Computation of Velocity, Pressure and Temperature Distributions near a Stagnation Point in Planar Laminar Viscous Incompressible Flow

E. Kaufman[1] and E. Gutierrez-Miravete[*,2]
[1]Pratt and Whitney, [2]Rensselaer at Hartford
*Corresponding author: 275 Windsor Street, Hartford, CT 06120, gutiee@rpi.edu

Abstract: As gas turbine engine turbine temperatures and component life requirements continue to rise, it becomes increasingly important to have a good understanding of the operating temperatures of turbine components. Recent work has explored the possibility of approximating the leading edge stagnation heat transfer coefficient through analogy with the Hiemenz flow solution for stagnating plane flow in front of a flat wall. The objective of this study was to set the framework for an exploration into whether a Hiemenz flow approximation based on measured static pressures near an airfoil leading edge can provide a good estimate of the leading edge heat transfer coefficient. To achieve the stated objective, the Hiemenz solution was reproduced using MATLAB in order to obtain a reliable baseline for comparison. Finite element and finite volume CFD solvers COMSOL and FLUENT, respectively, were each used to produce numerical solutions of the same problem and the results compared.

Keywords: Stagnation flow, laminar, incompressible, viscous fluid.

1. Introduction

The two-dimensional plane flow of an inviscid incompressible fluid approaching a wall is shown in Figure 1. The wall is located at y=0, while x=0 is a symmetry plane. The stagnation point is at the origin. The fluid moves towards the wall from the positive y-direction and is then deflected by it and moves along the positive x-direction.

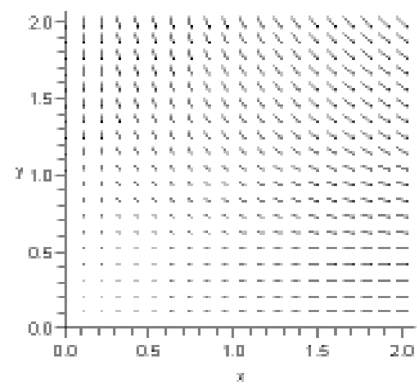

Figure 1. Velocity field near a stagnation point for the planar laminar flow of an inviscid incompressible fluid.

This flow can be solved analytically and the solution, in terms of the stream function is given by $\psi = axy$ [1]. This solution for inviscid flow in the vicinity of a stagnation point is the basis for the derivation of the similarity flow solution for the flow of a viscous fluid in the same situation. As outlined in [2,3], the inviscid stream function is modified so that the no-slip condition at the wall can be satisfied and one writes instead $\psi_{viscous} = axf(y)$, where the function f depends only on the coordinate y. Hence, once values for $f(y)$ have been determined, the velocity and pressure distributions for the viscous case can be determined.

The velocity components can be described in terms of the function f:

$$u = ax\frac{df}{dy}$$
$$v = -af(y)$$

The pressure distribution can be described as follows, introducing the function *F(y)*:

$$p_0 - p = \tfrac{1}{2}\rho a^2\left[x^2 + F(y)\right]$$

If the stagnation pressure p_0 is taken as the pressure at the stagnation point, where $x=y=0$, we can extract the condition:

$$F(y)\big|_{y=0} = 0$$

The incorporation of the above together with the no-slip condition and the Navier-Stokes equations yields Hiemenz equation for the dimensionless function $\phi(\eta)$:

$$\phi''' + \phi\phi'' - \phi'^2 + 1 = 0$$

Where

$$\phi(\eta) = f(y)/\sqrt{a\nu}$$

and

$$\eta = \sqrt{\frac{a}{\nu}}\,y$$

Here ν is the kinematic viscosity of the fluid. Hiemenz equation is a third order, non-linear ordinary differential equation that must be solved subject to the following boundary conditions:

$$\phi'\big|_{\eta=0} = 0$$

$$\phi\big|_{\eta=0} = 0$$

$$\phi'\big|_{\eta\to\infty} = 1$$

No analytical solution of the above problem is possible; therefore a numerical method must be used as described below.

2. Methodology

The boundary value problem described in the previous section is solved by first transforming the original equation into a system of first order ordinary differential equations than are then solved using a fourth order Runge-Kutta method combined with shooting within the MATLAB environment [4]. The results obtained using this procedure were in excellent agreement with previously published results and are used as our baseline for comparison against predictions using COMSOL and the CFD program FLUENT. Utmost care had to be used to ensure the far-field conditions in these simulations matched those used in the original method.

Both COMSOL and FLUENT [5] flow solvers were used to develop models of the viscous flow field in the case of a 2-dimensional, steady, incompressible viscous flow with constant properties. Selection of appropriate domain and boundary conditions are essential to obtaining a solution consistent with the results obtained from the Hiemenz equation. Several variations on the geometry and boundary conditions were investigated.

Each of these cases resulted in streamlines and pressures that were qualitatively consistent with the analytical solutions. However, pressure magnitudes did not generally agree well with the Hiemenz solution.

The most successful case for the purposes of comparison was one in which the domain was 6 meters by 6 meters, and inlet and exit boundary conditions were specified so as to be consistent with the boundary conditions used in the analytical case. The domain size of 6 was sufficient to ensure that the boundaries were effectively at far field; i.e. the velocities should have asymptotically achieved their far-field character by the boundaries of the domain. This was established based on the viscous exact solution, for which 99% of far-field conditions are achieved at 3 units from the wall. The COMSOL and FLUENT meshes used are shown in Figures 2a and 2b, respectively.

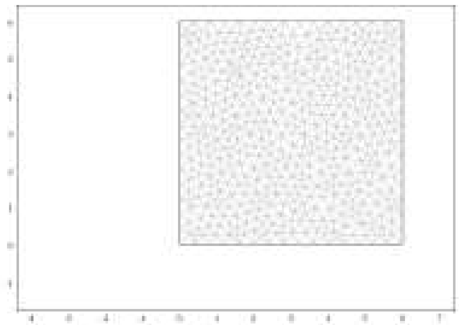

Figure 2a. Mesh used in the COMSOL model.

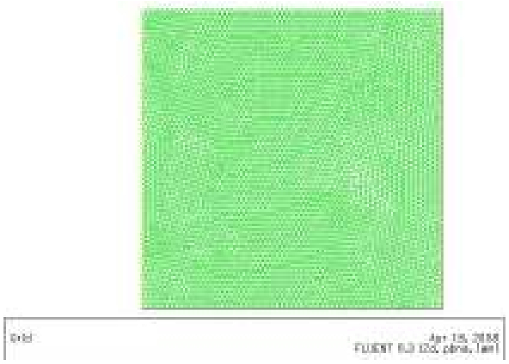

Figure 2b. Mesh used in the FLUENT model.

For the computations, the inlet velocity was set such that $U = x$ and V=-5.3521 (m/s). The inlet x-velocity comes from the boundary condition $U = ax$ far from the wall, and the inlet y-velocity was based on the results of the viscous (Hiemenz) solution. The specified inlet x-velocity defines the final parameter, a, required to ensure that the case is properly non-dimensionalized ($a = 1\ \mathrm{s^{-1}}$). The inlet y-velocity is equal to the analytical prediction at y=η=6. The density was set to 1 $\mathrm{kgm^{-3}}$ and viscosity was set equal to 1 $\mathrm{kgm^{-1}s^{-1}}$. These values simplify the comparison between the analytical and numerical results. Selecting this set of initial conditions and material properties ensures that the numerical values of the dimensionless function $\phi(\eta)$ are equivalent to those of the dimensional function $f(y)$, and that the values of the dimensionless coordinate η and the dimensional y are also equal. (The dimensions of both y and $f(y)$ are meters). The exit pressure was specified so as to be consistent with the analytical prediction at x=6, given a reference pressure p_0 of 100 Pa.

3. Results

The velocity fields computed using COMSOL and FLUENT are shown in Figures 3a and 3b, respectively. The velocity fields predicted by these 2 codes are nearly identical, with a maximum value of 8.04 m/s in each case. This maximum value is also consistent with the viscous analytical solution, which is (u,v)=(6,-5.3521) at the corner (x,y)=(6,6). This velocity magnitude of 8.04 m/s is slightly lower than that predicted by the inviscid solution, 8.49 m/s.

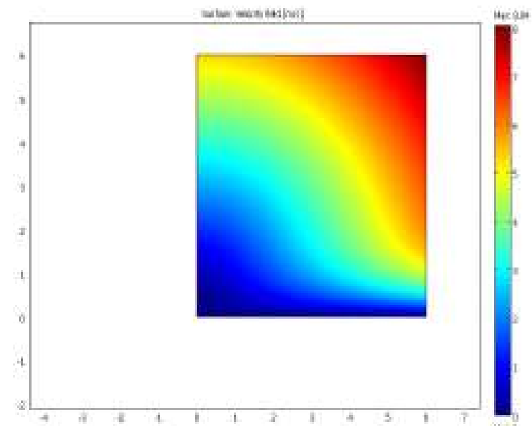

Figure 3a. Velocity field computed with COMSOL.

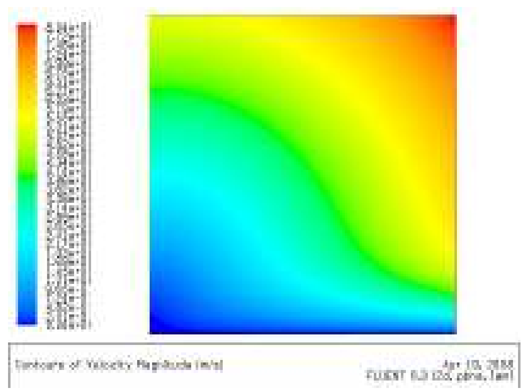

Figure 3b. Velocity field computed with FLUENT.

The pressure fields computed by COMSOL and FLUENT are shown in Figures 4a and 4b, respectively. They are both close in shape to the analytical solution, and in each case indicate a stagnation pressure very close to the specified value of 100 Pa. The contours are, however, slightly different between the 2 codes. This could be attributable to several factors, including differences in mesh density (which is visible in Figures 2a and 2b); the discretization of applied boundary condition profiles, which was comparable to the mesh density in the FLUENT case but finer than the COMSOL case, or perhaps to solver differences.

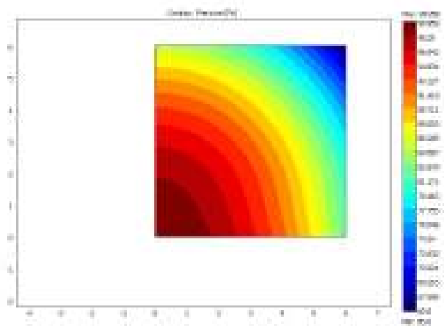

Figure 4a. Pressure field computed with COMSOL.

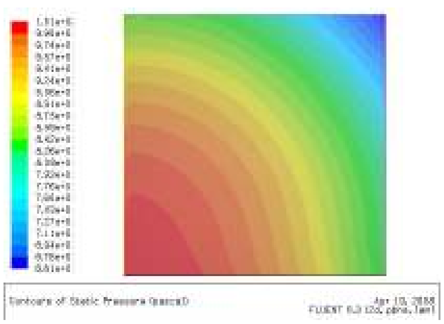

Figure 4b. Pressure field computed with FLUENT.

Figure 5, shows the computed pressure distributions along the symmetry line and in close proximity to the wall for the four models considered (namely, the inviscid case, Hiemenz, COMSOL and FLUENT). The figure shows that there is very good agreement between the Hiemenz and COMSOL results and small, but not negligible differences are obtained between the COMSOL and FLUENT results.

Finally, Figure 6 shows computed results for the temperature profiles along the symmetry line for the same four cases considered. The agreement is excellent for the three viscous methods.

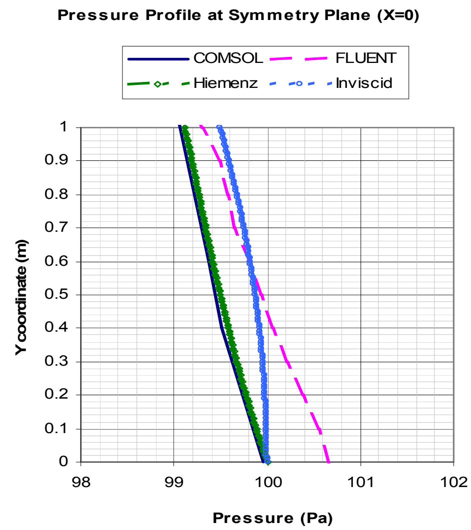

Figure 5. Pressure distributions computed along the symmetry line, near the stagnation point for all four cases considered.

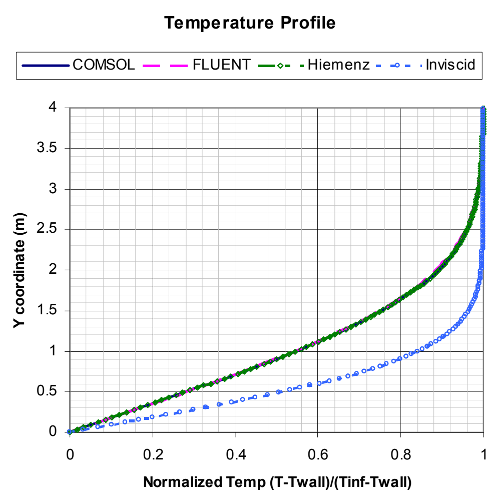

Figure 6. Temperature distributions computed along the symmetry line, near the stagnation point for all four cases considered.

4. Conclusions

We have investigated the planar laminar flow of an incompressible viscous fluid in the vicinity of a stagnation point. The solution first obtained by Hiemenz was reproduced and used as a baseline for comparison. Finite element (using COMSOL) and finite volume (using FLUENT) models of the problem were developed to investigate their performance against the baseline. Both, the COMSOL and FLUENT simulations were shown to be reliable for predicting the velocity, pressure and temperature distributions in this case.

8. References

1. F. B. Hildebrand, Advanced Calculus for Applications, 2nd ed. Prentice-Hall, Inc. 1976, p. 316.
2. H. Schlichting and K. Gersten, Boundary Layer Theory, 8th ed., Springer, NY, 200, p. 110.
3. Holly, Brian M. and Lee S. Langston, "Analytical Modeling of Turbine Cascade Leading Edge Heat Transfer using Skin Friction and Pressure Measurements", GT2007-28120, Proceedings of GT200 AMSE Turbo Expo 2007: Power for Land, Sea and Air, May 14-17, Montréal, Canada.
4. R. Burden and J.D. Faires, Numerical Analysis, 7th ed. Brooks-Cole, CA, 2001, chapters 5 and 11.
5. FLUENT Website (http://www.fluent.com/)