergence time is about equal to that of the SIMPLE code.

## Effect of Number of Pressure Sweeps

Once the optimum relaxation factor for each scheme was established, the calculations were optimized with respect to the number of sweeps through the pressure equations. The optimum relaxation factor for each scheme is used in all further comparisons. The optimum number of pressure sweeps varies from 5 to 10, depending on the scheme and flow situation. Table I shows typical variations in convergence times for different numbers of sweeps through the pressure equations for the PISO and SIMPLER algorithms. For the SIMPLE scheme, increasing the number of pressure sweeps beyond five resulted in an increased convergence time. Because of the neglected terms in the pressure correction equation, the pressure field at each iteration is quite inaccurate until the solution nears convergence. Sweeping the inaccurate pressure field several times does little to enhance convergence for the SIMPLE algorithm. Since the PISO and SIMPLER schemes both include the neglected terms, the equations are more exact and the pressure field at each iteration is not nearly as inaccurate as that in the SIMPLE code. By increasing the number of sweeps through the pressure equations, the pressure field becomes more accurate and thus speeds convergence.

## Effect of Grid Size

Computations were done for meshes ranging from 100 to 4700 nodes, with results shown in figure 4. All calculations were performed using the optimum relaxation factors and number of pressure sweeps determined previously. For the driven cavity (Fig. 4(a)) the PISO and SIMPLER schemes give nearly identical results, with the PISO scheme about 6 percent faster for the finest mesh. The relative efficiency, defined as

$$\frac{(\text{(CPU time for SIMPLE)} - \text{(CPU for scheme)})}{\text{(CPU for SIMPLE)}}$$

remained at around 60 percent for both PISO and SIMPLER throughout the calculations.

Laminar step flow results are shown in figure 4(b). The SIMPLER scheme has the same convergence times as the SIMPLE code for grid sizes up to 1800 nodes, and becomes faster for finer meshes, with an efficiency of about 40 percent at 4700 nodes. The PISO scheme efficiency improves from about 20 percent for the coarse mesh to 40 percent for the 4700 node calculation.

A convergence rate comparison (Fig. 5) gives insight into the directness or rapidity with which each scheme converges on a solution. These figures also indicate that the results presented herein are not sensitive to the choice of convergence level. Again, optimum relaxation factors and number of pressure sweeps are used to obtain these results. For each iteration step, the maximum residual error (made up of residuals of u momentum, v momentum and pressure) is established and graphed versus the execution time for that step. For the laminar driven cavity, the PISO and SIMPLER schemes have much steeper slopes and converge twice as fast as the SIMPLE algorithm for any given residual error value. In the step flow calculations, the PISO code again has a steeper slope and converges rapidly. The SIMPLE and SIMPLER codes performed similarly for this calculation.

## Turbulent Flow Calculations

To investigate the "stiffness" imposed by the turbulence model, calculations were performed for the step flow geometry at a Reynold's number of 30 000. "Stiffness" as it relates to turbulent flow calculations results from updating the turbulent viscosity at the end of each momentum pressure-correction iteration. In other words, the momentum equations are solved with the old values of turbulent viscosity and updated at the end of the iteration. This frequently slows convergence, i.e., imposes a "stiffness" on the calculation. These calculations (Fig. 6) were done for a residual error of $1 \times 10^{-2}$. The SIMPLER scheme gives the best performance, with the lowest convergence time at an optimum relaxation factor of 0 85 (Fig. 6(a)). The SIMPLER scheme also demonstrates a reasonable range of fast, stable operation for relaxation factors from about 0.80 to 0.90. Although the PISO code improves upon the convergence time of the SIMPLE scheme by a factor of 2, it appears to have a much shorter range of stable operation than either of the other two schemes.

For a series of increasingly finer meshes (Fig. 6(b)) the SIMPLER scheme speeds convergence by a factor of 3 with relative efficiencies ranging from 50 percent for the coarsest mesh to 60 percent for the finest mesh. The PISO scheme is faster than the SIMPLE scheme by a factor of 2 and has a relative efficiency of 45 percent for all grid sizes.

A convergence rate comparison for the turbulent flow case, again using the optimum relaxation factors for each scheme (Fig. 7), shows the PISO and SIMPLE schemes with steeper slopes and faster convergence times than the SIMPLER scheme for maximum residual errors up to $1 \times 10^{-2}$.

## Conclusions and Recommendations

Two improved solution algorithms, PISO and SIMPLER, have been tested in two flow situations, the driven cavity and flow over a backward-facing step. Effects of relaxation factor, number of

pressure sweeps and number of grid nodes on the convergence time of these schemes have been analyzed. These results show:

1. The implementation of the PISO or SIMPLER scheme accelerates convergence generally by a factor of 2 as compared to the SIMPLE algorithm.

2. The efficiency relative to the SIMPLE scheme of the PISO and SIMPLER algorithms improves as more nodes are added to the flow field.

3. The efficiency of the PISO and SIMPLER schemes is dependent upon the relaxation factor, flow situation and the number of sweeps through the pressure equations.

Obviously, since these solution algorithms are tested here for two flow cases, these results cannot be taken as universal. However, they are useful in giving insight and making some general conclusions which should apply in similar flow situations. For a given class of problems, once the optimum relaxation factor and number of pressure sweeps has been determined, we recommend that these numbers be used for all further calculations in this class of problems.

For general engineering use, it is recommended that a relaxation factor of 0.85 will give reasonable results for both the PISO and SIMPLER methods for laminar flow. The number of pressure sweeps to give fastest convergence is 10 for the SIMPLER method and 5 for the PISO method. For turbulent flow, a lower relaxation factor of 0.75 for the PISO scheme and 0.85 for the SIMPLER scheme should give optimal results. Calculations for the SIMPLE method should be done with a relaxation factor of 0.5 and 5 pressure sweeps for both laminar and turbulent flow.

As a final note, it has been observed that the accuracy with which the pressure correction equations are solved affects the convergence time for the PISO and SIMPLER schemes. A faster tridiagonal matrix algorithm solver may be implemented to make both of these schemes even more attractive.

## References


1. Patankar, S.V. and Spalding, D.B., "A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows," International Journal of Heat and Mass Transfer, vol. 15, Oct. 1972, pp. 1787-1806.

2. Patankar, S.V., "A calculation procedure for two-dimensional elliptic situations," Numerical Heat Transfer, vol. 4, Oct.-Dec. 1981, pp. 409-425.

3. Issa, R.I., "Solution of implicitly discretised fluid flow equations by operator-splitting," Internal Report, Dept. of Mineral Resources Engineering, Imperial College, London, 1982.

4. Gosman, A.D. and Ideriah, F.J.K., "TEACH-2E: A general computer program for two-dimensional, turbulent, recirculating flows," Department of Mechanical Engineering Report, Imperial College, London, 1976.

5. Jones, W.P., and Launder, B.E., "The calculation of low-Reynold's-number phenomena with a two-equation model of turbulence," International Journal of Heat and Mass Transfer, Vol. 16, June 1973, pp. 1119-1130.

6. Harlow, F.H., and Welch, J.E., "Numerical calculation of time-dependent viscous incompressible flow of fluids with free surface," Physics of Fluids, Vol. 8., Dec. 1965, pp. 2182-2189.

TABLE I. - DEPENDENCE OF CONVERGENCE TIME ON NUMBER OF PRESSURE SWEEPS

Driven Cavity 38x38

PISO

| Relaxation factor | No. pressure sweeps | Conv. time, sec |
|---|---|---|
| 0.85 | 5 | 74.2 |
| | 10 | 104.1 |
| | 20 | 136.0 |
| SIMPLER | | |
| .93 | 5 | 94.9 |
| | 7 | 83.6 |
| | 10 | 78.5 |
| | 12 | 85.2 |

Laminar flow over a step 62x22

PISO

| Relaxation factor | No. pressure sweeps | Conv. time, sec |
|---|---|---|
| .93 | 5 | 139.8 |
| | 7 | 139.1 |
| | 10 | 111.5 |
| | 15 | 151.4 |
| SIMPLER | | |
| .86 | 5 | 196.6 |
| | 7 | 206.7 |
| | 10 | 173.9 |
| | 15 | 224.9 |

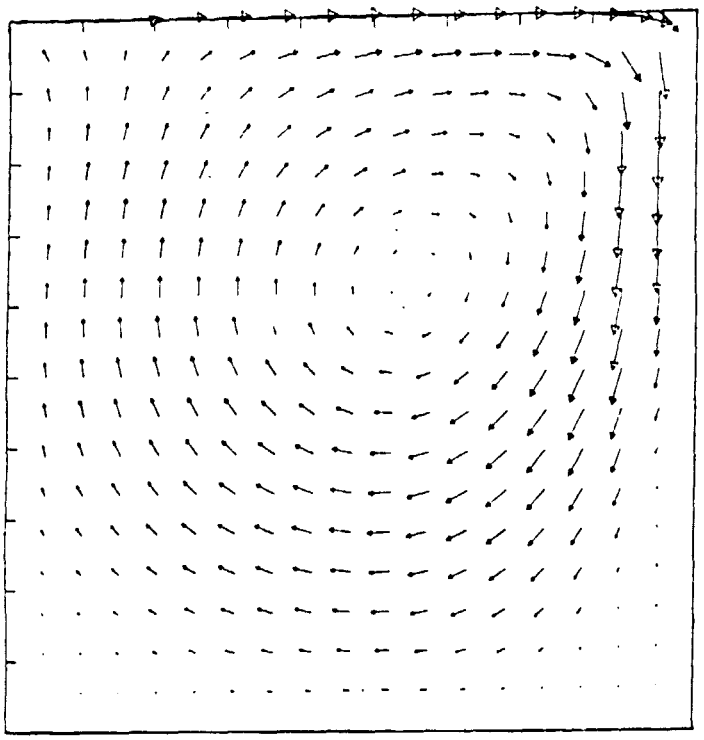


Figure 1. - Velocity vectors for driven cavity calculations.

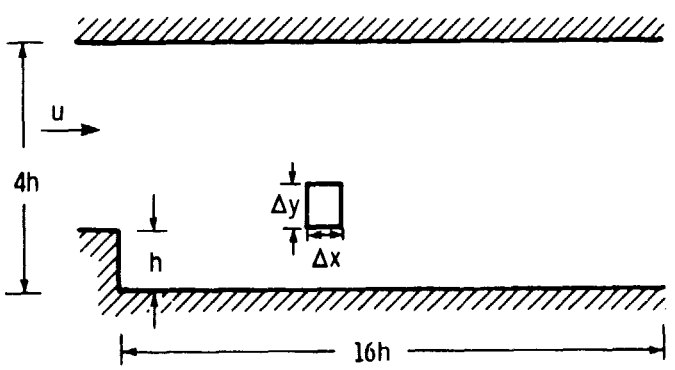



Figure 2. - Backward-facing step geometry.

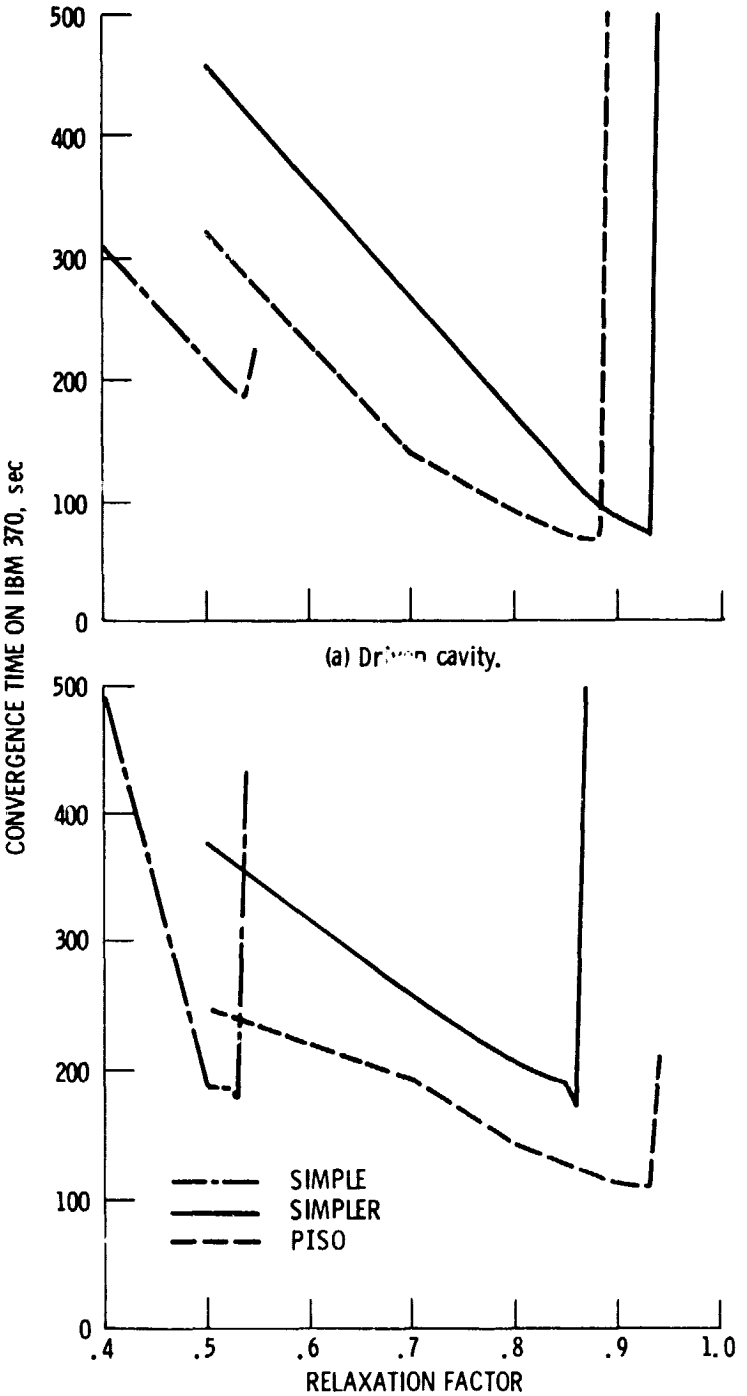
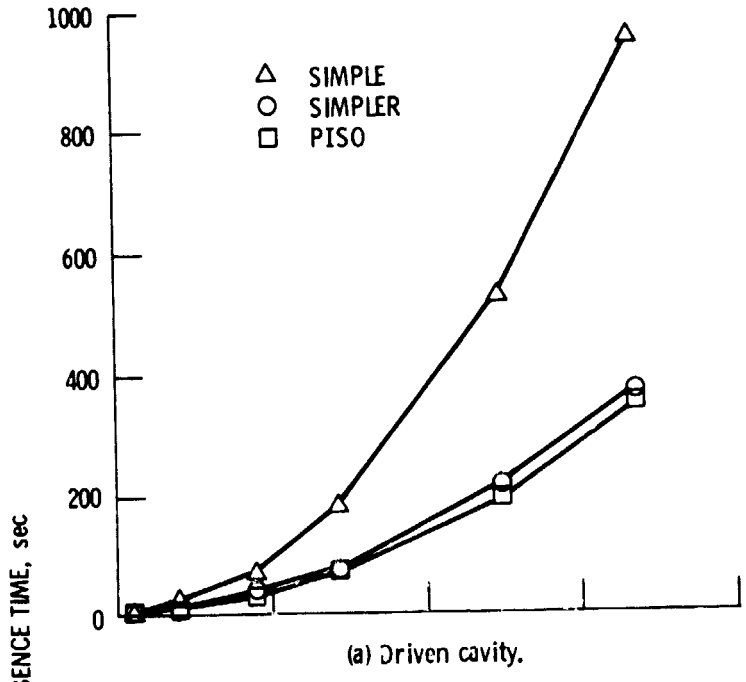
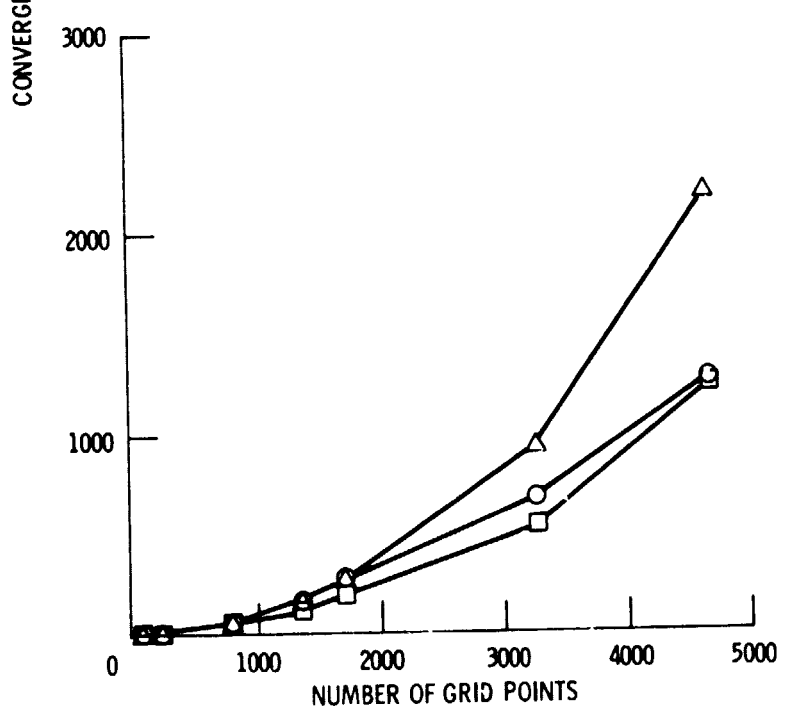



Figure 3. - Effect of relaxation factor on convergence time for laminar flow calculations. Reynolds number, 1000.




(a) Driven cavity.




(b) Flow over a step.

Figure 4. - Effect of grid size on convergence time for laminar flow calculations.

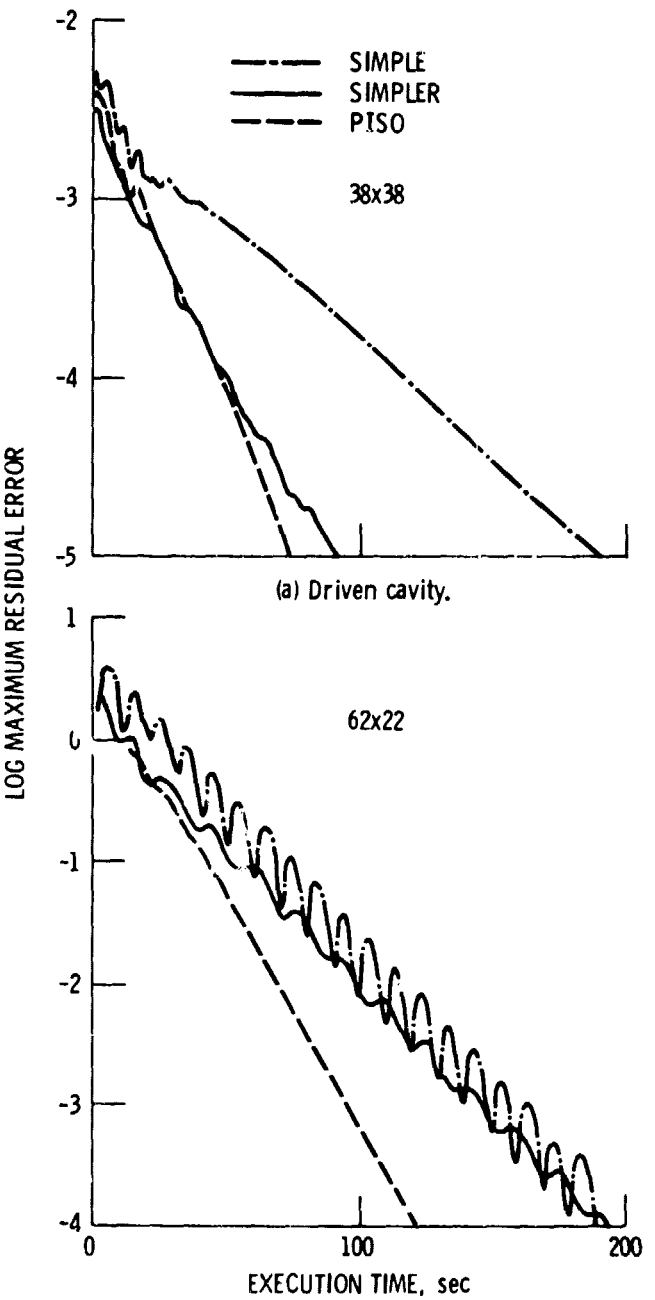
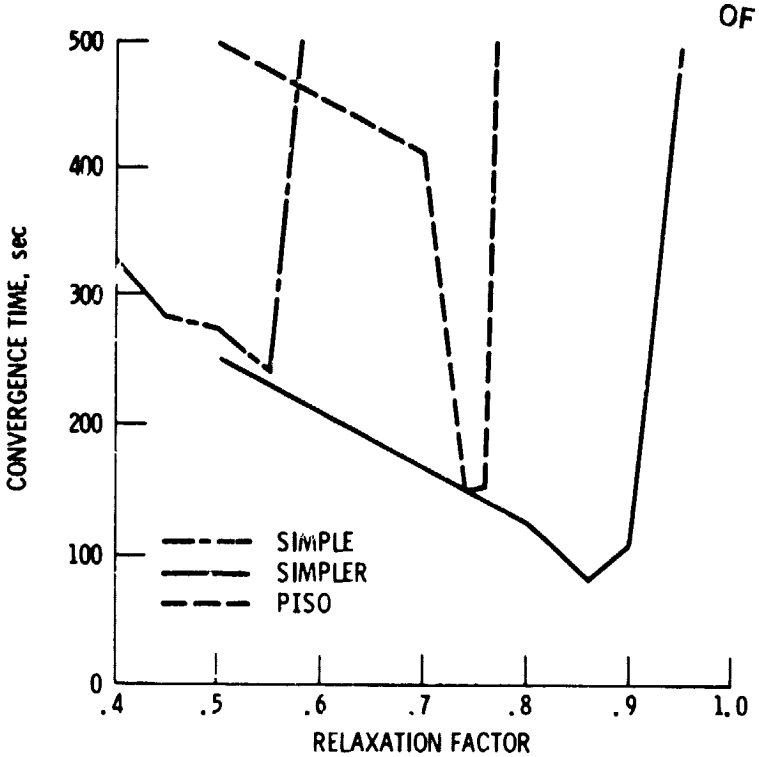
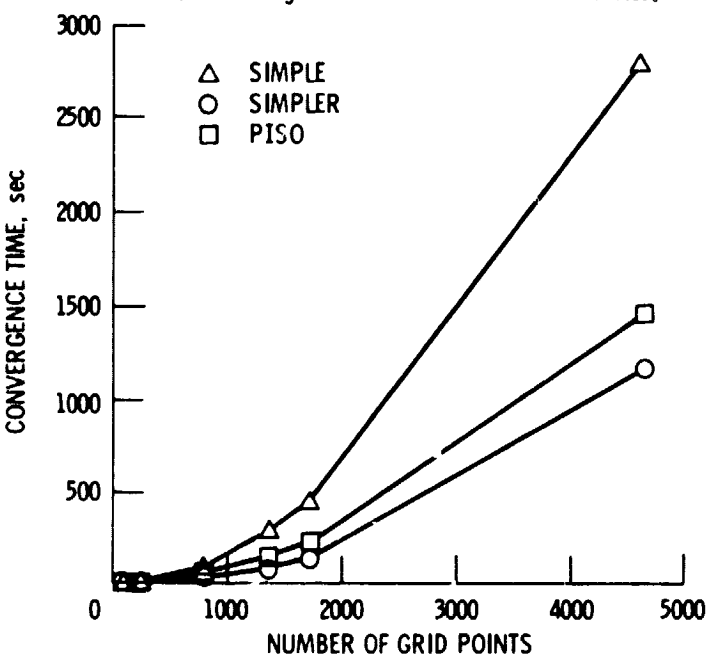



Figure 5. - Convergence rate comparison for laminar flow calculations. Reynolds number, 1000.




(a) Convergence time versus relaxation factor.




(b) Convergence time versus number of grid points.

Figure 6. - Turbulent flow calculations.

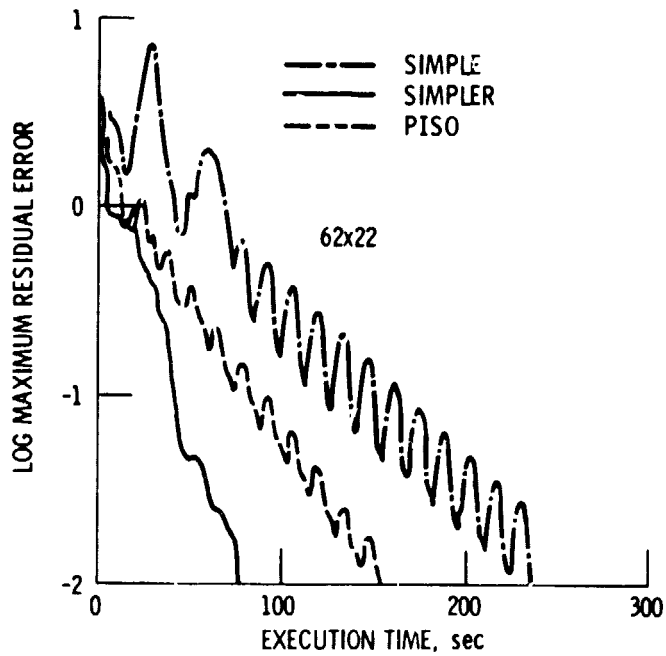



Figure 7. - Convergence rate comparison for turbulent flow calculations. Reynolds number, 30 000.