Verification of the Proteus Two-Dimensional Navier-Stokes Code for Flat Plate and Pipe Flows

Julianne M. Conley
Lewis Research Center
Cleveland, Ohio

and

Patrick L. Zeman
Arnold Engineering and Development Center
Arnold AFB, Tennessee

Prepared for the 27th Joint Propulsion Conference
cosponsored by AIAA, SAE, ASME, and ASEE
Sacramento, California, June 24–27, 1991
VERIFICATION OF THE PROTEUS TWO-
DIMENSIONAL NAVIER-STOKES CODE FOR FLAT
PLATE AND PIPE FLOWS

Julianne M. Conley
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

Patrick L. Zeman
Arnold Engineering and Development Center
Arnold AFB, Tennessee

Abstract

The Proteus Navier-Stokes Code is evaluated for two-
dimensional/axisymmetric, viscous, incompressible, internal
and external flows. The particular cases to be discussed are
laminar and turbulent flows over a flat plate, laminar and turbu-
 lent developing pipe flows and turbulent pipe flow with swirl.
Results are compared with exact solutions, empirical
correlations and experimental data. A detailed description of the
code set-up, including boundary conditions, initial conditions,
grid size and grid packing is given for each case.

Introduction

An effort is underway at the NASA Lewis Research Center
to develop a two and three-dimensional Navier-Stokes code,
called Proteus, for aerospace propulsion applications.(1) The
emphasis in this effort is not algorithm development or research
on numerical methods, but on the development of the code
itself. The objective is to develop a code that is user-oriented,
easily modified, and well documented. Code readability,
modularity, and both internal and external documentation have
been emphasized.

Proteus solves the Reynolds-averaged, unsteady,
compressible Navier-Stokes equations in strong conservation
law form. Turbulence is modeled using a Baldwin-Lomax(2)
based algebraic eddy viscosity model. The governing equations
are written in Cartesian coordinates and transformed into
generalized nonorthogonal body-fitted coordinates. They are
solved by marching in time using a fully-coupled alternating
direction implicit solution procedure with generalized first or
second order time differencing.(3-4) The boundary conditions
are also treated implicitly, and may be steady or unsteady. All
terms, including the diffusion terms, are linearized using second
order Taylor series expansions.

Two versions of the Proteus code exist: one for two-
dimensional planar and axisymmetric flow, and one for three-
dimensional flow. In addition to solving the full time-averaged
Navier-Stokes equations, Proteus includes options to solve the
thin-layer or Euler equations, and to eliminate the energy
equation by assuming constant stagnation enthalpy. Artificial
viscosity is used to minimize the odd-even decoupling resulting
from the use of central spatial differing for the convective
terms, and to control pre- and post-shock oscillations in super-
sonic flow. Two artificial viscosity models are available -- a
combination implicit/explicit constant coefficient model (5), and
an explicit nonlinear coefficient model designed specifically for
flows with shock waves.(6-7). At the NASA Lewis Research
Center, the code is typically run either on the CRAY X-MP or
the CRAY Y-MP computer, and is highly vectorized.

In order to assess the code's validity for calculating funda-
mental fluid flows encountered in most aerospace propulsion
applications, a series of validation cases have been run, using the
two-dimensional planar/axisymmetric version of the code.
These cases are for both internal and external incompressible
flows. This paper describes validation studies for laminar and
turbulent flat plate boundary layers with zero pressure gradient,
and for laminar and turbulent developing pipe flows and
turbulent pipe flow with swirl. Incompressible cases in Proteus
were simulated by running at a Mach number between 0.1 and
0.3. In the results for both flat plate and pipe flow to be
presented, constant total enthalpy was assumed, and the energy
equation was not solved.

Test Cases

Laminar Flat Plate Flow

Incompressible laminar flow over a flat plate with zero pres-
sure gradient can be compared with the exact solution of
Blasius.(8) The results of one such comparison are shown in
Figures 2-6, plotted with the results of Blasius. For this test
case, the freestream Mach number was 0.2 and Re x, the
Reynolds number based on x, ranged from 20,000 at the
upstream computational boundary to 100,000 at the downstream
computational boundary. A 201x101 grid was used, with,
packing in the vertical direction near the plate surface such that
the ratio of the minimum to maximum cell height, defined as the
packing ratio, was 0.05; the grid was uniform in the x-direction.
The grid extended horizontally from x/L = 0.25 to x/L = 1.25
and vertically from y/L = 0.0 to y/L = 0.05, where L is a
reference length used by Proteus to normalize input values. For
this test case, L = 52δmax, where δmax is the maximum
boundary layer thickness. A portion of the grid extending from
x/L = 0.25 to x/L = 0.29 is illustrated in Figure 1. For the
initial conditions, u, the horizontal x-velocity, and v, the vertical
y-velocity, were computed using the Blasius solution. The static
pressure, p, was set to p∞, the freestream static pressure,
everywhere. For the boundary conditions, at the upstream
boundary, p and v were held at the initial condition values. At
the downstream boundary, p = p∞, u = u∞ and ∂u/∂y = 0.
At the surface, ∂p/∂y = 0, and u = v = 0. At the freestream
boundary, p = p∞, u = u∞ and ∂v/∂y = 0.

The results shown in Figures 2-6 were obtained after 4100
iterations. Figure 2 shows the x-velocity profile plotted against
the Blasius similarity coordinate, η, where

\[ \eta = \sqrt{u_\infty / v x} \]

with ν as the kinematic viscosity. Here, the Proteus results are
indistinguishable from the Blasius profile, indicating excellent
performance by Proteus. In the y-velocity profile of Figure 3,
the Blasius results are also indistinguishable from the Proteus

*Aerospace Engineer
Member AIAA
Fig. 1. A portion of the grid used for laminar flat plate calculations.

Fig. 2. X-velocity profiles for laminar flat plate flow.

Fig. 3. Y-velocity profile for laminar flat plate flow.

Fig. 4. Local skin friction coefficient for laminar flat plate flow.

Fig. 5. Momentum thickness for laminar flat plate flow.

results. Figure 4 shows the local skin friction coefficient plotted against $Re_\theta$, the Reynolds number based on $\theta$, the momentum thickness. Figure 5 shows $\theta$ versus $x$ and Figure 6 shows the displacement thickness, $\delta^*$, versus $x$. Figures 4 through 6 all exhibit excellent agreement between the Proteus results and the Blasius solution. Thus, Proteus is capable of accurately calculating incompressible laminar flow over a flat plate.
A study was done to minimize the number of grid points in the streamwise and normal directions required to accurately compute the above described laminar flow. The results of this study are shown in Figures 7 and 8. In Figures 7a-7d, the number of streamwise grid points is decreased from 101 to 13 points, while the number of normal grid points is held constant at 101. The results begin to deviate from the Blasius solution below 26 points, as shown in the deviation of the 13x101 curve of Figure 7d. This shows that a minimum of 26 streamwise grid points are required for an accurate solution. In Figures 8a-8c, 26 grid points are used in the streamwise direction, and the number of grid points in the normal direction is decreased from 101 to 26 points. The results begin to disagree with the Blasius curve when fewer than 51 points are used, as seen by the deviation in the 26x26 curve of Figure 8c. Thus, the smallest grid needed to accurately calculate this laminar flow over a flat plate is a 26x51 grid.

Fig. 6. Displacement thickness for laminar flat plate flow.

Fig. 7a-d. Variation in the number of streamwise grid points.
Turbulent Flat Plate Flow

Results for incompressible turbulent flow over a flat plate are shown in Figures 9-11. For the cases shown, the freestream Mach number was 0.2 and $Re_\infty$ ranged from 4,000,000 at the upstream computational boundary to 16,000,000 at the downstream boundary. A 101x191 grid was used with packing in the vertical direction at the plate surface such that the packing ratio was 0.005. Grid packing was also used in the x-direction at the upstream boundary such that the packing ratio was 0.05. The grid extended from $x/L = 0.33$ to $x/L = 1.33$ and from $y/L = 0.0$ to $y/L = 0.048$, where $L = 58_{\text{max}}$. For the initial conditions, $u$ was determined from an expression developed by Musker(9), with $v = 0$, and $p = p_{\infty}$. The boundary conditions were identical to those for the laminar flat plate case.

Figure 9 shows the x-velocity plotted with $y/\delta$, where $\delta$ is the boundary layer thickness. Note that the Proteus results at the three different Reynolds numbers shown are each represented by a curve and agree so closely that it is difficult to distinguish them on the plot. This is to be expected since the profiles are plotted with similarity coordinates. The results also show good agreement with the experimental data of Klebanoff.(10) Figure 10 shows the same Proteus results plotted on a semi-log graph with $u^+$ and $y^+$ coordinates, where $u^+ = u/u_\tau$, with $u_\tau$ equal to the shear velocity, and $y^+ = yu_\tau/v$. These results show good agreement with the law of the wall correlation.(11) Figure 11 shows a plot of the local skin friction coefficient, $c_f$, versus $Re_\infty$, compared with the Karman-Schoenherr correlation(12) and the experimental data of Weighardt.(13) Notice that the Proteus results exhibit a drop in $c_f$ at the upstream boundary, where $Re_x = 4,000,000$ or $Re_\infty = 6,500$ with the remaining portion of the curve in agreement with the data of References 14 and 15. This drop at the upstream boundary is most likely a result of using an inexact boundary condition at this boundary. Recall that at the upstream boundary, a $u$-profile was approximated and $v$ was set to zero. Other upstream boundary conditions were also
considered, such as using a u-profile computed from the Musker expression and v either computed from the continuity equation or extrapolated, or moving the upstream boundary to the leading edge of the plate where \( u = u_{in} \) and \( v = 0 \). These boundary conditions, however, were not as effective as the chosen conditions. Overall, Figures 9-11 show that the Proteus performance is very good for incompressible, turbulent flat plate flow.

Laminar Developing Pipe Flow

The Proteus calculations for incompressible laminar developing pipe flow are shown in Figures 12 and 13 compared with the experimental data of Reshotko. For this test case, an average Mach number of approximately 0.1 was used, and \( \text{Re_R} \), the Reynolds number based on the pipe diameter, was 100. The pipe length was set to 10 diameters, and a 51 axial by 21 radial grid was used. For the initial flow field, \( u = v = 0 \) and \( p = p_r \), where \( p_r \) is the reference pressure which was set to standard sea level pressure. For the boundary conditions, the inlet and exit pressure were chosen to achieve a pressure drop calculated by pipe design formulas. For the remaining inlet boundary conditions, \( \partial^2 u / \partial x^2 = 0 \) and \( \partial v / \partial x = 0 \). The remaining exit conditions were \( \partial u / \partial x = \partial v / \partial x = 0 \). At the pipe wall, \( \partial p / \partial r = 0 \) and \( u = v = 0 \). The centerline boundary conditions were standard symmetry conditions such that \( \partial p / \partial r = \partial u / \partial r = 0 \) and \( v = 0 \).
Turbulent Developing Pipe Flow

The results for incompressible turbulent developing pipe flow are shown in Figures 14 and 15 compared with the experimental data of Barbin. This case had an average Mach number of approximately 0.09 and a Reynolds number of 388,000. The pipe length was set to 50 diameters, and a 101 axial by 51 radial grid was used with a packing ratio of 0.05 near the wall. The initial and boundary conditions used were identical to those of the laminar developing pipe flow case.

Figures 14 and 15 show the Proteus results in a manner analogous to Figures 12 and 13 for laminar developing pipe flow. The value of \( u \) at the pipe inlet is approximately equal to the average velocity, \( u_0 \), which would be expected for turbulent developing pipe flow. Also, the Proteus results closely agree with the experimental data, with a slight deviation in the near-wall region. Thus, Proteus is capable of calculating turbulent developing pipe flows.

Swirling Developed Pipe Flow

In this validation case, the Proteus results for swirling incompressible turbulent pipe flow are compared with the experimental data of Weske. The Mach number was 0.1 and Reynolds number of 30,000. The pipe length was set to 50 diameters with a 400 axial by 50 radial grid, and a packing ratio of 0.1 near the wall. The initial conditions for \( u \) were calculated using the 1/7th power law, with the boundary layer thickness approximated as 10% of the pipe radius. The initial swirl velocity profile was linear with the swirl velocity \( w = 0 \) at the centerline and increasing to a maximum of \( w = u_0 \) near the wall, where \( u_0 \) is the centerline axial velocity for this case. This gives the swirl number of \( \sigma = 1.0 \), where \( \sigma = \frac{w_{\text{max}}}{u_0} \). The remaining initial conditions were \( p = p_e \) and \( v = 0 \). For the boundary conditions, the inlet and exit pressure were chosen so that the pressure drop coincided with the design pipe calculation value, ignoring the unknown effects of the swirling velocity component. The inlet velocities were held at the initial condition values and at the exit, \( \frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = \frac{\partial w}{\partial x} = 0 \). At the pipe wall, \( \frac{\partial p}{\partial r} = 0 \) and \( u = v = w = 0 \). At the centerline, \( \frac{\partial p}{\partial r} = \frac{\partial u}{\partial r} = 0 \) and \( v = w = 0 \).
Figure 16 shows the Proteus swirl velocity profiles as curves and the experimental data as symbols. The plot shows a general agreement of the profile shape and swirl decay as the flow works its way down the pipe; however, the Proteus curves do not agree well with the data. The disagreement can be attributed to the inability of the algebraic eddy viscosity turbulence model to handle the anisotropies of this complex flow. Yoo et al. (17) describe the problems of computing the turbulence field for a similar flow.

Concluding Remarks

Validation cases for both laminar and turbulent incompressible flow over a flat plate at zero pressure gradient showed excellent agreement with exact solutions, empirical correlations and experimental data. It was also shown that a 26x51 grid with packing near the wall gives sufficient resolution to calculate laminar flat plate flow. The velocity profiles of both laminar and turbulent developing pipe flow agreed with experimental data, with slight deviations near the pipe wall. Pipe flow with a swirl number of 1.0 showed the expected profile shape and swirl decay; however, the swirl velocity profiles did not coincide with experimental data. This is a shortcoming of the algebraic eddy viscosity model used in Proteus for computing swirling pipe flows. With this exception, Proteus is proven to be effective for calculating simple internal and external, incompressible, viscous flows.

Validation of Proteus is ongoing. Future plans include verification of higher Mach number flows and flows with heat transfer.

References

Verification of the Proteus Two-Dimensional Navier-Stokes Code for Flat Plate and Pipe Flows

Julianne M. Conley and Patrick L. Zeman

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135-3191

National Aeronautics and Space Administration
Washington, D.C. 20546-0001


The Proteus Navier-Stokes Code is evaluated for two-dimensional/axisymmetric, viscous, incompressible, internal and external flows. The particular cases to be discussed are laminar and turbulent flows over a flat plate, laminar and turbulent developing pipe flows and turbulent pipe flow with swirl. Results are compared with exact solutions, empirical correlations and experimental data. A detailed description of the code set-up, including boundary conditions, initial conditions, grid size and grid packing is given for each case.