A hybrid vortex filament scheme with the capability of simulating bounded turbulent flows is described. Viscous generation of new vortex elements at solid surfaces is accomplished through the intermediary of solving the viscous vorticity transport equation on a thin boundary mesh via a finite difference and finite volume method. The transitional and turbulent boundary layer flow past a wide, finite thickness, flat plate with rounded edges is computed with a view toward validating the methodology and gaining new insight into the structural aspects of transition. The predicted mean velocity and related statistics are well matched to experimental and numerical data. The representation of transitional flow through vortex filaments is shown to suggest a need for reinterpreting the meaning of some of its basic structural features. In particular, the motions commonly attributed to the presence in the flow of discrete self-contained vortical structures may in some instances be more precisely described as representing differing aspects of the transitional vortex sheet covering the wall region.

Nomenclature

\begin{itemize}
  \item \( D \) Drag
  \item \( M \) Number of wall-normal grid levels
  \item \( L_T \) Average triangle edge length
  \item \( R_e \) Reynolds number
  \item \( s \) Axial vector on vortex tube
  \item \( U, u \) Streamwise velocity and fluctuation
  \item \( U_i \) Velocity vector
  \item \( U_\infty \) Far field velocity
  \item \( U_r \) Friction velocity
  \item \( V, v \) Wall normal velocity and fluctuation
  \item \( V_T \) Prism volume
  \item \( W, w \) Spanwise velocity and fluctuation
  \item \( x \) Streamwise coordinate
  \item \( y \) Wall normal coordinate
\end{itemize}
I. Introduction

That it may be advantageous to represent turbulent flow numerically via gridfree vortex filaments is a natural response to the highly rotational character of turbulence that includes at one extreme small-scale worm-like vortices and at the other well organized vortices associated with the flow origins.\(^1\) The recognition of the central importance of vortex dynamics to understanding turbulent physics is also implicit in the many studies that seek to extract vortex structure from computed velocity fields\(^2\) or even go so far as to reconstruct from the velocity field the time dependent behavior of the vortical structures.\(^3\) Such analyses − which are inherently difficult for traditional grid-based schemes − are straightforward for the vortex filament approach since the vortex dynamics are provided directly.

As in grid-based simulations of turbulence, it is not generally practical to represent the full range of scales with a filament scheme, so that the most useful context for such an approach is as a large eddy simulation (LES) where the effects of small, invisible, “sub-grid” motions on the resolved are modeled. Recent applications of such a gridfree LES method to turbulent free shear flows including the mixing layer\(^4\) and round, coflowing jet\(^5\) have demonstrated some of the advantages of the approach. The present paper is concerned with the implementation of this idea in the more general context of flows containing solid walls with a primary focus on the turbulent boundary layer.

In traditional grid-based LES, each vortex of resolvable scale that appears in the simulated flow field requires a collection of individual mesh points in its representation. The same vortex is generally more efficiently resolved in terms of vortex elements, though a potentially more significant advantage of a filament scheme is that without a mesh, flow features that otherwise may be smeared by numerical diffusion\(^6,7\) remain sharply defined. In the context of an LES it is also necessary to provide a subgrid-scale model that acts as a gatekeeper for two-way energy transfer between small dissipative scales and large resolved scales. Primarily for reasons having to do with numerical stability, subgrid models used in grid-based schemes tend to rely on diffusive models of questionable physical relevance that can also distort important vortical flow features as well as inhibit local backscatter. In contrast, the vortex filament method relies on a non-diffusive vortex loop removal algorithm\(^8-10\) that accounts for local energy dissipation without compromising the sharpness of vortical flow features or hindering backscatter.

Whatever advantages vortex filaments might have in representing turbulence structure, this is lost in the region flush against solid boundaries where the vorticity field acquires in part a largely two-dimensional sheet-like character. In fact, gradients in the vorticity field at the wall surface control viscous production of vorticity, and it is unlikely that tube-like elements even in prodigious numbers can do justice to the accurate representation of this type of field. It is also well known that essentially all scales of motion next to a wall need to be resolved since no part of the motion can be safely considered “subgrid” and suitable for modeling. These considerations motivate the belief that if the filament scheme is to be extended to include solid boundaries then it must be done so in such a way that the near wall vorticity field − specifically that part within which viscous production of new vorticity takes place − is simulated with the resolution and
accuracy of a conventional direct numerical simulation (DNS).

In this paper a hybrid scheme is described that combines the vortex filament representation of the flow outside the immediate vicinity of walls, with a finite volume and finite difference algorithm applied to the solution of the full three-dimensional viscous vorticity equation on a thin mesh situated against bounding solid surfaces. New filaments appear in the flow at the outer edge of the near-wall mesh. The latter is limited in thickness to the extent necessary to well represent near-wall viscous vorticity diffusion. In practice, the wall mesh is erected from a surface triangularization maintaining a sheet-like aspect ratio to the prism elements, and thus avoids many of the complexities normally associated with three-dimensional mesh development.

The turbulent boundary layer is treated here as an important step in validating the vortex filament methodology in the presence of solid walls. At the same time it will be seen that the vortex calculations offer a novel viewpoint with which to examine the physics of the boundary layer and from so doing arrive at new understanding of some of its essential features. A case in point will be the opportunity to take a more holistic view of the vortical field during transition than has been normally possible, and from this add to our understanding of the origin, meaning and behavior of previously noted vortical features.

Due to its fundamental nature, the boundary layer both transitional and fully turbulent has been the object of many previous numerical studies involving both DNS and LES. Among the first DNS calculations, Spalart \cite{spalart11} accounted for spatial growth via a mapping technique so as to enable the use of a spectral scheme with streamwise periodic boundary conditions. This study produced well documented averaged flow data for the turbulent boundary layer at several Reynolds numbers that is helpful for validating the present calculations.

The focus of the present work is the spatially growing boundary layer and an early study of this flow was performed by Rai and Moin \cite{raim12} using a zonal finite difference approach. They computed the boundary layer transition into the turbulent regime. Many subsequent DNS studies of the spatially developing boundary layer have focused on the transitional case in which a variety of different modes of the process have been investigated. A distillation of results on the structure of transition from such works, bolstered with insights from physical experiments, provides a helpful backdrop with which to view the findings of the present simulations. Among the prior numerical studies, Rist and Fasel \cite{rist13} studied the evolution of controlled disturbances and gained insights into the growth and breakup of vortical structures. Jacobs and Durbin \cite{jacobs14} simulated by-pass transition in a spatially growing boundary layer excited by free stream turbulence. They examined the origins of low speed streaks and their association with the mechanisms causing the appearance of turbulent spots. The dynamics of the so-called Klebanoff transition were studied via matched experiments and DNS by Bake, Meyer and Rist. \cite{bake15} A detailed scenario was developed connecting the onset of random disturbances in the boundary layer with the growth and evolution of vortical structures. Ovchinnikov, Choudhari and Piomelli \cite{ovchinnikov16} computed the effect of large amplitude free stream turbulence on the transitional boundary layer noting circumstances when the breakdown of low speed streaks occurs vs. the appearance of turbulent spots.

It is also of interest to contrast the performance of the present method, for which no distinction has to be made between transitional and fully turbulent flow, and traditional LES schemes that require specially adapted subgrid models in order to cover the full range of phenomena. For example, Ducros, Comte and Lesieur \cite{ducros17} simulated a forced, narrow, Mach 0.5 boundary layer through transition to turbulence using a sub-grid model that allows for weak perturbations without overly dissipating the turbulent field and were able to qualitatively describe some aspects of transition into turbulence. The turbulent region of the flow retained structural features such as streaks and bursting that are commonly observed in other simulations. Another application of LES to boundary layer modeling in the compressible regime was performed by Kawai and Fujii, \cite{kawai18} in which the effects of numerical and modeling parameters on the physicality of the transitional and turbulent structure were explored.

Despite the unorthodoxy of the vortex filament representation of the boundary layer used in the present study, it will be clear that the statistical measures of the computed field in terms of mean velocity and other properties conform to that found in traditional studies. Moreover, it will be seen that a connection can be made between the physical appearance of the filament field and the coherent vortical objects that have played a major role in categorizing the dynamical processes at work in the boundary layer – both
transitional and turbulent. It is through this connection that a potentially fruitful means of unifying many aspects of the physics of the boundary layer is created.

The hybrid vortex filament approach will be described in the next section with an emphasis on the finite volume and finite difference algorithm. This is followed by a discussion of the numerical problem to be considered with subsequent sections describing simulations that are separately concerned with transition and the fully turbulent state. Finally, some conclusions are drawn.

II. Numerical Method

Various aspects of the numerical simulation technique are now described with somewhat greater emphasis on the near-wall treatment. Additional information about the vortex filament computation can be found in the literature.4, 5, 8

II.A. Vortex Tubes

The vortex filament method uses short, straight, vortex tubes linked end-to-end forming filaments as the basic computational element. Vortex tubes that stretch beyond a threshold are subdivided. An essential aspect of the technique is the removal of vortex loops that form naturally out of the filaments as the tubes stretch and fold in turbulent flow regions. As first suggested by Chorin9, 10 and demonstrated in applications4, 5, 8 this is an effective means of accommodating non-local intermittent energy dissipation at inertial range scales without the expense of computing the details of viscous dissipation at much smaller scales. At the same time, loop removal does not hinder the tendency of the vortex filaments to combine forming large scale structures — essentially, the phenomenon of backscatter. Loop removal is also of considerable practical importance since it prevents what would ordinarily be an exponential growth in the number of tubes produced as a byproduct of the energy cascade to small dissipative scales caused by vortex stretching and folding.

Vortices are produced in the calculation from vorticity created at the solid surfaces as described in the next section. As the vortices stretch the number of vortices increases by subdivision and in turbulent regions their number is controlled via loop removal. It is also necessary to introduce a downstream boundary beyond which vortices are removed from the computation. For the present simulations, entire filaments are removed when each of the component vortices is past the exit plane. As has been mentioned in other contexts,4, 5 removal of vorticity outside the computational domain can be expected to have some effect on the flow inside due to the use of the Biot-Savart law in computing velocities. For the present work this means that a region just upstream of the exit plane will be less accurate and thus not suitable for analysis.

II.B. Finite Volume/Finite Difference Scheme

To accommodate the need for DNS resolution adjacent to solid boundaries, a fine mesh is constructed from a surface triangularization by erecting perpendiculars at the nodal points. \( M - 1 \) layers of thickness \( \Delta y \) are placed on top of an initial half-thickness layer19 giving a total thickness of the mesh equal to \( (M - 1/2)\Delta y \), where \( y \) is in the direction normal to the surface and \( x, z \) are the tangential coordinates. The region covered by triangular prisms is taken to be large enough to encompass the viscous sublayer and adjacent region of the turbulent boundary layer — that part of the flow where the wall-normal gradients of the vorticity are large. In terms of wall units \( y^+ = yU_\tau/\nu \), where the friction velocity \( U_\tau = \sqrt{\nu \partial U/\partial y(0)} \), the mesh is set to lie approximately within \( y^+ = 25 \) of the boundary. The precise distance in wall units at the outer edge of the mesh will vary from place to place in any given calculation.

An important consideration is the aspect ratio of the triangular prisms defined as the average edge length divided by the thickness of a prism. In the interest of efficiency — both to minimize the number of surface elements and to enhance the velocity computation described below, it is advantageous to have the prisms be “sheet-like” in character, that is the aspect ratio should not be too small. On the other hand, discretization errors will grow and resolution will be lost if the aspect ratio is too large. Empirical tests suggest a value
around 10 is acceptable so that if $M = 10$, then $\Delta y^+ \approx 3$ (with the half-sheet at the wall of thickness 1.5) and $\Delta x^+$, $\Delta z^+$ are no larger than 30. These dimensions are at the upper range of those required for a DNS – most significantly $\Delta z^+$ should be smaller – but prove to be adequate for most aspects of the present simulations as will be seen below.

The vorticity field in the mesh, $\Omega_i$, is determined as a solution to the vorticity transport equation

$$\frac{\partial \Omega_i}{\partial t} + U_j \frac{\partial \Omega_i}{\partial x_j} = \Omega_j \frac{\partial U_i}{\partial x_j} + \frac{1}{Re} \nabla^2 \Omega_i,$$

(1)

where $U_i$ is the velocity field and $Re$ is the Reynolds number. The finite volume/finite difference scheme is developed based on the properties of triangular prisms such as shown in Fig. 1. In this, the vorticity is taken to be constant over the prisms and equal to that at the prism center, while the velocity is computed at the top and bottom triangles, as indicated in the figure. The velocity at each of the six nodal corners of the prism is determined via area-weighted averaging of the velocities on the adjacent triangles.

![Figure 1. A typical triangular prism. ∗, location of $\Omega_i$; +, location of computed $U_i$.](image)

The convection term in Eq. 1 is approximated by first expressing it in the equivalent conservative form

$$U_j \frac{\partial \Omega_i}{\partial x_j} = \frac{\partial U_j \Omega_i}{\partial x_j},$$

(2)

averaging it over the prism, and then applying the divergence theorem, leading to

$$\frac{\partial U_j \Omega_i}{\partial x_j} \approx \frac{1}{V_T} \int_{V_T} \frac{\partial U_j \Omega_i}{\partial x_j} dV = \frac{1}{V_T} \sum_k A^k \Omega_i^k (U_j^k n_j^k),$$

(3)

where the sum is over the five faces of the prism, and for the $k$th face of the prism, $A^k$ is the area, $n_j^k$ is the outward pointing unit normal vector, and $\Omega_i^k$ is the vorticity. On the triangular surfaces the velocity needed in the sum is directly available, while on the side surfaces it is computed from averaging the four nodal values. $\Omega_i^k$ on the two triangular faces is determined from the value on the upwind prism. For example, if the velocity on the top face of the prism in Fig. 1 is up, then $\Omega_i^k$ is taken from the prism itself. Otherwise, it is taken from the prism lying above it. In the case of the fluxes through the quadrilateral sides of the prism, the vorticity is computed by a linear least square fit of the vorticities in prisms that are contiguous to the prism on the upwind side of the surface on the same and the immediately neighboring levels above.
and below, plus the one prism on the downwind side. This is illustrated in Fig. 2 for flow from left to right across the indicated plane.

The evaluation of the stretching term in (1) is done at the center of the prisms so the vorticity appearing in the expression is directly available. The computation of the velocity gradient, \( \partial U_i / \partial x_j \), is done following a similar approach as in computing the convection term. In this case, the scalar divergence theorem is used to obtain

\[
\frac{\partial U_i}{\partial x_j} \approx \frac{1}{V_T} \sum_k A^k U_i^k n_j^k.
\]

The value of \( U_i^k \) on each surface is determined the same way as in the convection term.

The evaluation of the diffusion term in (1) distinguishes between diffusion normal and parallel to the surface. In particular, the Laplacian is first expressed in a local rectangular Cartesian coordinate system with \( n \) in the direction normal to the solid surface and \( t_1 \) and \( t_2 \) tangent to the surface, giving

\[
\nabla^2 \Omega_i = \frac{\partial^2 \Omega_i}{\partial n^2} + \frac{\partial^2 \Omega_i}{\partial t_1^2} + \frac{\partial^2 \Omega_i}{\partial t_2^2}.
\]

The normal diffusion term in (5) is evaluated using a standard second-order finite difference formula using the vorticity values at the centers of the prisms. The two tangential terms are computed by differentiation of a polynomial determined by a second order least-square fit of the vorticities of prisms located within a given radius, say \( r_D \), of the prism center and in the layers immediately above and below the prism. The default radius is taken to satisfy \( r_D = 1.2 L_T \) where \( L_T \) is the average edge length of the triangles.

As for the accuracy of the discretizations, the upwind advection scheme is nominally second order as is the formula for the Laplacian in the normal direction. The diffusion scheme in the tangential plane is first order and the stretching terms also can be expected to be essentially second order. With these approximations the overall scheme offers stable solutions to the 3D vorticity equation with each separate part being consistent and convergent under mesh refinement.

**II.C. Time Integration**

An explicit first order in time Euler scheme is used for advancing the finite volume and finite difference calculation near the wall. In this, the time step \( \Delta t \) is limited by CFL conditions in the normal and tangential
directions, namely,

\[ \Delta t \leq \min(C_{CFL} \Delta y/V_{\text{max}}, C_{CFL} L_T/U_{\text{max}}) \]  

(6)

where \( C_{CFL} \) is a parameter usually taken to be 0.3, \( V_{\text{max}} \) is the largest wall-normal velocity in the mesh and \( U_{\text{max}} \) is the largest tangential velocity. The time step is also limited by a diffusive stability requirement that has

\[ \Delta t \leq \frac{1}{8R_e} L_T^2. \]  

(7)

In view of the relatively small size of \( \Delta y \) in comparison to \( \Delta x \) and \( \Delta z \) the condition in the normal direction in (6) tends to be the strictest. The time step for this part of the calculation is quite small justifying the use of the simple explicit time marching. In practice, \( \Delta t \) satisfying conditions such as (6) and (7) is smaller than it would have to be in order to compute accurate motion of the vortex tubes. Consequently, \( N_S \) time steps of the wall calculation on the mesh – referred to as subcycles – are performed before updating the positions of the vortex tubes. Typically, \( N_S = 10 \) and the overall time step in the advancement of the solution is \( \Delta t^* = N_S \Delta t \). With this larger time step a fourth order Runge-Kutta scheme is used to advance the vortex tubes in time.

II.D. Boundary Conditions

The vorticity components at the wall surface represent boundary conditions for the computation of vorticity in the mesh. The two tangential vorticity components are approximated from their component velocity derivatives using the divergence theorem to express their average over the half-thickness wall prism in terms of the velocity on the prism faces and the no slip condition on the wall. The wall-normal vorticity component is identically zero on solid boundaries. In the event that the local boundary element is not aligned with the coordinate system of the calculation, then the latter are determined by projection from the former.

To be able to solve for the vorticity in the wall mesh, it is also necessary to have boundary conditions for the vorticity at the top layer of prisms. Moreover, it is required to compute the amount of vorticity that arrives at the top layer to then be turned into a new vortex tube. Both goals are achieved by imposing a zero net flux condition on the top layer in which vorticity entering from the mesh below exits by convection as a new vortex tube. In this there is assumed to be no net gain of vorticity by viscous diffusion at the top sheet level which is tantamount to assuming that \( \partial^2 \Omega_n / \partial n^2 = 0 \). Moreover, as far as the convective flux between the top two prisms is concerned, when the flux is away from the surface the vorticity at the top sheet is not needed (in view of the upwind model), while if the flux is toward the surface it is taken to be zero since all incoming vorticity is expected to be in the form of vortex tubes.

II.E. Vortex Creation

Vorticity that accumulates during a prescribed time interval (which may be larger than \( \Delta t^* \)) in the top prism at any location due to the viscous and convective fluxes outward from the wall, is turned into a vortex tube as long as its magnitude is larger than a threshold. The orientation of the new vortex is clear from the relative magnitudes of the vorticity components. Its midpoint is at the center of the prism whose vorticity it will take. The length of the new tube, \( |s| \), where \( s \) is the axial vector along the tube, is set by the condition that its ends just intersect the sides of the prism. Its strength is determined via the relation

\[ \Gamma |s| = |\Omega| V_T, \]  

(8)

that forces the far field velocity from the new tube to match that coming from the prism of vorticity it replaces.
II.F. Velocity Evaluation

The velocity field is determined as the sum of contributions from the vortex elements (both sheets and tubes) as determined from approximations to the Biot-Savart law plus a potential flow given by a boundary element scheme that enforces the non-penetration boundary condition. The ith vortex tube contributes according to the relation

\[ \frac{-1}{4\pi |\mathbf{r}_i|^3} \mathbf{r}_i \times \mathbf{s}_i \phi(|\mathbf{r}_i|/\sigma), \]  

where \( \mathbf{x}_1^i, \mathbf{x}_2^i \) are the end points of a tube, \( \mathbf{s}_i = \mathbf{x}_2^i - \mathbf{x}_1^i \) is an axial vector, \( \mathbf{r}_i = \mathbf{x} - \mathbf{x}_i, \mathbf{x}_i = (\mathbf{x}_1^i + \mathbf{x}_2^i)/2, \) \( \Gamma_i \) is the circulation and \( \phi \) is a smoothing function made necessary by the simplicity of the approximation to the Biot-Savart integral that does not take into account the local vortex structure. As in previous studies

\[ \phi(|\mathbf{r}_i|/\sigma) = 1 - \left(1 - \frac{3}{2}(|\mathbf{r}_i|/\sigma)^3\right) e^{-((|\mathbf{r}_i|/\sigma)^3}, \]  

where the smoothing parameter \( \sigma \) determines the distance at which the smoothing takes place from the center of the tubes. Beyond a distance 2.34 \( \sigma, \phi = 1 \) and smoothing is not present.

The velocity associated with an individual prism sheet can be determined by the appropriate Biot-Savart integral over its individual volume using the assumption of constant vorticity. Assuming that the prisms have a sufficiently high aspect ratio it is reasonable to forgo integration in the normal direction in favor of a simple midpoint evaluation. This leaves integration over the triangles themselves, which may be carried out in closed form, though the formulas are lengthy and are not given here. It is also the case that the expense of evaluating the exact relations is such that they are only used for the computation of velocities in the immediate vicinity of any given prism, specifically within a radius \( R_P = 1.5L_T \). At further distances the sheets are regarded as contributing to the velocity as if they were tubes, and for this purpose their strengths are determined from (8). It is thus seen that apart from local formulas, the sheets and tubes contribute similarly to the velocity field.

In the boundary element scheme a source strength is determined on each surface triangle such that the sum of contributions produces a potential flow satisfying the non-penetration boundary condition. The contribution of each triangle is determined by integration of the exact formula assuming constant source strengths. The latter are determined by iteration using a generalized minimal residual (GMRES) method.

The numerical evaluation of the velocity field is carried out using a parallel implementation of the adaptive Fast Multipole Method (FMM) that replaces the nominal \( O(N^2) \) expense of computing the velocities associated with the motion of \( N \) vortices via the Biot-Savart law, by a more practical \( O(N) \) cost. As it is currently constituted, parallelization has been implemented fully for the CPU, but not for memory. Generally for problems of the scale reached in this paper, speed up by a factor of 32 is readily attainable through parallelization.

III. The Numerical Problem

The code used in this study is designed for the treatment of flow past arbitrary 3D smooth bodies, and is readily adapted to boundary layers by applying it to the flow past the smooth plate with rounded edges shown in Fig. For sufficiently wide plates, the mean velocities within a region surrounding the center have the high degree of two-dimensionality appropriate to a traditional boundary layer. After some experimentation, a plate with scaled dimensions 1.5 by 2.5 in the streamwise, \( x \), and spanwise, \( z \), directions, respectively, and thickness 0.05 in the \( y \) direction was adopted. Flow statistics in the subsequent analysis are taken from the middle region \(|z| < 0.25\) over which there is found to be no noticeable spanwise variation in the mean statistics. As will be seen below, this computational test section is sufficiently large to contain...
many independent structural features and is comparable in size to that commonly used in grid-based studies using spanwise periodic boundary conditions.

Smooth edges are applied to the plate by affixing semi-circular columns on all four sides and quarter-spheres at each of the corners. The incoming flow is in the positive x direction with unit velocity and zero turbulence level. A Reynolds number $R_e = U_\infty L/\nu$ is assigned where $L$ and $U_\infty$ are the length and velocity scales, respectively, used in scaling the problem.

Though the plate is relatively slender, it is still thick enough to cause some disturbance to the incoming flow that slightly delays the appearance of canonical boundary layers in the streamwise direction. Thinner plates were not used so as to simplify the generation of the triangularization on the plate sides and corners and to avoid the presence of relatively sharp edges.

The surface triangularization used as the basis for setting up the near-wall grid is also shown in Fig. 3. This contains 62272 triangles with an average length dimension of 0.0114 suggesting that the density of the triangles over the central region $0 \leq x \leq 1.5, |z| < .25$ is roughly equivalent to having a $44 \times 132$ mesh of square grid cells. The mesh of triangular prisms is grown outward automatically from the surface mesh. In all cases considered here, 10 levels of prisms are layered above the initial layer of half-thickness that is adjacent to the boundary.

As mentioned previously, the thickness of the wall mesh is determined consistent with having an aspect ratio of the prisms near 10. Having set this, then a realistic range of Reynolds number will have $\Delta y^+/2 \approx 1.5.$ Based on these considerations two thicknesses for the boundary mesh have been considered here, namely, 0.012 and 0.016 implying that $\Delta y = 0.00126$ and 0.00168, respectively. The average aspect ratios of the prisms for the two cases are then 13.7 and 10.3. For the thinner mesh a computation with $R_e = 80,000$ is performed and for the thicker mesh simulations are performed for $R_e = 30,000$ and 50,000. Each of the computations described here required approximately one week on a SGI Altix 4700 supercomputer at the Pittsburgh Supercomputing Center using 16 nodes.

It is within the capabilities of the code used in this study to allow for surfaces that are constructed from several individual pieces, referred to as “patches.” In fact, the plate in Fig. 3 is made up of 10 patches consisting of the top and bottom flat surfaces, the semi-circular curved sides and the four quarter-spherical corners. Any given surface patch is allowed by the software to be regarded as either inviscid or viscous for the purpose of setting boundary conditions. For the inviscid surfaces only the wall-normal velocity condition is imposed and not the no-slip condition with the consequence that such surfaces do not produce vorticity.

For the purposes of the present study where interest is confined to the flow in the central region, the side
curved edges and the four corners are set to be inviscid surfaces thus saving the cost of computing the motion of vortices created in these places. By the same motivation, the rear curved surface at the exit plane is held to be inviscid as well, so that the boundary layers forming upstream separate without vorticity generation on the back rounded surface.

Computations have revealed that the status of the front curved surface — whether viscous or inviscid — has significant consequences for the subsequent downstream flow development. The sensitivity of the transition to the geometry of the leading edge is well known, and, in particular, a bluff front face tends to produce earlier transition than a more streamlined shape. In the present circumstances, for the viscous blunt front surface a relatively quick transition occurs downstream. However, when this surface is made inviscid it is found that transition occurs much further downstream along the plate. Evidently this must reflect the absence of vorticity produced upstream of the beginning of the flat surfaces. The delayed transition is taken advantage of in this work by providing a boundary layer that allows for a more complete view of the transitional field. The subsequent discussion will make clear that the nature of the transition appears to be the same for both cases — it is just the location of transition that is different. The attention of this study will be directed at computations of the transitioning boundary layer with $R_e = 50,000$ using the inviscid front curved surface and the fully turbulent boundary layer with $R_e = 80,000$ using the viscous front surface.

**IV. Transiting Boundary Layer**

A view from above of the computed vortex field in the test section between $0 \leq x \leq 1.5$ on one side of the plate for the $R_e=50,000$ flow, for which the front curved surface is inviscid, is shown in Fig. 4. The vortices shown in the figure have been selected from the total in the computation by the condition that any part of a given filament be contained within $|z| \leq 0.25$. This explains why there is some lateral spread of the vortices beyond $|z| = 0.25$ in the figure. The flow is from left to right and is seen to be initially laminar, finally undergoing disturbances leading to a clearly visible vortex structure that makes up the transition region on the right. The alignment of vortices in the initial, upstream part of the boundary layer is purely in the spanwise direction. It is seen that despite the fact that individual triangles produce separate vortices, the collective motion is entirely smooth as they initially convect along the plate.

![Figure 4. View from above of the vortex elements in the plate flow with inviscid front boundary.](image)

As mentioned above, the transition in Fig. 4 occupies almost the full length of the plate so that there is just the beginnings of the turbulent field at the right side of the field of view. The first appearance of structure in the transition is plainly visible just beyond the midpoint of the figure at approximately $x = 0.85$. The regions at the right that are barren of vortices have been subjected to intense wallward sweeps of fluid; in actuality there is always substantial vorticity in the grid underlying the vortex elements that is not made visible in the figure.
IV.A. Flow Statistics

The finite thickness of the plate causes some substantial flow distortion in the vicinity of the leading edge. For the present case this is illustrated in Fig. 5 showing the spanwise averaged streamwise velocity profiles at several $x$ positions plotted with respect to $y/\delta$ where

$$\delta = \int_0^\infty \left(1 - \frac{U}{U_\infty}\right) dy$$  \hspace{1cm} (11)

is the displacement thickness. It is seen in the figure that there is an overshoot in the streamwise velocity that gradually dies out with distance along the plate, entirely disappearing by $x = 0.675$. In fact, a closer look at the velocity field shows that there is an approximately 3% enhancement to the free stream velocity outside of the boundary layer that persists along the length of the plate.

![Figure 5. Streamwise velocity for $Re = 50,000$ simulation with inviscid leading edge. (a), $x = 0.225$; (b), $x = 0.375$; (c), $x = 0.525$; (d), $x = 0.675$.](image)

Beyond $x = 0.675$ where the effect of the front end distortion is no longer felt until the clear onset of transition at $x = 0.85$ it is appropriate to compare the computed solution to that of the Blasius boundary layer. In fact, Figure 6 containing a comparison of the computed streamwise velocity and the Blasius profile at $x = 0.675$, shows that the vortex solution does have the characteristics of the Blasius similarity solution. In this, an empirically determined virtual origin of $x = 0.25$ is used in the definition of the similarity variable $\eta = y\sqrt{Re/(x - 0.25)}$. As is the case in other simulations,\(^{15}\) this is necessary when there is some ambiguity inherent in the determination of a leading edge for the boundary layer. The close agreement with the Blasius velocity field in Fig. 6 holds up until $x \approx 0.85$ where the transitional structures begin to dominate the physics of the flow.

Another indication of the compatibility with the Blasius boundary layer is given in Fig. 7 comparing the computed momentum thickness

$$\theta = \int_0^\infty \frac{U}{U_\infty} \left(1 - \frac{U}{U_\infty}\right) dy$$  \hspace{1cm} (12)

to that of the Blasius solution, namely,

$$\theta_b = 0.664Re^{-1/2}(x - 0.25)^{1/2}. \hspace{1cm} (13)$$

Here, the same virtual origin is used as in Fig. 6. According to Fig. 7, throughout the region where the Blasius boundary layer exists the agreement between computed and theoretical $\theta$ values is excellent.
Downstream of $x = 0.85$ as transition is entered, there is a rapid growth in $\theta$ beyond that of the Blasius result that is indicative of the thickening of the boundary layer that occurs during the transition to turbulent flow.

A plot of the average drag on the plate in Fig. 8 shows the drag crisis as the flow transitions into turbulence. Until $Re_x \equiv Re_x = 42,500$ corresponding to $x = 0.85$ where transition begins, the drag decreases somewhat in agreement with the Blasius boundary layer result $D = 1.328/\sqrt{Re_x}$. This is followed by a rise in the mean drag as the flow transitions to turbulence. It may be noticed that the values of $Re_x$ are somewhat lower than are ordinarily encountered in a transitioning Blasius boundary layer. This, apparently, reflects the relative coarseness of the vortex element representation that encourages a rapid response to disturbances. The use of greater numbers of vortex elements of lesser strength may counter this tendency. In any case, the degree of resolution provided in the present study is sufficient for the purposes at hand.
IV.B. Structural Aspects

The use of vortex elements in representing the flow field provides an alternative perspective with which to view the transition process out of which may come new understanding of its physics. Faintly visible in the vicinity of the center of Fig. 4 are the very gradual beginnings of undulations in the streamwise direction of the vortex filaments in the $x - z$ plane that are a precursor to the transitional structures appearing downstream. A similar effect has been observed in a similar place within transition in grid-based simulations where vortex lines have been reconstructed. Beyond the midplane in Fig. 4 distinct organized features aligned in the streamwise direction begin to be observed. These have some similarity in a local sense with the visualizations of spanwise and streamwise vorticity isosurfaces reported by Rist and Fasel in a DNS simulation of transition. Almost immediately after they appear, the streamwise disturbances to the vortex layer develop a spanwise structure as well. Each individual coherent event separated in the spanwise direction undergoes a subsequent breakdown into a more complex arrangement of vortex filaments and eventually merges with the neighboring vortical structures.

The initiation of streamwise bending of the vortex filaments in Fig. 4 conforms to the presence of alternating high and low perturbations in the streamwise velocity with the entire structure a reflection of an underlying instability to the vortex layer covering the solid surface. The degree to which vortices are perturbed forward and backwards out of the spanwise direction grows with downstream distance and evidently leads to liftup at the downstream pointing perturbations. Indeed, the favored rising of vortices is plainly visible in end-on views of the vortex field starting at $x \approx 0.8$ and becomes increasingly pronounced downstream. Fig. 9 shows the phenomenon at locations $x = 0.95, 1.05$ and 1.15 where just those vortices intersecting narrow spanwise cuts at these locations are plotted. The structures in this figure, viewed from above, correspond to the five clearly visible raised streamwise furrows to the right of center of Fig. 4, so it is clear that the latter are composed of a continuous sequence of uplifted vortices that form along the downstream direction. Taken together, the phenomenon has the character of a buckling of the vortex sheet along streamwise creases, at least up to the point where the individual structures merge into more complex interacting vortices at the far right of Fig. 4.

The appearance of low speed streaks in the transitioning boundary layer is a common occurrence under a relatively wide set of circumstances. In fact, they underlie the elongated streamwise features in Fig. 4 as is apparent from Fig. 10 in which contours of streamwise velocity on the plane $y = 0.0076$ corresponding to $y^+ \approx 22$ are plotted over roughly the same field of view. The low speed streaks in the
latter figure exactly coincide with the streamwise structures in Fig. 4 including extending to well within the region where the raised sections of vorticity have evolved to a more or less chaotic state. The spacing of the streaks, or equivalently the vortex furrows, are on the order of the boundary layer thickness (approximately 0.1 in this calculation) and thus in agreement with general observations about the properties of streaks in boundary layers.\textsuperscript{14, 25}

In the region covered by Fig. 9, the downstream increase in the size of the uplifted vortex features is rapid. A clearer view of this development is provided by Fig. 11 giving a view of the vortex tubes that intersect narrow planes at fixed spanwise positions. Specifically, the planes at $z = -0.0775, -0.1275$ and -0.1775 are shown where the middle plot is along the center of the most prominent of the uplifted vortices in Fig. 9, and the first and last of the images are to either side of it in regions that are nominally undisturbed by streamwise features. It is evident from these figures that something resembling vortex rollup forming spanwise aligned
structures occurs to varying degrees across the span. The presence of such spanwise coherent concentrations of vortices is also clearly evident in Fig. 4 where they are seen to form almost immediately with the first appearance of the streamwise folds.

Figure 11. Vortex tubes that intersect narrow streamwise cuts. (a), $z = -0.0775$; (b), $z = -0.01275$; (c), $z = -0.1775$.

It is clear from Fig. 4 that turbulent spots and the mechanisms that cause their appearance (e.g. free stream turbulence) are not in evidence in the present simulation. Rather, the transition mode here has the appearance of the Klebanoff-type\textsuperscript{15} that includes low speed streaks with their accompanying vortical structure that breaks down into turbulence. In this kind of transition, low speed streaks are formed within the streamwise legs of \Lambda-vortices. The downstream evolution of this structure generally results in a nested group or packet of hairpin-like, or arch-like vortices. Other terms for these vortices are horseshoe, or, possibly, omega-shaped hairpins or \Omega-vortices; we use all these names somewhat interchangeably in this paper. Further evolution of the boundary layer disturbances, that may include the appearance of additional kinds of structures, terminates in the fully turbulent regime.

It is helpful at this point to consider how the structural features that have been observed thus far (e.g. in Fig. 4) tie in with the traditional findings concerning the vortical make up of the early and middle stages of the transitional boundary layer. In fact, phenomena that are analogous to the latter kinds of structures and have the same dynamical effect are present in the filament simulations, though to make this connection requires reinterpreting the nature of the structures in terms of the full vortical field that is provided by the filament scheme. One point to make before proceeding is to note that while a number of other structures have been discussed as part of the transitional and turbulent boundary layers,\textsuperscript{27} such as vortex rings, chains of ring-like vortices and soliton-like coherent structures, these tend to be associated with the late transition and/or turbulent regime and will not be considered here.

The formation of concentrations of spanwise vortices with somewhat the appearance of rolled-up vorticity was previously noted in regards to the structure that is evident in Fig. 4. In fact, the idea that vortex roll-up of raised shear layers is responsible for the presence of one or more arch vortices has had substantial support in past studies.\textsuperscript{27,28} The same set of physics and reasoning is likely to have relevance to the present situation,
particularly where the spanwise roller-type vortices in Fig. 4 intersect the raised up vortex furrows. To firm up this connection it is necessary to show that the objects seen in the vortex simulation are one and the same as the objects identified as hairpin packets in other studies. In fact, Figs. 12 and 13 show the identical group of arched vortices viewed two different ways. In the first, the vortex filaments in a subset of the region previously discussed in reference to Figs. 9 and 11 are plotted in 3D view revealing the presence of at least three nested arch vortices of increasing elevation whose top parts form the three bulges between $x = 1$ and $x = 1.1$ in Fig. 11(b). What appears to be the same three objects in the same locations are also visible in Fig. 13 where this time the structures are marked using an iso-surface approach, often used in grid-based schemes, to locate $\Lambda$ and arch vortices. In this, negative iso-surfaces (in this case with magnitude $-40$) of the middle eigenvalue belonging to the matrix $S^2 + W^2$ are employed, where $S$ and $W$ are respectively the symmetric and anti-symmetric parts of the velocity gradient tensor $(\nabla u)_{ij}$. This quantity is singled out for its ability to represent local regions of low pressure associated with vortical structures. It should be noted that the implementation of Fig. 13 in the present context has been done by evaluating velocity derivatives from centered finite difference formulas applied to the velocities computed on a fine mesh. This explains the presence of some noise, particularly below the horseshoe structures.

![Figure 12. Close up 3D view of vortex filaments revealing arch vortices.](image)

Since the arch vortices straddling low speed streaks are often described as having “legs” in the form of counter-rotating streamwise vortices, it is expected that this kind of object should also be present in the filament simulation. While no such objects are plainly visible in Fig. 4, nonetheless this motion is present and, moreover, it is in precisely the location and context that it is expected to be, namely, along the twin edges of the raised furrows that connect up with the arched vortices. Thus, in fact, counter-rotating streamwise vortex motion is responsible for the uplifting of the vortex filaments forming the furrows. As it turns out, viewing the streamwise vortex motion in the present case using the eigenvalue method is made difficult by an excess of noise in the computation of the velocity gradient tensor. This is not a problem, however, for the simulation at $Re = 80,000$ as shown below, where, evidently, the signal is stronger.

An alternative route to finding the presence of streamwise oriented vortices is to plot just those vortex

16 of 30
American Institute of Aeronautics and Astronautics
Figure 13. Isosurfaces of the middle eigenvalue of $S^2 + W^2$ equal to -40.

tubes in the calculation that have a substantial stream-wise component. Such an image is given in Fig. 14 where only vortex tubes for which the condition $|x_2 - x_1| > 0.75|z_2 - z_1|$ is satisfied are plotted, where the subscripts 1 and 2 denote the ends of the tubes. Tubes with $y_2 - y_1 > 0$ are shown as blue while those with $y_2 - y_1 < 0$ are plotted as red. The figure shows the apparent presence of counter-rotating pairs of “leg” vortices that match up exactly with the arch vortices in the previous figures. These streamwise oriented structures sit exactly at the edges of the uplifted furrows in Fig. 4, or viewed another way, are coincident with the wall-normal tilted vortex tubes on either side of the uplifted structures in Fig. 9. This last connection has been noted previously.\textsuperscript{17,30} Finally, a quiver plot of the velocity vectors on a cut at $x = 1.1$ through the structure centered at $z = -0.1275$ is given in Fig. 15 that shows the clear presence of vortex motion bracketing either side of the uplifted vortex field. This counter-rotating swirling motion is visible on any cross plane cut taken along the length of the raised vortex elements and when assembled together forms a pair of streamwise vortices.

Besides the counter-rotating motion at the same position as the vertically oriented filaments seen in Fig. 15 there is a powerful ejection of fluid from close to the wall and a substantial sweeping of fluid toward the wall outside the uplifted vortices. The dynamics here tie in to traditional ideas of ejections and sweeps as is clear in Fig. 16 that combines the velocity vectors with contours of streamwise velocity. The region between the uplifted vortices is relatively slow moving fluid ejected from close to the wall that forms the low-speed streaks in Fig. 10, while regions of high speed fluid exist close to the wall having been swept to the surface on either side of the uplifted vortex structures. The similarity of these results with the findings of traditional grid-based LES\textsuperscript{18} is clear and adds further evidence that the present simulation offers up the same physics as traditional schemes.

An important conclusion that can be drawn from the discussion thus far is that the common idea that streamwise vortices exist as independent, and spatially self-contained entities within a background vorticity field, is in need of revision. In fact, the streamwise structures appear to be largely an artifact of the way in which they are defined in terms of the presence of rotational motion. A more precise description of these
Figure 14. Overhead view of vortex tubes satisfying $|x_2 - x_1| > 0.75|z_2 - z_1|$, $y_2 - y_1 > 0$, blue tubes; $y_2 - y_1 < 0$, red tubes.

Figure 15. Vortex tubes that intersect a narrow spanwise cut at $x = 1.1$ with superimposed velocity vectors.
Figure 16. Vortex tubes that intersect a narrow spanwise cut at \( x = 1.1 \) with superimposed contours of streamwise velocity.

structural elements is to say that they are the velocity signature of the edges of the streamwise oriented deformations to the buckled vortex sheet, that is, the streamwise vortices represent the collective effect of an elongated raised region of vortex lines.

This interpretation of longitudinal vortex pairs fits in naturally with the aforementioned idea that the arch-like nested vortex packets also owe their existence to the lifted vortex sheet. The vortex packets represent the intersection with the raised sheet of the rolled up or concentrated spanwise vorticity that has a tendency to form in shear layers. As is the case in mixing layers and jets, Figs. 4 and 9 suggest that the spanwise structure tends to have a significant transverse extent, often well beyond the limits of any particular arch-like vortex. This fits in with the idea that hairpin-like structures consisting of an arch vortex seamlessly connected to streamwise-oriented leg vortices do not generally exist as independent entities. The spanwise orientation of the vortex filaments at the sides of the arch vortices plotted in Fig. 12 provide additional evidence for this conclusion. The link between the apparent vortex legs and the arched vortices appears to be the fact that they are all associated with a specific wrinkle in the surface vortex sheet.

Alongside the continuous lines of low speed fluid beneath the vortex sheet furrows as shown in Fig. 10, are elongated regions of high speed fluid containing a suggestion of a spatially intermittent structure. A close up view of this is shown in Fig. 17 that contains streamwise and wall-normal velocity contours in the plane \( y = 0.0126 \) where \( y^* \approx 48 \) in the region surrounding the raised vorticity field at \( z = -0.1275 \). In this case, the clearly evident intermittent pattern in the \( V \) velocity contours exactly matches up with and may thus be associated with the concentrated spanwise vortices that are visible in Figs. 4 and 11. Thus, the sweeps that appear at a discrete sequence of downstream locations next to the ejecting flow reflect the action of the individual rolled-up spanwise vortices. A central cause of the sweep motion must also be attributable to the action of the wall-normal oriented sides of the arch vortices as is evident from Fig. 15. Fig. 17 also shows that the sweeps are interspersed with regions of slightly ejecting fluid, though beyond \( x = 1 \), where the furrow is more highly lifted, all the motion is wallward. It is interesting to note in Figs. 11(a) and (c) that the dominance of sweeps over ejections creates the appearance of a shear layer with orientation such
that the faster moving fluid is beneath the slower moving fluid — the opposite of what occurs, for example, at the center of the streak.

![Velocity contours on the plane $y = 0.0126$ where $y^+ \approx 48$ in the neighborhood of a fold in the vortex sheet: (a), $U$; (b), $V$.](image)

Some idea of the last stages of transition that ends with a fully turbulent field is provided by the image in Fig. 9(c) where now the uplifted vortices have deformed into an omega-shaped vortex. The presence of the latter in the boundary layer as a later stage in the evolution of hairpin packets has been observed in physical experiments and computations.\textsuperscript{15,31} Such objects continue to breakdown into the random field of vortex structure as seen in Fig. 4 as they interact with other nearby structures. One aspect of this may include the phenomenon by which hairpin vortices pinch off from the wall while their “legs” reconnect\textsuperscript{15,32} to form a new hairpin. However, to be consistent with the previous conclusion that vortex “legs” are an artifice of how the vorticity field is viewed, it is perhaps more appropriate to conceptualize this pinch-off reconnection process in terms of the dynamics of the wall vortex sheet. Thus, in this scenario, after the separation of an omega-shaped vortex from the wall layer, the uplifted vorticity along the furrow, through a roll-up process, recovers a local structure with the appearance of an arch vortex. The apparent “reattachment” of the latter with vortex “legs” is, in fact, a reflection of the common association of the vortex “legs” and the new arch vortex with the same streamwise deformation of the surface vortex sheet.

The previous discussion suggests that there may be some advantage to understanding the stability of the boundary layer and its transition to turbulence through consideration of the properties of a deforming vortex sheet over a solid surface. This may provide a cohesiveness to the discussion that is difficult to achieve via analysis of structural elements whose precise form is a matter of subjective interpretation. To some extent, such an approach is more in line with the common description of shear layers and jets in terms of their vortex behavior. It should be noted that some steps toward analyzing the stability of boundary layers through vortex sheets have been made previously.\textsuperscript{33} In particular, the stability of a streamwise corrugated vortex sheet — with some similarity to the central region of Fig. 4 — has been considered. This has predicted the appearance of streamwise vorticity within the evolving sheet. A next step may be to consider the further stability of the streamwise deformations within the sheet that grow with downstream distance and form the...
hairpin vortices. This view may be contrasted with the current practice of directly considering the stability of the streaks themselves.26,34

V. Turbulent Boundary Layer

The boundary layer simulations begin impulsively from initially quiescent conditions by imposing a unit velocity over the flow domain. Figure 18 shows the time history of the number of filaments, \(N_f\), and number of vortex tubes, \(N_t\), for the calculation at \(Re = 80,000\) that will be the basis for the following discussion. In this, the exit plane is held at \(x = 1\) until \(t = 0.94\) when it is increased to \(x = 1.25\). At \(t = 1.51\) the exit plane is moved back to the rear end of the plate at \(x = 1.5\) for the remainder of the computation. Test calculations suggest that this procedure speeds up the computation by allowing for a relative equilibrium to form upstream and then spread downstream, as against computing the transition to an equilibrium everywhere at the same time.

![Figure 18. Variation of \(N_f\) (lower curve) and \(N_t\) (upper curve) with time.](image)

Beginning at about \(t = 2.2\) the number of vortex elements and filaments stabilizes with approximately \(N_f = 2.7 \times 10^6\) and \(N_t = 22 \times 10^6\) signaling that global equilibrium has been reached. In this, production and destruction of tubes are in balance with the average filament containing about 8 vortex tubes. In fact, the filament population contains many with one tube that are relatively recently created and a significant number downstream containing hundreds of tubes. Flow statistics to be presented are based on the data from the start of equilibrium at \(t = 2.2\) until the end of the computation at time step 1000 which is at \(t = 3.2\), so that the elapsed time for computing averages is 1 dimensionless time unit. This proves to be mostly adequate for predicting the mean flow, but less than optimal for complete convergence of higher order statistics such as the Reynolds stresses.

A rendering of the complete vortex field as viewed from above at \(t = 2.2\) is shown in Fig. 19 and from the side in Fig. 20. As mentioned previously, the presence of the viscous rounded front to the plate causes relatively rapid transition leading to a fully turbulent field much further upstream than was seen previously in Fig. 4 for the inviscid front. In fact, transition here is now before the midpoint of the plate so that an extensive fully turbulent field is available for analysis. In terms of the average displacement thickness taken from the central part of the turbulent region in the figure, the computational test section is approximately 110 \(\delta\) in length and 37 \(\delta\) in width, a size that is of a magnitude similar to or larger than that of other studies.11
Figure 19. View from above of the vortex elements for the $Re = 80,000$ simulation with viscous front boundary.

Figure 20. View from side of the vortex elements for the $Re = 80,000$ simulation with viscous front boundary.
As in the case with the inviscid front, the flow that first reaches the plate must negotiate the rounded surface leading to an overshoot in the mean velocity near the leading edge. The effect, however, is much reduced for the viscous front surface and is not evident beyond $x \approx 0.2$. In this case, the external streamwise flow outside the turbulent wall region is raised to approximately 1.04 along the length of the plate. The gap between the vortices and the wall surface near the front of the plate in Fig. 20 gives an indication of where the outer edge of the wall mesh is located. As the flow evolves and becomes turbulent the vortices become distributed down toward the plate surface. The figure also reveals the corrugations in the outer edge of the boundary layer that generally represent the footprint of turbulent eddies formed from many individual filaments.

The transitional structures seen to the left in Fig. 19 have a similar appearance to those discussed previously in regards to Fig. 4. It is then not surprising that an application of the eigenvalue approach to finding structure as shown in Fig. 21, (used previously in producing Fig. 13), yields similar kinds of results as before. In this case, for the central, left-most region in Fig. 19, the two prominent streamwise perturbations are seen to have the familiar hairpin-packet structure that was discussed previously. The streamwise oriented vortex pairs that correspond to the raised vortex folds are visible, and, in fact, are positioned as expected along their respective sides. Downstream from this, Fig. 19 gives a clear indication of how the continued growth and breakdown of the streamwise disturbances, in parallel with the evolution of strong spanwise oriented roller-type vortices, results in a fully turbulent field. There appears to be at least some qualitative similarity between this process and its manifestation in mixing layers and jets. Finally, it is evident that there is a noticeable presence of relatively large scale organized eddies in the fully turbulent field that mirror in size those in the late transition.

To give a context for viewing the current simulation of the turbulent boundary layer in comparison to other studies, Fig. 22 gives a plot of the computed trend in $R_\theta = U_\infty \theta / \nu$ along the streamwise extent of the plate. There is rapid growth in $R_\theta$ (or, equivalently, $\theta$) from the leading edge that slows at $x = 0.35$ rising
relatively uniformly until $x = 1$ where there is a rapid drop until the end of the plate. The latter effect is a direct result of the relatively crude downstream boundary condition and the presence of an acceleration of the flow over the rear end of the body. The initial part of the boundary layer is strongly affected by the rounded front end so that acceptable data for analysis must come from the central region only. For the purposes of the present study, the turbulent boundary layer between $x = 0.7$ until the peak in $R_\theta$ at 1.0 is considered to be a test section used to acquire flow statistics. Velocity data is computed on a mesh covering this region with averaging done in the spanwise direction at fixed $x$ locations. For most of the following discussion averaging is expanded to include the data throughout the test section by connecting values using a commonly scaled $y$ coordinate, such as $y^+, y_\delta = y/\delta$ or $y_\theta = y/\theta$. The mean value of $R_\theta$ over the test section is 737 and this value will be associated with the flow statistics averaged over the same region.

![Figure 22. Variation of $R_\theta$ across the plate.](image)

A traditional semi-log plot of the average streamwise velocity scaled in wall units is shown in Fig. 23 together with the mean field at $R_\theta = 670$ computed by Spalart. Taking into account the slightly larger $R_\theta = 737$ of the present simulation, the quantitative agreement in the far field velocity is excellent as is the qualitative behavior including the presence of a log-type law and $U^+ = y^+$ trend near the wall. The fitted log law has intercept $B = 5.12$ and Kármán constant $k = 0.383$ with the fit done by least squares over the region $30 \leq y^+ \leq 80$. These values are within the range of variation seen in experiments and may also possibly reflect the existence of subtle differences between the plate flow and a true zero-pressure gradient boundary layer.

To enable a more direct comparison with DNS the normalized mean velocity $U/U_\infty$ as a function of distance scaled in wall units is plotted in Fig. 24. There is little difference between the solutions from this point of view. Two other alternatives for scaling the wall normal direction are with the displacement thickness $\delta$ and the momentum thickness $\theta$. As shown in Fig. 25 the present simulation and the $R_\theta = 670$ solution compare extremely well from this viewpoint. Overall the results in these several figures give an indication of the inherent accuracy and physicality of the vortex filament prediction of the mean velocity in the turbulent boundary layer.

Some idea of how the present computation fits in with results from a wider range of experiments is provided by Fig. 26 showing the predicted friction coefficient and the shape factor which is defined as the ratio $\delta/\theta$. The results for the vortex filament calculation are given from the position $x = 0.125$ through to $x = 1$ over which range $R_\theta$ monotonically increases from 380 to 790. In regards to Fig. 26 (a), below $R_\theta = 670$ the friction coefficient angles off toward the Blasius result generally following a trend similar to that reported in other studies. The computed results for $R_\theta > 650$ are within the fully turbulent test section, and for these $c_f$ fits in closely with data taken from a variety of sources. Much the same conclusions can be drawn in the case of the shape factor that is well inside the range of data from other studies in the
Figure 23. Semilog plot of $U$ (solid line) compared to $R_\theta = 670$ DNS solution. Dashed lines represent $U^+ = y^+$ and the fitted log law $U = 1/0.383 \log(y^+) + 5.12$.

Figure 24. Comparison of $U/U_{\text{max}}$ in terms of wall variables. —, vortex filament scheme; ○, $R_\theta = 670$ DNS solution.
turbulent regime above $R_\theta = 670$ and branches upward to the laminar flow result below this.

As mentioned previously, the length of time completed in the simulation of the $Re = 80,000$ boundary layer is not long enough to expect fully converged predictions of higher order moments such as the Reynolds stresses. Nonetheless, it is of some interest to examine the basic trends in these quantities and see how they fit in with prior computations. Results for the normal Reynolds stresses compared to DNS\textsuperscript{11} at $R_\theta = 670$ and 1410 are shown in Fig. 27. These generally have the correct qualitative behavior with the principal exception being the absence of the distinctive peak in streamwise Reynolds stress near the wall. The magnitude of the Reynolds stresses appear to be somewhat over predicted, with the behavior of the spanwise Reynolds stress near the wall, in particular, indicating the need for greater resolution in the wall mesh. In the outer flow the Reynolds stresses appear to have magnitudes that are more aligned with the $R_\theta = 1410$ computations than the data taken from the simulation with the more similar Reynolds number.

The predicted Reynolds shear stress $\overline{uv}$, shown in Fig. 28 (a), is seen to have properties consistent with Fig. 27 including especially an overestimation of $\overline{uv}$ that matches that of the normal stresses. The second figure in 28 displays the scaled Reynolds shear stress which is seen to have some qualitative consistency with DNS, with the greatest distortion showing up next to the wall. The far field trend in this case follows that of the more appropriate of the two DNS data sets.

To some extent the Reynolds stress results in Figs. 27 and 28 reflects the bias that comes with sampling over a relatively short time interval. Other factors influencing the comparisons are the coarseness of the near-wall grid and differences between the plate flow and a zero-pressure gradient boundary layer including the outward acceleration at the leading edge and the vortex removal at the end of the plate. An additional consideration that is known to cause over-prediction of the Reynolds stresses\textsuperscript{4, 5} is the maximum length of the vortex tubes. Though this is kept to the relatively small value of 0.01 in this work, it is still larger than the magnitude of 0.005 that was recommended in an earlier study.\textsuperscript{4} It is interesting to note that whatever discrepancies appear with the Reynolds stresses as they have been computed thus far, it does not appear to translate into a noticeable loss of accuracy in computing the mean velocity field. It may be surmised that this is consistent with the observation\textsuperscript{36} that from one boundary layer experiment to another, the Reynolds stresses tend to show a wider range of variability than does the mean velocity.
Figure 26. (a) friction factor, and (b) shape factor vs. $R_\theta$. $\circ$, current simulation with $Re = 80,000$; $+$, DNS$^{11}$; $\cdots$, DNS$^{14}$; $-$, $\times$, $*$, various experiments.$^{11,36}$

Figure 27. Normal Reynolds stresses in $Re = 80,000$ simulation. $-$, filament computation; $\circ$, DNS$^{11}$ $R_\theta = 670$; $+$, DNS$^{11}$ $R_\theta = 1410$. 

American Institute of Aeronautics and Astronautics
VI. Conclusion

A hybrid vortex filament/finite difference and finite volume scheme has been developed with a view toward efficiently and accurately accounting for the physics of turbulent flow in the context of a large eddy simulation. Vortex filaments offer a succinct means of representing essential turbulent vortical structure away from boundaries with minimal numerical diffusion and with the opportunity to incorporate loop-removal as a physically motivated “sub-grid model” for regulating energy flow between large and small scales. A finite difference and finite volume scheme for solving the full viscous flow equations on a thin boundary mesh supplies new filaments to the calculation while respecting the need for high resolution in the wall region where the idea of a LES is suspect.

For the simulation of the transitional boundary layer it was found that beyond agreeing with prior studies in a number of basic quantitative and qualitative features, the filament methodology had the advantage of providing a somewhat more holistic view of the structural elements of the vorticity field than has generally been possible with other techniques. From this has come an apparent need to reinterpret the meaning of some of the prominent structural aspects of transition that have been identified to date. For example, the streamwise oriented counter-rotating fluid motion that is usually described as being the “legs” of hairpin vortices was found to be more accurately categorized as the flow naturally associated with the edges of raised furrows in the deformed vortex sheet over the solid boundary. Moreover, the computations showed how arch-type vortices produced during transition can be understood as representing the intersection of raised streamwise vortex furrows with concentrated or rolled-up regions of spanwise vortex filaments. Thus the connection between arch and leg vortices lies in their common association with raised streamwise creases in the surface vortex sheet. In particular, it may be imagined that newly created arch vortices do not so much acquire new vortex “legs” as share in the structure of the raised vortex furrow that they are associated with.

Beyond these considerations, there may be advantages to investigating the stability of the boundary layer directly in terms of the properties of the wall-layer vortex sheet. In particular, the stability of the raised spanwise vortices forming streamwise features should be relevant to understanding the occurrence of streak breakdown and the last stages of transition leading to turbulent flow. One may also move beyond the transition mode observed in this work to speculate that similar processes as have been describe here also underlie the local behavior of turbulent spots as well as those places within a fully turbulent boundary layer where coherent packets of hairpin vortices and their associated streamwise vortices have been noted.
Simulation of the fully turbulent boundary layer revealed that the methodology was fully capable of capturing the essential properties of the mean velocity field including an appropriate log law, friction coefficient and shape factors. An examination of the Reynolds stresses suggested that greater wall resolution and possibly shorter vortex tubes than were employed in the present simulation will be necessary to bring these quantities closer in line with experiments. Similarly, future work may need to be devoted more directly to simulating a classical zero-pressure gradient boundary layer to remove the possible influence of the leading and trailing edges of the plate flow on the computed details of the Reynolds stresses.

Acknowledgments

This research was supported in part by the National Science Foundation through TeraGrid resources provided by the Pittsburgh Supercomputing Center.

References


