

NASA Technical Memorandum 105593  
ICOMP-92-07; CMOTT-92-04

84447  
p. 18

# An Investigation of DTNS2D for Use as an Incompressible Turbulence Modelling Test-bed

(NASA-TM-105593) AN INVESTIGATION OF DTNS2D  
FOR USE AS AN INCOMPRESSIBLE TURBULENCE  
MODELLING TEST-BED (NASA) 18 p CSCL 12A

N92-22230

Unclas

G3/64 0084447

Christopher J. Steffen, Jr.  
*Lewis Research Center*  
*Cleveland, Ohio*  
*and Institute for Computational Mechanics in Propulsion*  
*and Center for Modelling of Turbulence and Transition*

March 1992





# **An Investigation of DTNS2D for Use as an Incompressible Turbulence Modelling Test-bed**

Christopher J. Steffen  
NASA Lewis Research Center  
Cleveland, Ohio 44135  
and Institute for Computational Mechanics and Propulsion  
and Center for Modelling of Turbulence and Transition

## **Abstract**

This paper documents an investigation of a two dimensional, incompressible Navier-Stokes solver for use as a test-bed for turbulence modelling. DTNS2D is the code under consideration for use at the Center for Modelling of Turbulence and Transition (CMOTT). This code was created by Gorski at the David Taylor Research Center and incorporates the pseudo compressibility method. Two laminar benchmark flows are used to measure the performance and implementation of the method. The classical 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. Velocity profiles, convergence histories, and reattachment lengths are used to quantify these calculations. The organization and adaptability of the code are also examined in light of the role as a numerical test-bed.

## **Introduction**

The Center for Modeling of Turbulence and Transition (CMOTT) at NASA Lewis has the mission to improve understanding of turbulence modeling applications. One vital aspect of this improved understanding results from extensive testing and validation. Therefore a series of numerical testbeds must be chosen with which to conduct these turbulence modelling surveys. This paper documents an investigation of the two dimensional flow code DTNS2D created by Gorski at the David Taylor Research Center. The code utilizes the established pseudo compressibility technique to solve the incompressible Navier-Stokes equations.

The approach is to utilize laminar analysis to validate the implementation of established numerical schemes, and turbulent analysis to investigate the behavior of new turbulence modelling theories. However, the interaction of models and governing equations demands that a variety of numerical integration methods be utilized to draw any proper conclusions from the testing of a broad range of turbulence theories. Thus a candidate code must prove to be versatile in this respect.

Two test cases will be used to validate this code in house. First, the classic solution of Blasius for laminar flat plate flow will be analyzed. The role of pseudo compressibility as a convergence parameter will be studied as well as high Reynolds number flow and grid stretching in the cross-flow direction. In addition, the backward facing step problem will also be examined. The 2D laminar flow will be examined for accuracy of the reattachment length and velocity profiles, following the format of Armaly et al. Although that experiment concentrates primarily upon transitional flow regime, there is sufficient data in the laminar region to draw some conclusions as far as accuracy is concerned.

## Method of Pseudo Compressibility

The absence of a variable density term in the continuity equation governing incompressible flow complicates the numerical integration procedure. However, several techniques are available for solving the system of incompressible equations numerically. One approach is the stream function-vorticity formulation. While the pressure is eliminated by this formulation, the extension to three dimensions is not obvious. It is preferable to work with the primitive variables, pressure ( $p$ ) and the velocity components ( $u$  and  $v$ ), as in the pressure correction scheme of Harlow and Welch [1]. However, this scheme involves a decoupling of the pressure and velocity fields which adversely affects the convergence to steady state. Thus a three dimensional calculation is very expensive. A third approach applies the efficient time marching strategies of compressible flow to the solution procedure. Straightforward computation is realizable for two and three dimensional steady and unsteady incompressible flows with the pseudo compressibility technique.<sup>1</sup>

The DTNS2D code, created for the solution of two dimensional, incompressible flow, is based upon the method of pseudo compressibility. This idea was first put forward by Chorin [2] and presented a novel approach to solving this system of equations using primitive variables. Steger and Kutler [3] adopted a similar method, applying the implicit approximate factorization scheme of Beam and Warming [4]. More recently Chang and Kwak[5], Kwak and Chakravarthy [6], Soh [7], Rizzi and Eriksson [8], and others have found this method suitable for solving incompressible flow. This particular implementation has been validated by Gorski [9–11], for several benchmark flow situations.

Examine the system of equations solved in the pseudo compressibility method and notice that they differ from the steady state incompressible flow equations by the addition of a time

---

<sup>1</sup> Only the steady, two dimensional formulation is discussed in this paper.

dependent term in the continuity equation.

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

Here,  $x$  and  $y$  are the independent variables and  $Re$  refers to the Reynolds number. 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  $\beta$ , whereas the physical sound speed is infinite. Chang and Kwak [5] 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  $\beta$  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  $\beta$ . However, an upper bound on  $\beta$  is strictly scheme dependent.

Several authors have attempted to give a more specific guideline as to the optimal choice of this parameter. Chang and Kwak [5] have suggested a formulation based upon a comparison of the time scales involved in the propagation of convective and acoustic waves. This results in a lower bound on  $\beta$  which is a function of the Reynolds number. Soh [7] has given an argument based upon the ratio of largest to smallest eigenvalues. This generates a value of  $\beta$  which is three times the magnitude of an appropriate reference velocity. Choi and Merkle [12] have found maximum convergence speed in two dimensional flow simply corresponds to  $\beta = 1$ . While all agree that the optimum value is of order one for external flow, no one rule seems generally more successful than the others. In a later section, it will be shown that  $\beta = 1$  does not always converge for external flow, and in fact the optimal value is often an order of magnitude less than one. One other interesting possibility for the selection of  $\beta$  was presented by Rizzi and Eriksson [8]. The value was set "... in the spirit of local timestep scaling ..." at a value proportional to the local velocity magnitude squared, with a prescribed minimum limit.

The boundary conditions for this system of equations can be evaluated from analyzing the direction of the pseudo characteristics,  $\lambda = u, u \pm \sqrt{u^2 + \beta}$ . For an inflow condition, there are two right running characteristics which propagate into the domain; at outflow there is one left running characteristic which likewise propagates into the domain. Thus at inflow we prescribe two physical ( $u$  and  $v$ ) and one numerical ( $p$ ) boundary conditions, and just the opposite at outflow.

## Flat Plate Flow

The similarity solution of Blasius for laminar flow over a flat plate is familiar to all students of fluid mechanics. It states that the streamwise and transverse components of velocity can be reduced to similarity profiles if scaled with the parameter  $\eta = y\sqrt{\frac{Re_x}{x}}$ , where  $\eta$  is a nondimensional coordinate and  $Re_x$  is the Reynolds number based upon distance from the leading edge,  $x$ . This solution is derived from the boundary layer equations and is seen to accurately predict the velocity distributions measured in various experiments. Here, of course, we are solving the incompressible Navier-Stokes equations.

One important aspect of this boundary layer derivation is that the velocity of potential flow is constant, and therefore  $\frac{dp}{dx} = 0$ . Our numerical solution is complicated by this pressure condition. Given a limited number of grid cells, we can either include the leading edge in the numerical domain and neglect the flow influenced by the stagnation pressure region (edge capturing, figure 1(a)), or start the domain downstream of the leading edge by prescribing a Blasius inflow condition (edge bypass, figure 1(b)). While both strategies are successful, for all cases where  $Re_L < 10^6$  the former approach is chosen due to the more general nature of the freestream velocity inlet conditions. At issue is the downstream influence of the stagnation flow. If, however, one is willing to accept an incomplete resolution of the stagnation region, then the boundary layer flow downstream of this can be accurately resolved with a very coarse grid.

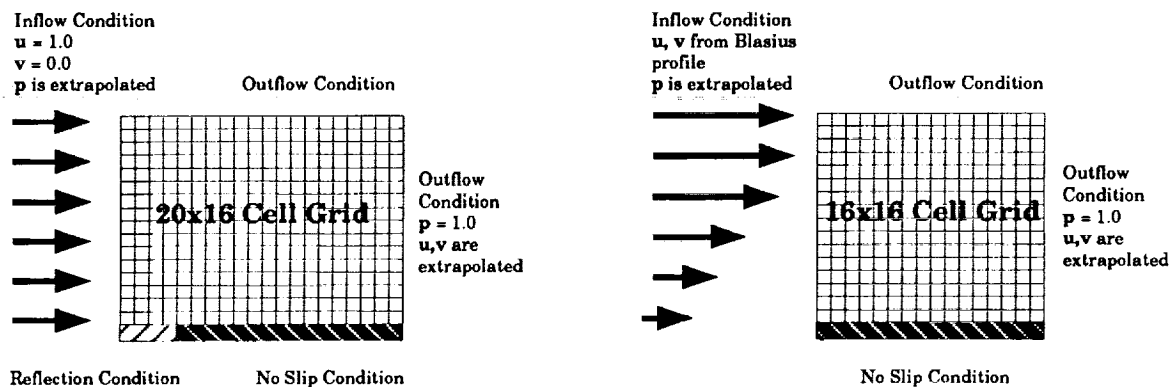


Figure 1 Flat plate flow grids—(a) edge capturing and (b) edge bypass.

The numerical domain is set up such that the leading edge is at  $x = 0$ . The boundary layer thickness  $\delta_L = \delta(Re_L)$  is used to scale the  $y$ -direction. For instance, an unstretched  $(16 \times 16)$  cell Cartesian grid is generated with dimensions  $L \times \delta_L$ . In addition, a  $(4 \times 16)$  cell grid is added upstream of the leading edge. This allows the influence of stagnation to propagate forward enough so that freestream velocity conditions can be applied. The boundary conditions can thus be summarized in figure 1.

The velocity profiles shown in figure 2 correspond to a calculation of flow along a plate where  $Re_L = 10^5$ . Figure 2(a) demonstrates that the streamwise component is well resolved on this  $(20 \times 16)$  cell grid. Notice that the data of figure 2(b), which corresponds to a  $(40 \times 32)$  cell grid, does not show any substantial improvement in the solution quality. However, the transverse component of velocity reveals a different story. The data of figure 3 corresponds to the same calculations described in figure 2. These velocity component profiles indicate that the  $(20 \times 16)$  cell calculation is not fully resolved. This is not surprising if one remembers that the Blasius theory provides a profile which scales as a product of the square root of Reynolds number:

$$\frac{v}{U_\infty} = \frac{1}{2} \sqrt{\frac{1}{Re_x}} [\eta f'(\eta) - f(\eta)]$$

$$f(\eta) = \frac{\psi}{\sqrt{\nu x U_\infty}}$$

where  $f(\eta)$  is the dimensionless stream function. Thus  $\frac{v}{u} \propto \frac{1}{\sqrt{Re_x}}$  and is more sensitive to grid resolution. The behavior of the transverse profile at  $Re_x = 100,000$  is still not fully resolved even on the finer grid, though. The constant pressure boundary conditions imposed at the outflow ( $x=L$ ) and freestream ( $\eta=5.5$ ) are believed to contribute to this condition. The zero pressure gradient which has been imposed at these boundaries is not strictly valid. Further refinement and extension of the numerical domain has been shown to alleviate this problem. However, in keeping with the spirit of this calculation, the accuracy of the velocity profiles demonstrated above in figures 2(b) and 3(b) is sufficient.

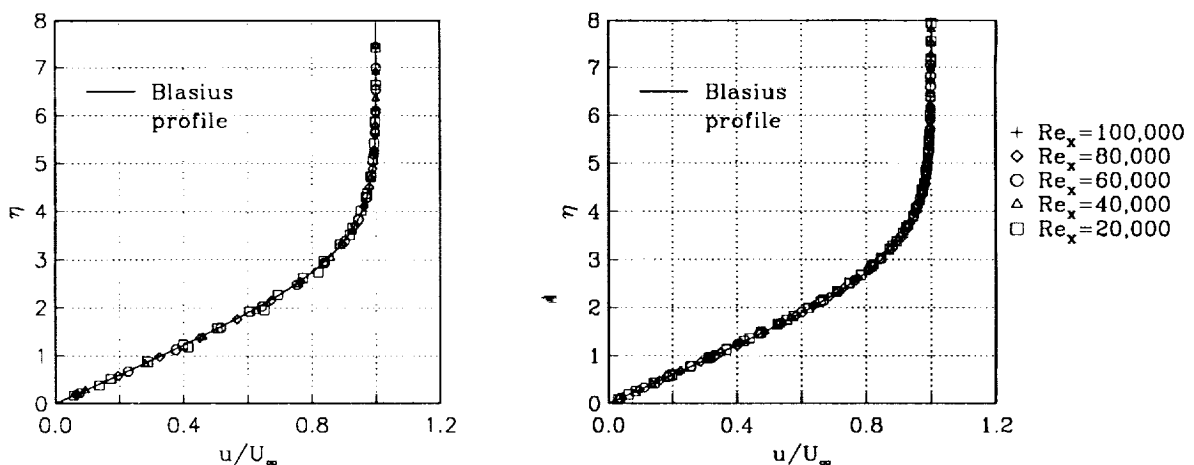


Figure 2 Streamwise velocity profiles—(a)  $(20 \times 16)$  cell grid and (b)  $(40 \times 32)$  cell grid.

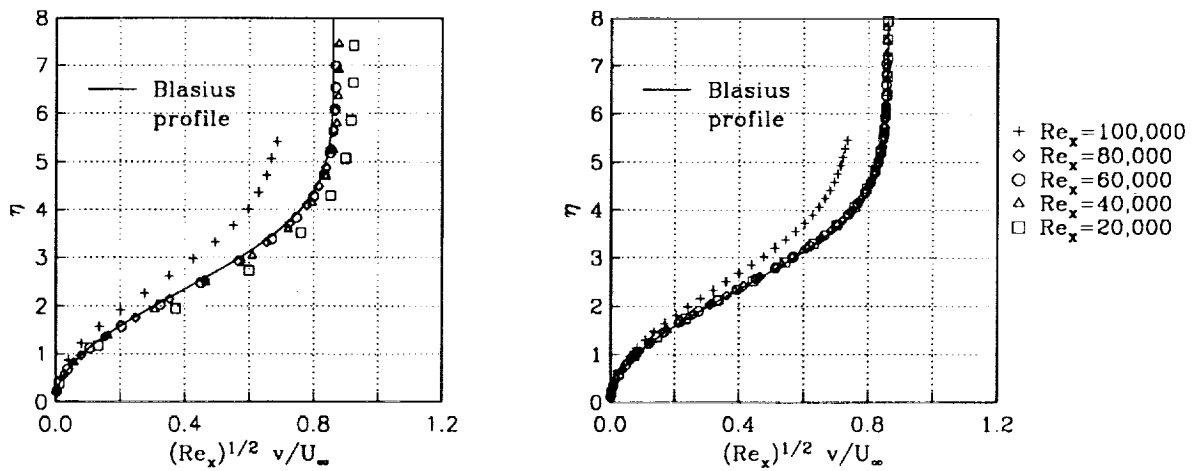


Figure 3 Transverse velocity profiles—(a) (20x16) cell grid and (b) (40x32) cell grid.

The convergence of this flat plate flow from a uniform initial condition can be seen in figure 4. Several values for the pseudo compressibility parameter at a CFL number of 13 are seen in figure 4(a). Various CFL number values are also shown at  $\beta = 0.15$  in figure 4(b). As noted earlier, a pseudo compressibility parameter of unity does not insure convergence of a two dimensional laminar flow field. However, optimum value does reduce the root mean squared value of  $(\Delta_t p/\text{cell volume})$  by 10 orders of magnitude in 900 iterations.

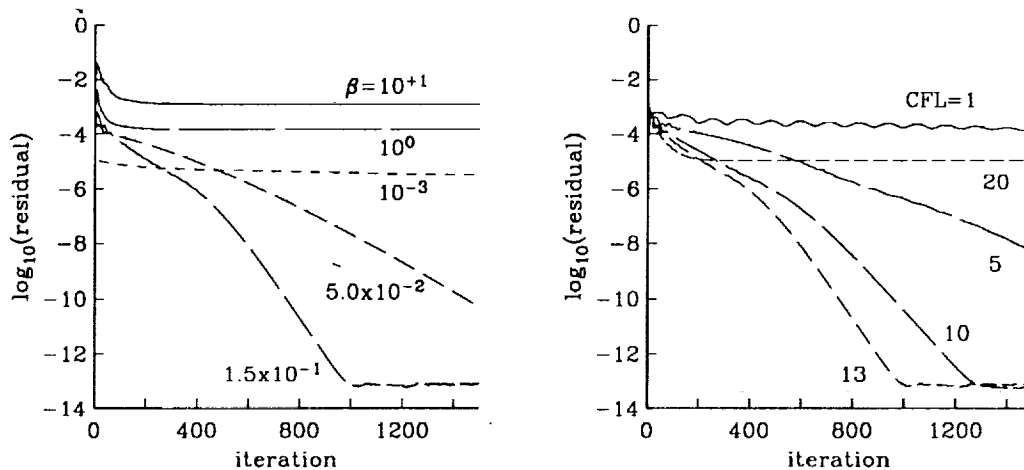


Figure 4 Convergence histories—(a) CFL=13 with various pseudo compressibility parameter ( $\beta$ ) values and (b)  $\beta=0.15$  with various CFL number values.

In a turbulent calculation, high Reynolds number flows with near wall modeling are often encountered. Resolving the drastic variation in length scales present in such a flow



demands some form of grid stretching. Thus response of the DTNS2D code to a high Reynolds number flow over a stretched grid is very important. Figure 5 shows the results of  $Re_L = 1.01 \times 10^7$  over a  $32 \times 32$  cell cartesian grid. This grid is stretched from an aspect ratio of  $9.9 \times 10^3$  at the wall to a value of  $7.2 \times 10^{-1}$  at the freestream boundary, using the Roberts [13] transformation:

$$x = \bar{x}L$$

$$y = \delta_L \frac{(\alpha + 1) - (\alpha - 1) \left\{ \left[ \frac{(\alpha + 1)}{(\alpha - 1)} \right]^{1 - \bar{y}} \right\}}{\left[ \frac{(\alpha + 1)}{(\alpha - 1)} \right]^{1 - \bar{y}} + 1}$$

where the parameter  $\alpha$  governs stretching and  $(\bar{x}, \bar{y})$  are normalized coordinates. The edge bypass strategy was used because a prohibitively large value of  $Re_{cell}$  was demanded by the edge capturing method. The CFL number for this calculation is 100. Figure 5 illustrates that both accuracy and efficiency can be maintained under these conditions.

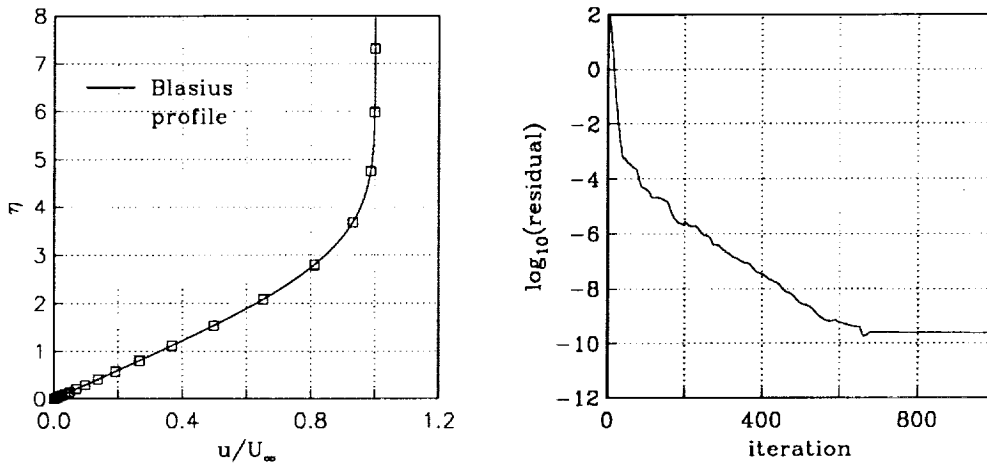


Figure 5  $Re_x=1.01 \times 10^7$  flow over a stretched grid—aspect ratio of 9800 at wall (a) streamwise velocity profile and (b) convergence history.

## Flow Over a Back Facing Step

Another excellent benchmark flow geometry is the back facing step. The presence of recirculation and developing internal flow make this a challenging flow to simulate with DTNS2D. There is a large body of work in this area for both laminar and turbulent flows. The interested reader is referred to the following references: [14–17].

The particular experiment used to validate this code is that of Armaly et al. [18] This experiment was chosen mainly because it involves a detailed examination of the flow from

$70 \leq Re_D \leq 8000$ , where  $Re_D = \bar{U}_{inlet} \frac{2h}{\nu}$  is a function of the mean inlet velocity measured two channel heights upstream of the step, and the hydraulic diameter (see figure 6). This Reynolds number region encompasses the two dimension laminar and turbulent regions as well as the highly three dimensional flow between these extremes. Although we have only examined the performance of DTNS2D over a small portion of this Reynolds number regime,  $50 \leq Re_D \leq 1095$ , future 2D and 3D flow computations can utilize the same extensive data set for validation. The experimental velocity measurements were made with a laser-Doppler anemometer. The tunnel was constructed so that the inlet flow is fully developed two dimensional channel flow over the whole Reynolds number range. The exit is located over one hundred step heights downstream of the step plane and ensures fully redeveloped channel flow at the exit for the particular Reynolds number span used in this validation. This is particularly convenient for numerical simulation because it creates a very general boundary condition eliminating the need for experimental inlet velocity profiles.

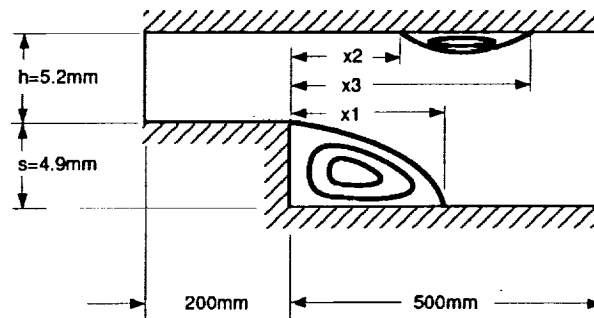
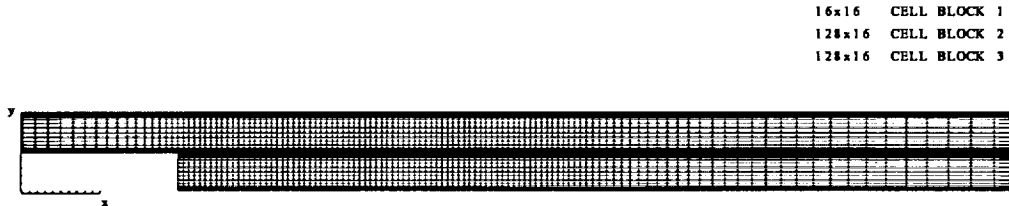


Figure 6 Back facing step—experimental setup.

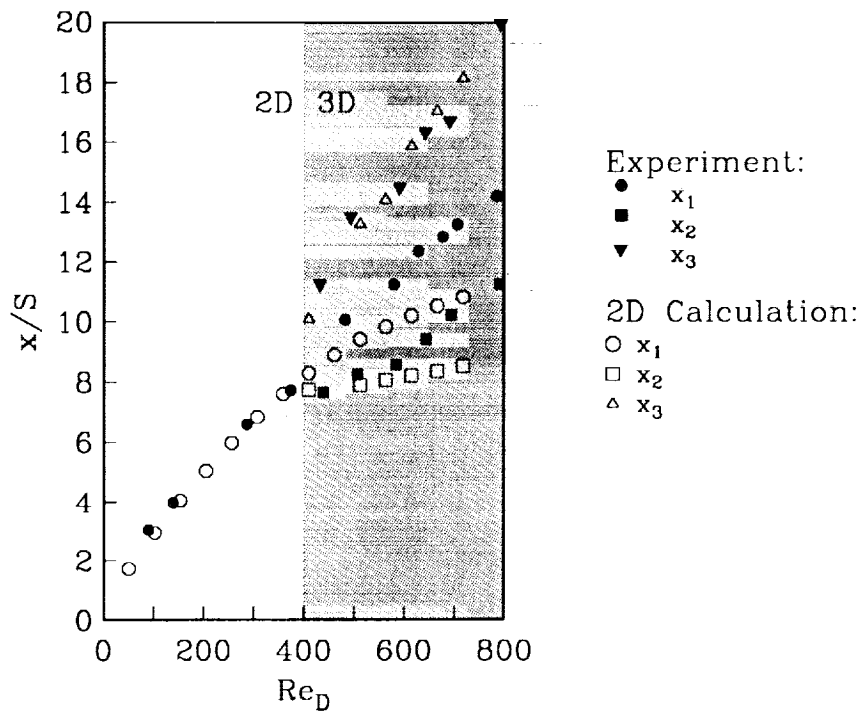
Preliminary analysis has shown that the computational domain can be significantly smaller than the actual tunnel dimensions if the fully developed flow assumptions are utilized. An inlet channel of length equal to two hydraulic diameters is appropriate for applying the parabolic flow profile. The channel downstream of the step was truncated to four times the length of the primary recirculation region  $x_1$ , allowing the Von Neumann boundary condition for the velocity components at the exit. Grid stretching was used in both the  $x$  and  $y$  directions, concentrating on both the near wall regions and the primary recirculation zone. The mesh density was held constant throughout this recirculation zone. Because the length of this recirculation zone varies from nearly zero to sixteen step heights in the Reynolds number range we are investigating, two grids were generated. For flow at  $50 \leq Re_D \leq 500$  a 4352 cell grid was generated (figure 7) and for  $500 \leq Re_D \leq 1095$  a 7424 cell grid was

generated. The mesh density in the recirculation zones is the same in both, and has been shown to generate grid independent results when compared to a 13312 cell mesh.



**Figure 7** Backward facing step grid—4352 cell version for flow at  $Re_D \leq 500$ . Note that the grid downstream of the step has been truncated.

The experimental data clearly defines the Reynolds number region of two dimensional laminar flow as  $Re_D \leq 400$ . Qualitatively, this corresponds to the appearance of a second recirculation zone along the wall opposite the step. Figure 8 demonstrates the accurate prediction of the primary reattachment length,  $x_1$  within this region. Additionally, the Reynolds number corresponding to the onset of secondary recirculation is predicted. Armaly et al., indicate that spanwise variation in the primary reattachment location at Reynolds numbers greater than 400 is evidence of an onset of three dimensional flow. Thus one would expect that the primary reattachment  $x_1$  and secondary detachment  $x_2$  locations do not accurately predict the experimental findings, taken at midspan, for  $Re_D > 400$ . However, the coupling of the primary and secondary reattachment locations exhibited in the experiment ( $x_2 < x_1 < x_3$ ) is preserved. Surprisingly, the secondary reattachment point,  $x_3$ , follows the predicted experimental results quite well. It is interesting to note that the reattachment point furthest from the step is accurately predicted up to a Reynolds number of 800. This trend is also observed in the data of Chen [19]. Perhaps a pocket of strongly three dimensional flow initially exists only in the vicinity of the secondary detachment and primary reattachment locations. Unfortunately, no spanwise measurements of secondary reattachment location are available in this Reynolds number region. Further investigation using a three dimensional version of DTNS2D is beyond the scope of this paper.



**Figure 8** Reattachment length predictions—comparison with experiment [18].  $x_1$ ,  $x_2$ , and  $x_3$  are the primary reattachment, secondary detachment, and secondary reattachment lengths, respectively.

Similar conclusions can be drawn from the velocity profile data at three different Reynolds numbers. Figure 9 displays the profile comparisons at five different locations downstream of the step for a Reynolds number of 100. The calculation is well within the two dimensional flow regime and the predictions match the experimental results very well. Similar comments can be made about the velocity profiles at a Reynolds number of 389, shown in figure 10. The results for the  $Re_D = 1095$  case seen in figure 11 are slightly different, as we would expect. However, the overall shape of the primary and secondary recirculation regions are clearly visible.

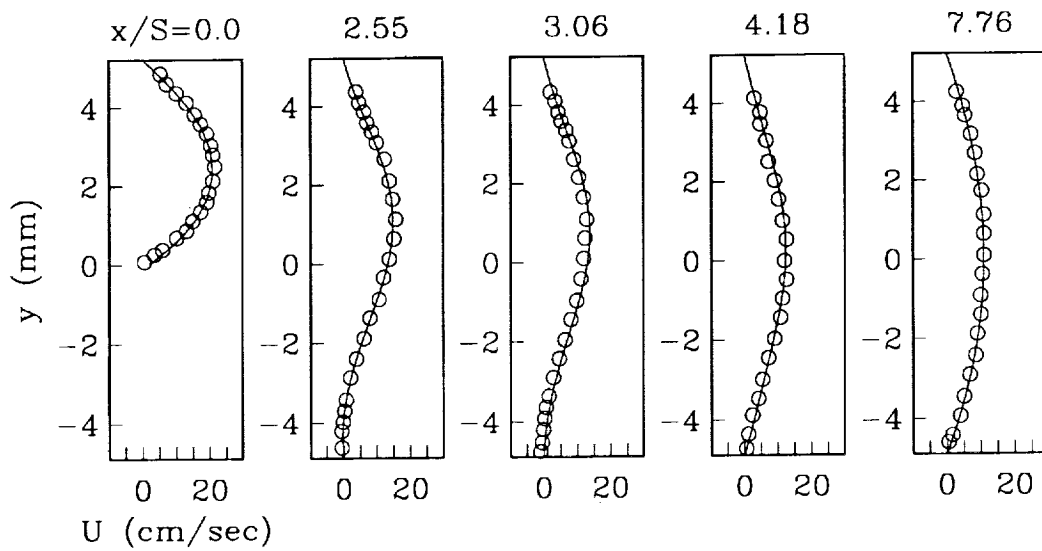


Figure 9  $Re_D=100$  velocity profiles—comparison with experiment [18].

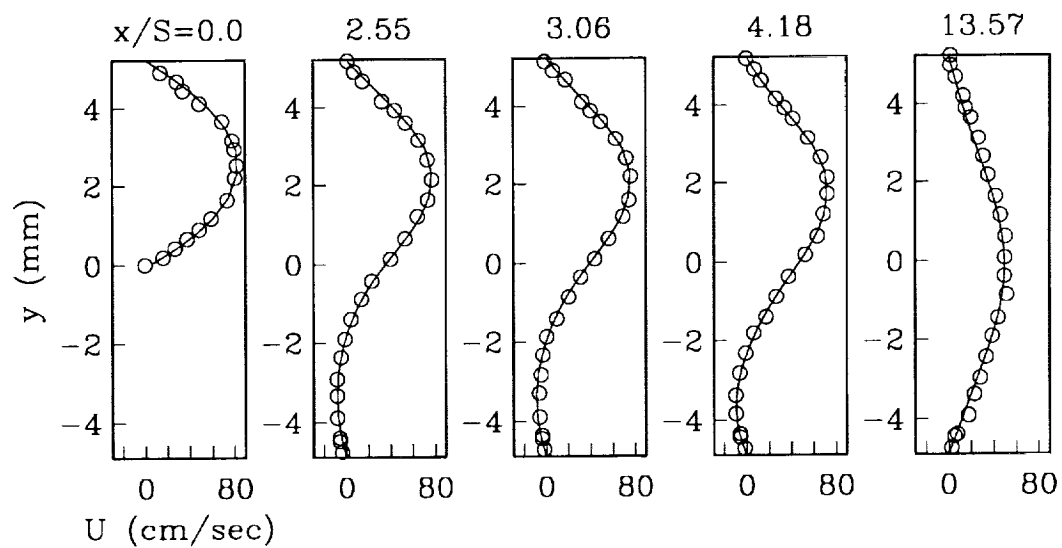


Figure 10  $Re_D=389$  velocity profiles—comparison with experiment [18].

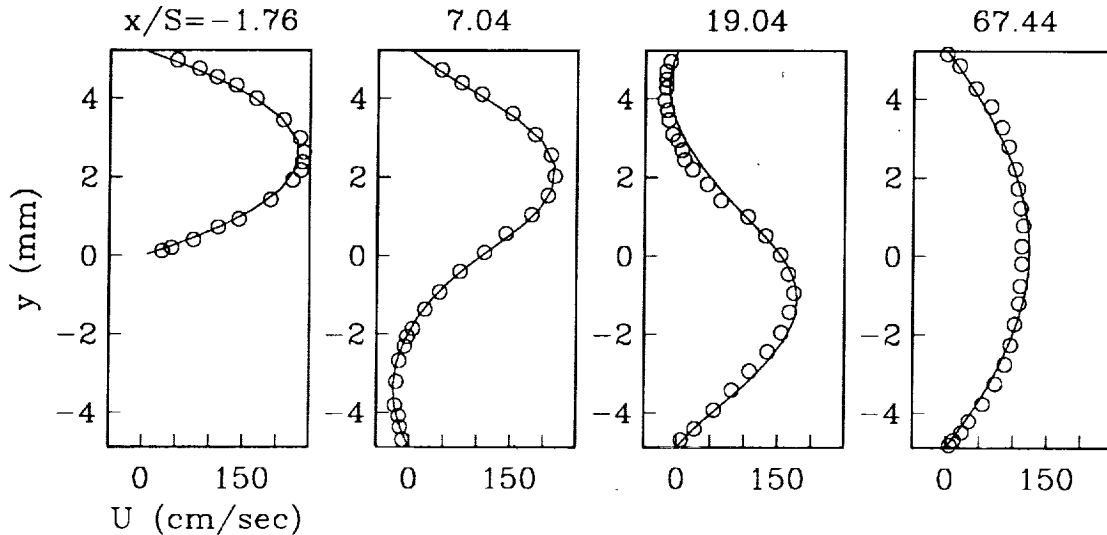
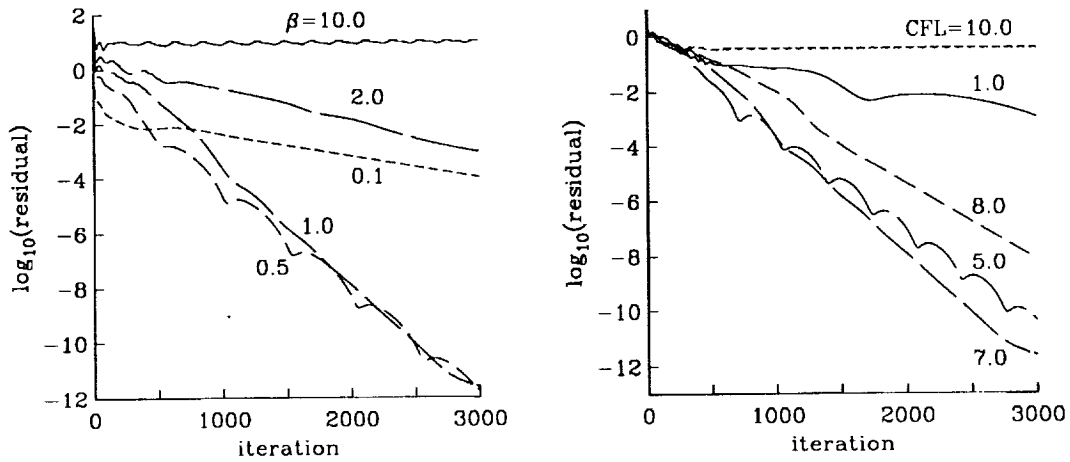


Figure 11  $Re_D=1095$  velocity profiles—comparison with experiment [18].

The performance of the DTNS2D code in the two dimensional backward facing step flow is demonstrated in the convergence histories of figure 12. The calculation is of the  $Re_D=389$  flow, similar to that shown above in figure 10. Figure 12(a) shows that an optimum value of the pseudo compressibility parameter has a value around unity. An order of magnitude more or less in value severely restricts the convergence properties of the scheme. This behavior was typical of that seen throughout the two dimensional laminar flow range, with the optimum consistently valued around unity. The maximum CFL number can be seen from figure 12(b) to be seven for this case. This CFL limit was seen to gradually decrease in value as the Reynolds number increased, which is to be expected. For example, at a Reynolds number of 100, the CFL limit was observed to be twenty. The approximate factorization scheme does possess the ability to operate at an infinite CFL condition in the linear case; however, the nonlinearity of the Navier-Stokes equations prohibits such performance. This three block calculation with 4352 cells performed at a speed of  $2.19 \times 10^{-5} \left( \frac{\text{CPU seconds}}{\text{iteration-gridpoint}} \right)$  on the Cray Y-MP at NASA Lewis Research Center.



**Figure 12** Convergence histories for backward facing step flow at  $Re_D=389$ —(a)  $CFL=7$  with various pseudo compressibility parameter ( $\beta$ ) values and (b)  $\beta=1.0$  with various CFL number values.

## Implementation

The DTNS2D implementation of the pseudo compressibility method offers several important choices to the CMOTT turbulence modelling community. One aspect is the choice of approximate Riemann solver. The user is able to choose a third order accurate Chakravarthy and Osher TVD method. This method is based upon Roe's scheme and is widely accepted as an accurate and efficient method for solution of hyperbolic systems. The user is also able to choose between several time integration schemes, including: approximate factorization, lower-upper symmetric successive over-relaxation, or explicit Runge-Kutta methods. Another helpful feature is the multiblock grid capability. This makes the solution of backward facing step flow possible in both sections of the tunnel. The code has already incorporated the algebraic eddy viscosity model of Baldwin and Lomax [20] as well as a differential  $k-\epsilon$  approach of Gorski [21]. Further turbulence modelling efforts can effectively build upon this ground work. Overall, the code is well organized: the three dimensional and the axisymmetric versions share the same structural framework. This will be most advantageous when upgrading to three dimensional studies. As mentioned earlier, DTNS2D is capable of resolving unsteady flow, although validating the implementation is beyond the scope of this paper.

## Conclusions

The accuracy and efficiency of the 2D incompressible flow code DTNS2D were established for two laminar benchmark flow geometries: flat plate and backward facing step.

Accurate Blasius velocity profiles were presented for flows in the range  $2.0 \times 10^4 \leq Re_x \leq 1.01 \times 10^7$  over uniform and nonuniform cartesian grids. The corresponding convergence patterns, although sensitive to the value of the pseudo compressibility parameter  $\beta$ , were very good. Similarly encouraging results were observed for the backward facing step calculations. The accuracy observed in the two dimensional flow regime of  $Re_D \leq 400$  can be seen in the primary reattachment location,  $x_1$ , and the velocity profiles taken along the tunnel downstream of the step. The onset of three dimensional flow, corresponding to the appearance of a secondary recirculation zone, was accurately predicted.

Analysis indicates that the efficiency can be improved further by concentrating efforts on the Riemann solver and TVD routines. Future plans include an implementation of the advection upwind splitting method (AUSM) developed by Liou and Steffen [22]. This scheme, coupled with a high order accuracy monotone interpolation may simplify the solution algorithm without sacrificing any performance.

Although an optimum value of the pseudo compressibility parameter is fairly difficult to predict a priori, it was relatively easy to find empirically. Often a value for  $\beta$  of unity was satisfactory, if not optimum. This was especially true for the backward facing step flow. This problem needs to be addressed more thoroughly. Nevertheless, the method of pseudo compressibility is a proper framework for numerical solution of the incompressible Navier Stokes equations; DTNS2D is recommended as a satisfactory implementation for use as a test-bed for turbulence modelling.



## References

- [1] F. H. Harlow and J. E. Welch, "Numerical Calculation of Time-Dependant Viscous Incompressible Flow with Free Surface," *Physics of Fluids*, vol. 8, no. 12, pp. 2182–2189, 1965.
- [2] A. J. Chorin, "A Numerical Method for Solving Incompressible Viscous Flow Problems," *Journal of Computational Physics*, vol. 2, pp. 12–26, 1967.
- [3] J. L. Steger and P. Kutler, "Implicit Finite Difference Procedures for the Computation of Vortex Wakes," *AIAA Journal*, vol. 15, no. 4, pp. 2182–2189, 1977.
- [4] R. M. Beam and R. F. Warming, "An Implicit Finite Difference Algorithm for Hyperbolic Systems in Conservation-Law Form," *Journal of Computational Physics*, vol. 22, pp. 87–110, 1976.
- [5] J. L. C. Chang and D. Kwak, "On the Method of Pseudo Compressibility for Numerically Solving Incompressible Flows," 1984. AIAA paper 84–0252.
- [6] D. Kwak and S. R. Chakravarthy, "A Three Dimensional Incompressible Navier-Stokes Flow Solver Using Primitive Variables," *AIAA Journal*, vol. 24, pp. 87–110, 1986. 3.
- [7] W. Y. Soh, "Time Marching Solution of Incompressible Navier-Stokes Equations for Internal Flow," *Journal of Computational Physics*, vol. 70, pp. 232–252, 1987.
- [8] A. Rizzi and L. E. Eriksson, "Computation of Inviscid Incompressible Flow with Rotation," *Journal of Fluid Mechanics*, vol. 153, pp. 275–312, 1985.
- [9] J. J. Gorski, "Incompressible Cascade Calculations Using an Upwind Differenced TVD Scheme," in *Advances and Applications in Computational Fluid Dynamics* (O. Baysal, ed.), vol. 66 of *ASME-FED*, pp. 61–69, 1988.
- [10] J. J. Gorski, "TVD Solutions of the Incompressible Navier-Stokes Equations With an Implicit Multigrid Scheme," in *Proceedings of the AIAA/ASME/SIAM/APS 1st National Fluid Dynamics Congress*, vol. 1, pp. 394–401, 1988.
- [11] J. J. Gorski, "Solutions of the Incompressible Navier-Stokes Equations Using an Upwind Differenced TVD Scheme," in *Proceedings of the 11th International Conference on Numerical Methods in Fluid Dynamics*, Lecture Notes in Physics, pp. 278–282, 1988.
- [12] D. Choi and C. L. Merkle, "Application of Time Iterative Schemes to Incompressible Flow," *AIAA Journal*, vol. 23, pp. 87–110, 1985. 10.
- [13] D. A. Anderson, J. C. Tannehill, and R. H. Pletcher, *Computational Fluid Mechanics and Heat Transfer*, pp. 247–249. Hemisphere Publishing Corporation, 1984.
- [14] K. Morgan, J. Periaux, and F. Thomasset, eds., *Analysis of Laminar Flow Over a Backward Facing Step: a GAMM Workshop*, vol. 9 of *Notes on Fluid Mechanics*. 1984.

- [15] M. K. Denham and M. A. Patrick, "Laminar Flow Over a Downstream-Facing Step in a Two-Dimensional Flow Channel," in *Transactions of the Institute of Chemical Engineers*, vol. 52, pp. 361–367, 1974.
- [16] S. Thangam, "Analysis of Two Equation Turbulence Models For Recirculating Flows," NASA Contractor Report 187607 ICASE Report No. 91–61, 1991.
- [17] S. Thangam and C. G. Speziale, "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.
- [18] B. F. Armaly, F. Durst, J. C. F. Pereira, and B. Schönung, "Experimental and Theoretical Investigation of Backward-Facing Step Flow," *Journal of Fluid Mechanics*, vol. 127, pp. 473–496, 1983.
- [19] Y. S. Chen, "Viscous Flow Computations Using a Second-Order Upwind Differencing Scheme," 1988. AIAA paper 88–0417.
- [20] B. S. Baldwin and H. Lomax, "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," 1978. AIAA paper 78–257.
- [21] J. J. Gorski, "A New Near Wall Formulation for the  $k$ - $\epsilon$  Equations of Turbulence," 1986. AIAA paper 86–0556.
- [22] M.-S. Liou and C. J. Steffen, Jr., "A New Flux Splitting Scheme," NASA Technical Memorandum 104404, 1991.







# REPORT DOCUMENTATION PAGE

*Form Approved*  
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

<b>1. AGENCY USE ONLY</b> ( <i>Leave blank</i> )	<b>2. REPORT DATE</b> March 1992	<b>3. REPORT TYPE AND DATES COVERED</b> Technical Memorandum	
<b>4. TITLE AND SUBTITLE</b> An Investigation of DTNS2D for Use as an Incompressible Turbulence Modelling Test-bed		<b>5. FUNDING NUMBERS</b>  WU-505-62-21	
<b>6. AUTHOR(S)</b> Christopher J. Steffen, Jr.			
<b>7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)</b>  National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		<b>8. PERFORMING ORGANIZATION REPORT NUMBER</b>  E-6924	
<b>9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES)</b>  National Aeronautics and Space Administration Washington, D.C. 20546-0001		<b>10. SPONSORING/MONITORING AGENCY REPORT NUMBER</b>  NASA TM-105593 ICOMP-92-07; CMOTT-92-04	
<b>11. SUPPLEMENTARY NOTES</b> Christopher J. Steffen, Jr., NASA Lewis Research Center and Institute for Computational Mechanics in Propulsion and Center for Modelling of Turbulence and Transition. Responsible person, Christopher J. Steffen, Jr., (216) 433-8508.			
<b>12a. DISTRIBUTION/AVAILABILITY STATEMENT</b>  Unclassified - Unlimited Subject Category 64		<b>12b. DISTRIBUTION CODE</b>	
<b>13. ABSTRACT</b> ( <i>Maximum 200 words</i> )  This paper documents an investigation of a two dimensional, incompressible Navier-Stokes solver for use as a test-bed for turbulence modelling. DTNS2D is the code under consideration for use at the Center for Modelling of Turbulence and Transition (CMOTT). This code was created by Gorski at the David Taylor Research Center and incorporates the pseudo compressibility method. Two laminar benchmark flows are used to measure the performance and implementation of the method. The classical 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. Velocity profiles, convergence histories, and reattachment lengths are used to quantify these calculations. The organization and adaptability of the code are also examined in light of the role as a numerical test-bed.			
<b>14. SUBJECT TERMS</b> Computational fluid dynamics; Incompressible flow; Artificial compressibility; Upwind scheme		<b>15. NUMBER OF PAGES</b> 18	
		<b>16. PRICE CODE</b> A03	
<b>17. SECURITY CLASSIFICATION OF REPORT</b> Unclassified	<b>18. SECURITY CLASSIFICATION OF THIS PAGE</b> Unclassified	<b>19. SECURITY CLASSIFICATION OF ABSTRACT</b> Unclassified	<b>20. LIMITATION OF ABSTRACT</b>