Excerpt from the Proceedings of the COMSOL Conference 2010 Boston

Influence of Inlet Fluctuations on the Development of the Turbulent Two-Stream Mixing Layer

Jonathan A. Tawfik and Rao V. Arimilli
The University of Tennessee
414 Dougherty Engineering Building, The University of Tennessee, Knoxville, TN 37996

Abstract: The finite element method applied to the k-ε turbulence model is used to investigate the two-stream turbulent mixing layer. Whereas the model is known as one of the most popular of the turbulence models to date, the model has yet to be applied to the classical mixing layer problem to the best of our knowledge. A transient k-ε turbulence model in COMSOL version 3.5a is used to solve this problem. In this work, a new method for visualization of vortex-shedding is demonstrated by solving for the transient response with a uniform inlet velocity superposed with stream-wise fluctuating velocity components upon the individual streams. The random fluctuating velocity imposed is similar to velocity measurements encountered in previously reported experiments with user-controlled frequency and amplitude factors. This method predicts good agreement with the motivating literature but does not show self-similarity for the fluctuating terms until $x/\delta_s(0) \cong 57$. The results show that the time-averaged results of a transient solution of this type will yield an identical result to the steady-state solution.

Keywords: mixing layer evolution, turbulent flow, fluctuating velocity, k-ε model

1. Introduction

Free shear flows are found both in nature and in important engineering applications, including chemical process control, combustion, and environmental dispersion. Thus, the ability to understand and predict free shear flow behavior is a necessary and significant task of many research establishments. Furthermore, the classical two-stream turbulent mixing layer problem is the subject of much experimental work in order to understand and quantify the physical nature of turbulent flow, and the ability to accurately represent the classical problem will validate the method for a variety of engineering applications.

The classical mixing layer problem has been studied as early as 1943 when Görtler in [1] suggested a self-similar profile of the form $U^* = \frac{1}{2}(1 + \mathrm{erf}(\eta))$. Several authors have simulated the natural evolution of mixing layer vortex phenomenon of experiments by imposing time-varying inlet conditions in their numerical studies of turbulent flow. Sandham [2] applied a random-walk to the phase of the forcing eigenfunctions at the inlet boundary using a finite difference method without the use of a turbulence model. The random-walk in his work is appropriate for simulating actual turbulent flow velocity measurements one might obtain from experiment. Inoue [3] applied forcing functions at the inlet prescribed as a superposition of the fundamental frequency with its sub-harmonic frequencies; he showed the effectiveness of this technique to control the mixing layer growth using a vortex method. Stanley [4] investigated the effects of inflow forcing to the mixing layer problem by direct numerical simulation and showed that their results are in agreement with the self-similar measurements obtained by Plesniak [5].

The purpose of the present work is to investigate new methods to predict the overall flow-field quantities of spatially evolving free shear flows. The two-stream turbulent mixing-layer problem is solved using superposition of random turbulent fluctuations upon the individual streams using the finite element model applied to the k-ε turbulence model. The affect of superposing random frequencies on the uniform inlet velocities is investigated in order to simulate the random velocity measurements obtained in experiment. The finite element method applied to the k-ε turbulence model has not been previously applied to the turbulent two-stream mixing layer to the best of our knowledge. Both the steady and time-averaged unsteady results were obtained using COMSOL version 3.5a. Spatial evolution of the mixing layer is determined from the unsteady solution. The self-similarity of the results discussed demonstrated agreement with previous experiments and numerical methods.

2. Computational Procedure

2.1. Overview

The problem is solved in a three-step process because the domain and mesh is loosely dependent on the results. A broad overview of the method is informative before details of the procedure are discussed. We proceed under the assumption that the steady-state results will identically represent time-averaged transient results. As a first step, a large domain is initialized of the order of approximately 3 meters wide and 10 meters long with a mean flow direction to the right. From this model, the initial shear layer thickness, $\delta_s(0)$, is calculated for which model parameters (overall dimensions, mesh element lengths, turbulent length scale, etc.) are non-dimensionalized in the next steps. This initial shear layer thickness is effectively taken at a near-field x-location, that is, before vortical structures are expected in a transient model. For the purposes of this investigation, $\delta_s(0)$ is taken at $x = 0.5$ m. In the second step, the domain size and mesh elements are reconfigured with respect to the shear layer thickness calculated in the previous step, and a second steady-state result is obtained. The steady-state result is used to initialize the transient problem and transient results are obtained in the third step with random fluctuations applied at the inlet.

2.2. Computational Domain and Boundaries

The essential characteristic of an accurate model of free shear turbulent flow in an open space is that the boundary conditions are chosen such that they do not interact with the physical nature of the flow. The size of the computational domain is chosen such that a small area of interest is imbedded in a much larger fluid domain, placing the boundaries far from the area of interest. The boundary condition specification has been cited by previous authors as one of the greatest difficulties in the simulation of a flow that is essentially in an open domain. Previous efforts have brought forth boundary conditions that are non-reflective for the Euler equations. Giles [6], Thompson [7] and Poinsot [8] are among the authors that have been cited in the formation of boundary conditions for open flow computations.

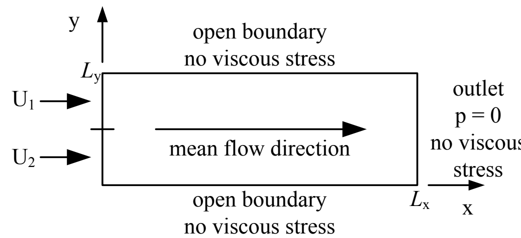

Figure 1. Boundary Condition Specifications of Transient Model

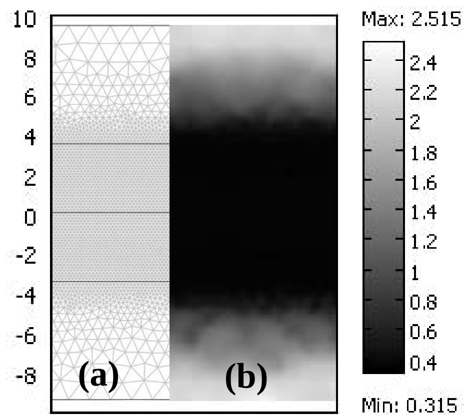

Figure 2. Cross section of computational fluid domain showing (a) mesh and (b) dimensionless mesh element distribution, $\boldsymbol{h/\delta_s(0)}$.

The boundary conditions of the mixing layer problem in a semi-infinite domain are specified as $u(+y \to \infty) = \bar{U}_1$ and $u(-y \to \infty) = \bar{U}_2$, according to Görtler [1]. We expect that fluid flows tangentially to the computational fluid domain and mass does not cross the boundaries in a steady-state solution, deeming symmetric boundary conditions appropriate for the upper and lower boundaries in steady-state computations. The steady-state result is readily obtained and used to initialize the transient model. A foregoing assumption for obtaining a transient response is that information that passes across these boundaries is essentially lost and not important if the domain is large enough. The open boundaries require that the viscous stresses be zero with an additional pressure correction at the outflow boundary.

Figure 1 shows a description of the boundary conditions used in the computational domain of the transient mixing layer problem for this investigation. One can obtain the shear layer thickness before computing the transient response since the solution is initialized with a steady-state response. The shear or vorticity thickness is defined here as (1).

$$\delta_s(x) = \frac{\bar{U}_1 - \bar{U}_2}{(\partial \bar{u}/\partial y)_{max,y=0}} \quad (1)$$

The initial shear thickness calculated using (1) at $x = 0.5$ m is $\delta_s(0) = 0.0786$ m. With the shear thickness known, the domain and mesh for the transient model are reconfigured according to Stanley [4] as a guideline, and foregoing parameters are non-dimensionalized with respect to the shear thickness as defined above. A new domain with a size of $L_x = 143\delta_s(0)$ and $L_y = 38\delta_s(0)$ is split into an inner and outer sub-domain. An inner sub-domain of a width $L_y = 14\delta_s(0)$ extends the length of the entire computational domain in order to concentrate a fine mesh around the mixing layer where gradients are expected to be high; and the outer domain close to the upper and lower boundaries is given a relatively courser mesh. Figure 2 shows a cross-sectional view of the computational domain with the resultant mesh plot and mesh element length, $h/\delta_s(0)$ at each y-location. Four hundred mesh elements are placed along the $y = 0$ line, for which the inner sub-domain is meshed and elements are not allowed to grow. In the outer sub-domain, an element growth rate of 1.15 is specified.

2.3. Inlet Conditions

One attribute of the k-ε turbulence model as given in COMSOL 3.5a is the user's ability to either specify the turbulent length scale and the turbulence intensity or an initial value for k and ε, denoted as k_0 and ε_0, respectively. Details of the vortex shedding phenomenon inside the mixing layer are more apparent when the turbulent length scale is minimized since this parameter directly translates to the diameter of the smallest eddy resolved in the model. For this reason, we specified the turbulent length scale and intensity at the inlet. Vortex shedding is observed for transient results of a mixing layer model even when the domain is first initialized with a steady-state result. Stanley [4] and Sandham [2] have initialized their transient

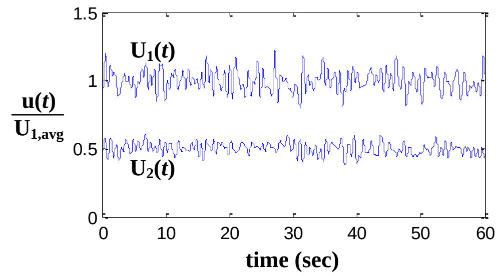

Figure 3. Inlet Velocity Profile

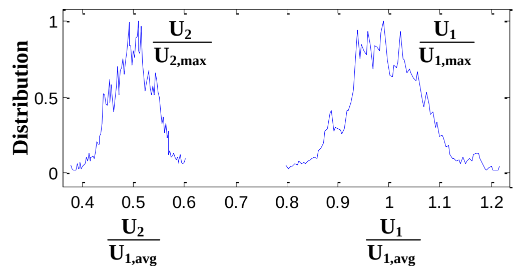

Figure 4. Relative distribution of inlet velocity

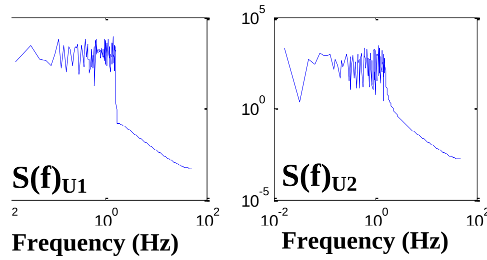

Figure 5. Spectral Analysis of Inlet Velocity Profile

results with the hyperbolic tangent shear-layer profile at the inlet in order to model flow over a splitter plate. For the purpose of this investigation, an initial condition is specified in the entire computational domain rather than only at the inlet boundary.

A new aspect of the computational procedure is in the characteristics of the inlet velocity giving rise to a vortex shedding phenomenon. Previous authors have attempted many methods beginning with understanding the natural frequency of the system that causes the flow to be most unstable. Essentially, we are modeling the velocity measurements of a turbulent flow experiment by superposing multiple random frequencies over a mean input velocity. This approach gains merit since the results of the computations are readily matched with the results of the experiments. The method adds additional controlling parameters on the model

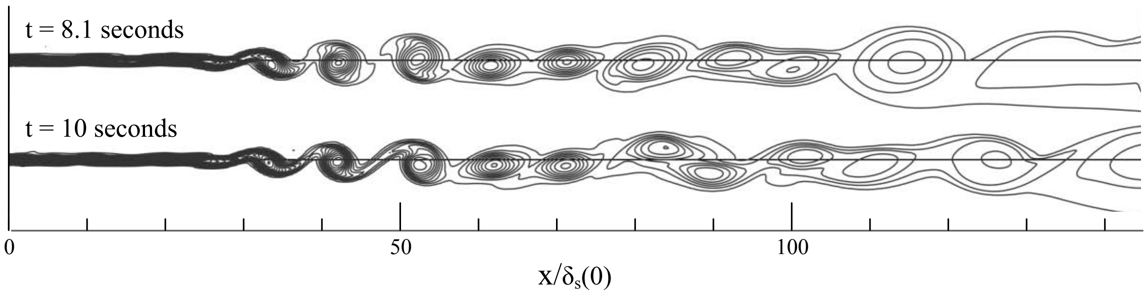

Figure 6. Vorticity, Ω, contour plot at two separate times. Vorticity range for contours shown is $-60 < \Omega < -0.2\ s^{-1}$.

which are amplitude and frequency factors. Figure 3 is a computer-generated inlet velocity with amplitude and frequency factors scaled appropriately to model an actual turbulent flow velocity signal.

In Figure 3, the nominal velocity ratio for the simulation is $U_2/U_1 = 0.5$ which is consistent with the values used by other researchers cited. This ratio also holds for the standard deviation and the average absolute error as well. Note that the amplitude of the fluctuations is proportional to the mean velocity of the individual stream but the frequency remains consistent. This is more apparent in observance of the velocity distribution and spectral analysis given in

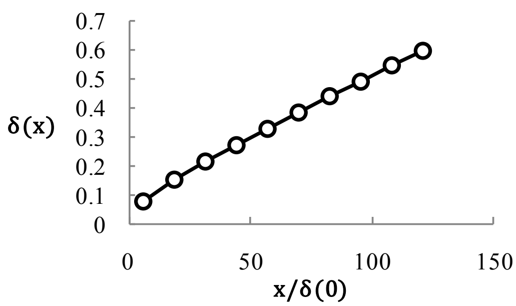

Figure 7. Growth of the shear layer thickness

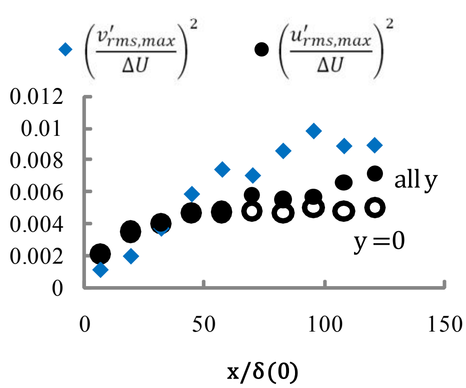

Figure 8. Growth of fluctuating velocity components

Figures 5 and 6, respectively. These figures are representative of uniform distribution of fluctuating components about a mean velocity. Although a Gaussian distribution exists about the mean velocity, this is considered as a low noise system because the spectral analysis trails off above approximately 10^0 Hz.

3. Results.

Computational results were obtained for water fluid properties, and with an inlet velocity profile as previously discussed. The results capturing the vortex shedding phenomenon of water for ten seconds was used to compute the similarity profiles and these are in agreement with those presented by Plesniak [5]. The turbulent length scale for the flow visualization in Figure 6 is $L_T/\delta_s(0) = 0.002$.

Time-averaging a transient mixing layer problem yields results identical to the steady-state solution with the advantage that fluctuating velocity characteristics can be analyzed. Note that fluctuating components of velocity observed are in the y-direction even though the inlet profile has only fluctuations specified in the streamwise direction. In order to demonstrate self-similarity, the x- and y-velocity components are extracted from the model at intervals of $\Delta t = 0.1$ seconds. The x- and y-velocity components are fluctuating with time as is the derivative of the mean x-velocity component, $\partial\bar{u}/\partial y$, from which much of the physics of the problem is derived. The maximum time-averaged value of $\partial\bar{u}/\partial y$ is used to calculate the shear layer thickness using (1), and implicitly, the growth of the mixing layer shown here as Figure 7.

The growth of the fluctuating velocity components are given in terms of their root mean square values in Figure 8. Of particular interest

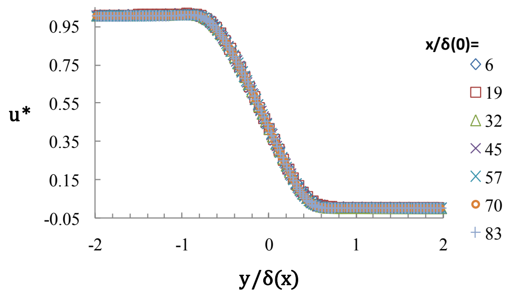

Figure 9. Self similarity of the mean velocity at different streamwise locations

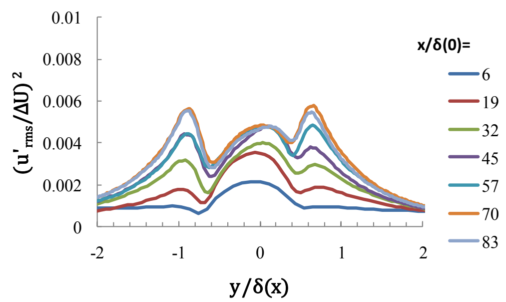

Figure 10. Self similarity of the $\boldsymbol{y}$-component of fluctuating velocity reached for $\boldsymbol{x/\delta(0)} \geq \mathbf{45}$

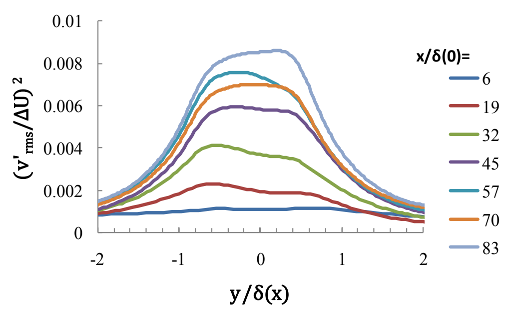

Figure 11. Approaching self similarity of the $\boldsymbol{y}$-component of fluctuating velocity for $\boldsymbol{x/\delta(0)} = \mathbf{57}$

is that the fluctuating components of velocity are still growing at the exit of the domain when plotted together with the flow direction. Self-similarity is demonstrated with respect to the shear layer thickness in light of the foregoing results and is given here in Figures 9-11. The model problem shows very good agreement with the literature and the same results is found for both the steady-state and the time-averaged transient solution, yet the fluctuating components of velocity in the flow and normal directions are self-similar and nearly self-similar at $x/\delta(0) = 57$, respectively.

4. Discussion

Since the k-ε turbulence model is known for application in free shear flows, it follows that it is an appropriate model for the turbulent two-stream mixing layer, which is demonstrated here using COMSOL 3.5a. The growth of the mixing layer can be attributed to the amplitude scaling of the inlet velocity fluctuations prescribed using this method. Although the model correctly predicted linear growth of the shear layer thickness, it under-predicted the rate of growth $d\delta(x)/dx = 0.0045$, compared with Stanley [4] of 0.045 and the results from the k-ε turbulence model are expected to change if the amplitude factor of the inlet fluctuations is increased. Although the scope of this investigation is limited to an introduction of the method, the growth of the shear layer is used as an indication of how the amplitude factor should be modified in succeeding simulations by examination of the mixing layer growth.

The results presented in Figure 8 indicate the growth of $\left(u'_{rms,max}/\Delta U\right)^2$ at $y = 0$ and for "all y" and is understood when one realizes that the results are a reproduction of the self-similar results in Figure 10. It should be noted that $u'_{rms,max}$ does not have a maximum at $y = 0$. The maximum value of these fluctuations originally occur at the $y = 0$ location, but then changes to the edge of the mixing layer, and can possibly be attributed to the boundary conditions. However, boundary conditions have proven to be an important factor in the eventual convergence of such a model.

5. Conclusion.

A new method is investigated in solving the classical two-stream turbulent mixing layer problem. The finite element method applied to the k-ε turbulence model subject to random streamwise fluctuations prescribed at the inlet boundary is employed, and the results are in good agreement with the motivating literature of various experiments and computational methods. The aforementioned method investigated here is shown to simulate both qualitative and quantitative observed phenomena of turbulent mixing and vortex patterns.

The scope of this investigation covers an introduction of the new method and two new factors of the model contribute to the results presented here, namely a frequency and amplitude factor introduced in the algorithm used to generate the fluctuating inlet velocity profile. A recommendation for future study is to conduct a parametric study of the factors influencing the results for which boundary conditions, the turbulent length scale, and amplitude and frequency factors may be included. A future investigation specific to the influence of the boundary conditions and possible implementation of the non-reflecting boundary conditions in the literature is an equally important recommendation.

6. References

1. Görtler, H. "Berechnung von Aufgaben der freien Turbulenz auf Grund eines neuen Naherungsansatzes." Z. Angew. Math. Mech. vol. 22, pp. 244-254. (1942)

2. Sandham, N.D., and Reynolds, W. C. "Some inlet plane effects on the numerically simulated spatially two dimensional mixing layer." in Symposium on Turbulent Shear Flows, 6th, Toulouse, France, Sept. 7-9, 1987, Proceedings (A88-38951 15-34). University Park, PA, Pennsylvania State University. pp. 22-4-1 to 22-4-6. (1987)

3. Inoue, Osamu. "Note on multiple-frequency forcing on mixing layers." Fluid Dynamics Research. vol. 16, pp. 161-172. (1994)

4. Stanley, Scott and Sarkar, Sutanu. "Simulations of Spatially Developing Two-Dimensional Shear Layers and Jets." Theoretical and Computational Fluid Dynamics. vol. 9, pp. 121-147. (1997)

5. Plesniak, Michael W.; Mehta, Rabindra D.; Johnston, James P. Curved two-stream turbulent mixing layers: Three-dimensional structure and streamwise evolution, *Journal of Fluid Mechanics*, **270**, pp. 1-50. (1994)

6. Poinsot, T.J., and Lele, S.K. Boundary conditions for direct simulation of compressible viscous flows. *J. Comput. Phys.,* **101**, pp. 104-129. (1992)

7. Giles, M.B. Nonreflecting boundary conditions for Euler equation calculations. *AIAA J.*, **28**(12), pp. 2050-2058. (1990)

8. Thompson, K.W. Time dependent boundary conditions for hyperbolic systems. *J. Comput. Phys.*, **68**, pp. 1-24. (1987)

7. Appendix: MATLAB Code for Inlet Velocity

```
dt = 0.01; % change in time
nfluct = 2000; % frequencies
tfinal = 60; % seconds
AmpFluct =  0.0025; % amplitude
pFactor = 0.1; % frequency

t = 0:dt:tfinal; % time vector
rand('state',sum(100*clock));

% Upper velocity functions
U1mean = 10;
a1 = rand(1,nfluct)/pFactor;
U1 = U1mean;
for n = 1:length(a1)
    U1 = U1 + ...
         ((-1)^n)*...
         U1mean*AmpFluct*...
         sin(a1(n)*t);
end; clear n;

% Lower velocity functions
U2mean = U1mean/2;
a2 = rand(1,nfluct)/pFactor;
U2 = U2mean;
for n = 1:length(a2)
    U2 = U2 + ...
         ((-1)^n)*...
         U2mean*AmpFluct*...
         sin(a2(n)*t);
end
```