Center for Modelling of Turbulence and Transition
Research Briefs - 1991

Two Equation Modelling and the Pseudo Compressibility Technique

C. J. Steffen, Jr.

1. Motivation and Objective

The primary objective of the Center for Modelling of Turbulence and Transition (CMOTT) is to further the understanding of turbulence theory for engineering applications. One important foundation for this research is the establishment of a data base encompassing the multitude of existing models as well as newly proposed ideas. The research effort described in the next few pages is a precursor to an extended survey of two equation turbulence models in the presence of a separated shear layer.

Recently, several authors have examined the performance of two equation models in the context of the backward facing step flow. Conflicting results, however, demand that further attention is necessary to properly understand the behavior and limitations of this popular technique, especially the low Reynolds number formulations. The objective of this research is to validate an incompressible Navier Stokes code for use as a numerical test-bed. In turn, this code will be used for analyzing the performance of several two equation models.

2. Work Accomplished

To date, the validation of the incompressible code DTNS is complete. The details of this validation study are documented in reference[1]. The code is based upon the pseudo-compressibility technique and incorporates the approximate factorization scheme for time integration. Two laminar benchmark flows are used to measure the performance and implementation of the numerical methods. The classic solution of the Blasius boundary layer is used for validating the flat plate flow, while experimental data is incorporated in the validation of backward facing step flow.

An initial result for the standard high Reynolds number k-ϵ equations has also been calculated to demonstrate an initial performance level of the solution technique for the turbulence equations.

2.1 Numerical Method

The absence of a variable density in the continuity equation governing incompressible flow complicates the numerical integration procedure. One solution technique that has been well received is that of pseudo-compressibility. This idea was first put forward by Chorin[2] and enables the equations to be solved using the primitive variables. Recently, Chang and Kwak[3], Rizzi and Eriksson[4], Kwak and Chakravarthy[5], Michelassi and Shih[6], and Turkel[7] have found this method suitable for resolving incompressible flow. This particular implementation has been validated by Gorski[8–10] for several different benchmark flows.

Examine the system of equations involved in the pseudo compressibility method and notice that they differ from the incompressible Navier Stokes equations by the addition of a time dependant pressure term in the continuity equation.

$$
\begin{aligned}
\frac{1}{\beta}\frac{\partial p}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \\
\frac{\partial u}{\partial t} + \frac{\partial}{\partial x}\left(u^2 + p - \frac{1}{Re}\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(uv - \frac{1}{Re}\frac{\partial u}{\partial y}\right) &= 0 \\
\frac{\partial v}{\partial t} + \frac{\partial}{\partial x}\left(uv - \frac{1}{Re}\frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(v^2 + p - \frac{1}{Re}\frac{\partial v}{\partial y}\right) &= 0
\end{aligned}
\tag{1}
$$

Here, x and y are the independent variables and Re refers to the Reynolds number. The constant β is known as the pseudo compressibility parameter. This system is hyperbolic in nature while the incompressible flow equations are elliptic. The pseudo sound speed, $c = \sqrt{u^2 + \beta}$, is governed by the value of the parameter β, whereas the physical sound speed is infinite. Chang and Kwak [3] have shown that for $\beta > 0$ the finite speed pseudo waves vanish as time progresses and yield the proper incompressible solution at the steady state limit. It is through this parameter β that the convective and acoustic waves are decoupled, and thus convergence is governed. In choosing an optimum value for this parameter, the goal is to avoid giving the viscous effects time to react to the passing of the nonphysical transient pressure waves. Thus a lower bound on the acoustic speeds translates into a lower bound on β. However, an upper bound on β is strictly scheme dependent.

The approximate factorization is rather straight forward. Equations (2) can be rewritten as

$$
\begin{gathered}
\frac{\partial}{\partial t}(\mathbf{q}) + \frac{\partial}{\partial x}(\mathbf{f_1}+\mathbf{g_1}) + \frac{\partial}{\partial y}(\mathbf{f_2}+\mathbf{g_2}) = 0 \\
\mathbf{q} = \begin{bmatrix} \frac{p}{\beta} \\ u \\ v \end{bmatrix}, \quad \mathbf{f_1} = \begin{bmatrix} u \\ u^2 + p \\ uv \end{bmatrix}, \quad \mathbf{f_2} = \begin{bmatrix} v \\ uv \\ v^2 + p \end{bmatrix} \\
\mathbf{g_1} = \frac{-1}{Re}\begin{bmatrix} 0 \\ \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial x} \end{bmatrix}, \quad \mathbf{g_2} = \frac{-1}{Re}\begin{bmatrix} 0 \\ \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \end{bmatrix}
\end{gathered}
\tag{2}
$$

or in generalized coordinates $(\xi(x,y), \eta(x,y))$ as seen here in equation (3).

$$
\begin{gathered}
\frac{\partial}{\partial t}(\overline{\mathbf{q}}) + \frac{\partial}{\partial \xi}(\overline{\mathbf{f_1}}+\overline{\mathbf{g_1}}) + \frac{\partial}{\partial \eta}(\overline{\mathbf{f_2}}+\overline{\mathbf{g_2}}) = 0 \\
\overline{q} = \frac{\mathbf{q}}{J} \\
\overline{\mathbf{f_1}} = \frac{\xi_x \mathbf{f_1} + \xi_y \mathbf{f_2}}{J}, \quad \overline{\mathbf{f_2}} = \frac{\eta_x \mathbf{f_1} + \eta_y \mathbf{f_2}}{J} \\
\overline{\mathbf{g_1}} = \frac{\xi_x \mathbf{g_1} + \xi_y \mathbf{g_2}}{J}, \quad \overline{\mathbf{g_2}} = \frac{\eta_x \mathbf{g_1} + \eta_y \mathbf{g_2}}{J} \\
J = \frac{\partial(\xi,\eta)}{\partial(x,y)}
\end{gathered}
\tag{3}
$$

The third order accurate Chakravarthy and Osher TVD scheme is used to discretize the convective terms while central differencing is used for the viscous fluxes. Thus the approximate factorization of this method is expressed in terms of flux Jacobians A and B as

$$\begin{aligned}&\left\{\frac{\mathbf{V}}{\Delta t}+\frac{\partial^-}{\partial\xi}\mathbf{A}^+ +\frac{\partial^+}{\partial\xi}\mathbf{A}^- +\frac{\partial^-}{\partial\xi}\mathbf{A}_v\right\}\Delta\overline{\mathbf{q}}^* = \frac{\partial\left(\overline{\mathbf{f_1}}+\overline{\mathbf{g_1}}\right)}{\partial\xi}+\frac{\partial\left(\overline{\mathbf{f_2}}+\overline{\mathbf{g_2}}\right)}{\partial\eta}\\ &\left\{\frac{\mathbf{V}}{\Delta t}+\frac{\partial^-}{\partial\eta}\mathbf{B}^+ +\frac{\partial^+}{\partial\eta}\mathbf{B}^- +\frac{\partial^-}{\partial\eta}\mathbf{B}_v\right\}\Delta\overline{\mathbf{q}} = \frac{\mathbf{V}}{\Delta t}\Delta\overline{\mathbf{q}}^*\end{aligned} \tag{4}$$

where $\mathbf{V}$ is the cell volume, and $\Delta t = \frac{CFL}{1+\sqrt{J}}$ is the timestep. The matrices $\mathbf{A}_v$ and $\mathbf{B}_v$ are the viscous flux Jacobians. The interested reader is encouraged to see reference[9] for more details.

The k-ϵ equations are solved in a similar manner as that seen in equation 2. Consider the standard high Reynolds number form of the transport equations of the turbulent kinetic energy and dissipation of turbulent genetic energy as in equation (5):

$$\begin{gathered}\frac{\partial}{\partial t}(\mathbf{q}^t)+\frac{\partial}{\partial x}(\mathbf{f}_1^t+\mathbf{g}_1^t)+\frac{\partial}{\partial y}(\mathbf{f}_2^t+\mathbf{g}_2^t)=\mathbf{S}^t\\ \mathbf{q}^t=\begin{bmatrix}k\\ \epsilon\end{bmatrix},\ \mathbf{f}_1^t=\begin{bmatrix}uk\\ u\epsilon\end{bmatrix},\ \mathbf{f}_2^t=\begin{bmatrix}vk\\ v\epsilon\end{bmatrix}\\ \mathbf{g}_1^t=-\frac{1}{Re}\begin{bmatrix}\nu_k\frac{\partial k}{\partial x}\\ \nu_\epsilon\frac{\partial\epsilon}{\partial x}\end{bmatrix},\ \mathbf{g}_2^t=-\frac{1}{Re}\begin{bmatrix}\nu_k\frac{\partial k}{\partial y}\\ \nu_\epsilon\frac{\partial\epsilon}{\partial y}\end{bmatrix},\ \mathbf{S}^t=\frac{1}{Re}\begin{bmatrix}P-\epsilon Re\\ C_1\frac{\epsilon}{k}P-C_2\frac{\epsilon^2}{k}Re\end{bmatrix}\end{gathered} \tag{5}$$

with

$$\begin{aligned}\nu_k&=\nu+\frac{\nu_t}{\sigma_k}\\ \nu_\epsilon&=\nu+\frac{\nu_t}{\sigma_\epsilon}\end{aligned} \tag{6}$$

and the production of turbulent genetic energy defined as shown below.

$$P=\nu_t\left[2(u_x^2+v_y^2)+(u_y+v_x)^2\right] \tag{7}$$

Again, we can translate these equations in a manner similar to that of equation (3) and create the following system for generalized coordinates:

$$\frac{\partial}{\partial t}\left(\overline{\mathbf{q}^t}\right)+\frac{\partial}{\partial\xi}\left(\overline{\mathbf{f}_1^t}+\overline{\mathbf{g}_1^t}\right)+\frac{\partial}{\partial\eta}\left(\overline{\mathbf{f}_2^t}+\overline{\mathbf{g}_2^t}\right)=\overline{\mathbf{S}^t} \tag{8}$$

This can be solved via approximate factorization as shown here in equation 9.

$$\begin{aligned}&\left\{\frac{\mathbf{V}}{\Delta t}+\frac{\partial^-}{\partial\xi}\mathbf{A}^{t+}+\frac{\partial^+}{\partial\xi}\mathbf{A}^{t-}+\frac{\partial^-}{\partial\xi}\mathbf{A}_v^t-\alpha_\xi\mathbf{H}^t\right\}\Delta\overline{\mathbf{q}^t}^* = \frac{\partial\left(\overline{\mathbf{f}_1^t}+\overline{\mathbf{g}_1^t}\right)}{\partial\xi}+\frac{\partial\left(\overline{\mathbf{f}_2^t}+\overline{\mathbf{g}_2^t}\right)}{\partial\eta}+\overline{\mathbf{S}^t}\\ &\left\{\frac{\mathbf{V}}{\Delta t}+\frac{\partial^-}{\partial\eta}\mathbf{B}^+ +\frac{\partial^+}{\partial\eta}\mathbf{B}^- +\frac{\partial^-}{\partial\eta}\mathbf{B}_v-\alpha_\eta\mathbf{H}^t\right\}\Delta\overline{\mathbf{q}^t} = \frac{\mathbf{V}}{\Delta t}\Delta\overline{\mathbf{q}^t}^*\end{aligned} \tag{9}$$

Again, the scheme is written in terms of the turbulence flux Jacobians and the approximate source Jacobian, $\mathbf{H^t}$. The coefficients α_ξ and α_η are provided to enhance the diagonal dominance of the implicit method. While the convective flux terms are linear, the linearization of the source terms is not straight forward. Furthermore, the low Reynolds number formulation of equation 5 includes other nonlinear correction terms in the source vector $\mathbf{S^t}$. The interested reader is encouraged to see the recent paper by Michelassi and Shih[6].

2.2 Discussion

Overall the performance for laminar flow documented in reference[1] is very encouraging. The flat plate results shown agree with the theoretical Blasius solution for several different grid configurations. The laminar back facing step flow was also well resolved both in terms of the primary recirculation zone reattachment length, and corresponding velocity profiles as seen in figures 1 and 2. However, performance of the two dimensional flow code drops off dramatically when the Reynolds number corresponding to the onset of 3D flow is exceeded. Armaly et al [11] observed this phenomenon occur at a Reynolds number of 400, based upon the hydraulic diameter of the inlet channel. Again, the numerical results bear this finding out.

Figure 1 Reattachment length predictions—comparison with experiment [11]. x_1, x_2, and x_3 are the primary reattachment, secondary detachment, and secondary reattachment lengths, respectively.

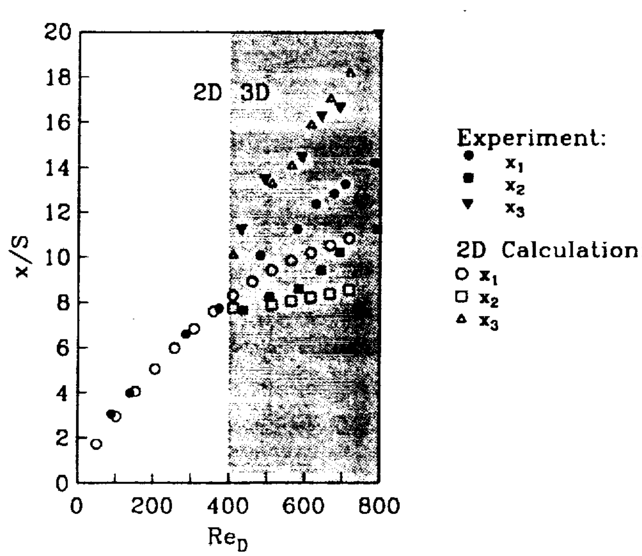
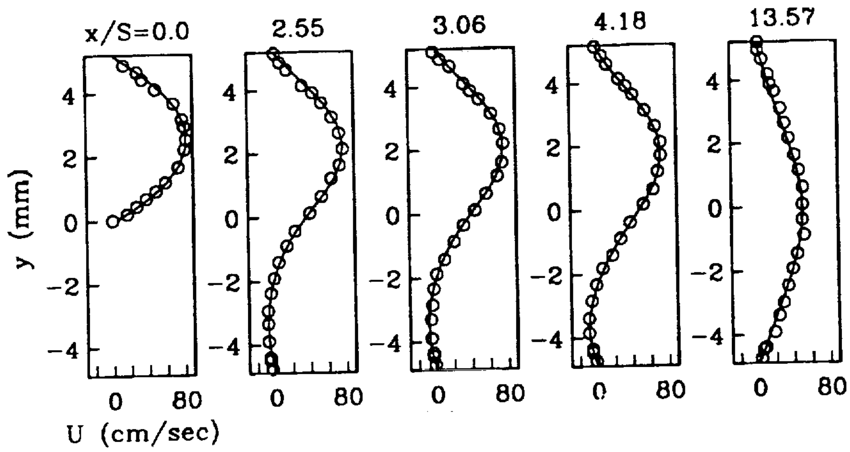


Figure 2 Re_D=389 velocity profiles—comparison with experiment [11].



The current thrust of effort is in establishing a comparison between the standard high Reynolds number form k-ϵ model and a version which incorporates the low Reynolds number corrections, such as that of Jones and Launder[12]. This model is particularly convenient in that the damping functions are independent of the parameter, y^+. Thus, multiple walls bounding the computational domain do not need special attention. An initial calculation using the high Reynolds number approach agrees with the published results of Speziale and Thangam[13]. However, the Jones and Launder model is proving to be quite challenging.

3. Future Plans

Eventually, several two equation models will be examined for their behavior in the back facing step flow. The following is a partial list of the two equation models of primary interest:

1. Jones and Launder
2. Chien
3. Yang and Shih
4. Shih and Lumley

The current research effort will result in an increased understanding of the need for proper modelling of the near wall region. However, the increase in accuracy maybe offset by an unwarranted increase in effort. Further numerical experiments are necessary before any conclusions can be drawn.

4. References

1 Steffen, Jr., C. J., "An Investigation of DTNS2D for Use as an Incompressible Turbulence Modelling Test-bed," NASA Technical Memorandum 105593 (1992).

[2] Chorin, A. J., "A Numerical Method for Solving Incompressible Viscous Flow Problems," J. Comput. Phys., **2**, 12–26 (1967).

[3] Chang, J. L. C. and Kwak, D., "On the Method of Pseudo Compressibility for Numerically Solving Incompressible Flows," AIAA paper 84–0252 (1984).

[4] Rizzi, A. and Eriksson, L. E., "Computation of Inviscid Incompressible Flow with Rotation," J. Fluid Mech., **153**, 275–312 (1985).

[5] Kwak, D., Chang, J. L. C., Shanks, S. P., and Chakravarthy, S. R., "A Three Dimensional Incompressible Navier-Stokes Flow Solver Using Primitive Variables," AIAA J., **24**, 390–396 (1986).

[6] Michelassi, V. and T.-H. Shih, , "Elliptic Flow Computation by Low Reynolds Number Two Equation Turbulence Models," NASA Technical Memorandum 105376 (1991).

[7] Turkel, E., "Preconditioned Methods for Solving the Incompressible and Low Speed Compressible Equations," J. Comput. Phys., **72**, 277–298 (1987).

[8] Gorski, J. J., "Incompressible Cascade Calculations Using an Upwind Differenced TVD Scheme," *Advances and Applications in Computational Fluid Dynamics*, **66** of ASME-FED, edited by O. Baysal, 61–69 (1988).

[9] Gorski, J. J., "TVD Solutions of the Incompressible Navier-Stokes Equations With an Implicit Multigrid Scheme," *Proceedings of the AIAA/ASME/SIAM/APS 1st National Fluid Dynamics Congress*, **1**, 394–401 (1988).

[10] Gorski, J. J., "Solutions of the Incompressible Navier-Stokes Equations Using an Upwind Differenced TVD Scheme," *Proceedings of the 11th International Conference on Numerical Methods in Fluid Dynamics*, 278–282 (1988).

[11] Armaly, B. F., Durst, F., Pereira, J. C. F., and Schönung, B., "Experimental and Theoretical Investigation of Backward-Facing Step Flow," J. Fluid Mech., **127**, 473–496 (1983).

[12] Jones, W. P. and Launder, B. E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," Int. J. Heat Mass Transf., **15**, 301–314 (1972).

[13] Thangam, S. and Speziale, C. G., "Turbulent Separated Flow Past a Backward Facing Step: a Critical Evaluation of Two-Equation Turbulence Models," NASA Contractor Report 187532 ICASE Report No. 91–23 (1991).