Structure and Energetics of a Turbulent Trailing Edge Flow

T. G. THOMAS, Y. F. YAO AND N. D. SANDHAM
Aeronautics and Astronautics, School of Engineering Sciences
University of Southampton
Highfield, Southampton, SO17 1BJ, U.K.
(Received April 2001; accepted July 2001)

Abstract—Results from a direct numerical simulation (DNS) of turbulent flow over a rectangular trailing edge geometry are used to study energy transfers in detail. The numerical procedure employs a separate boundary layer simulation which is used to generate inflow turbulence and provide the inflow boundary condition data for the trailing edge flow simulation. The calculations are performed at a Reynolds number of 1000 based on the trailing edge thickness and free stream velocity. Characteristics of the vortex shedding are identified from instantaneous flow visualizations, two-point time correlations, and spectra. Turbulence kinetic energy production is found to be negative in the region behind the trailing edge, in agreement with simple analysis. The topology of the turbulence kinetic energy flux field is studied in conjunction with the production and dissipation, revealing pathways by which turbulence energy is transported from zones of creation to zones of destruction. © 2003 Elsevier Ltd. All rights reserved.

Keywords—Turbulent flow, Trailing edge, Energy balance, Vortex shedding, Direct numerical simulation.

1. INTRODUCTION

Turbulent flow past a finite-thickness rectangular trailing edge is a difficult problem to model computationally because the sudden removal of the solid boundary surface allows the upper and lower boundary layers to interact and the presence of a finite-thickness step creates an attached recirculation bubble. The bubble is unstable and the flow oscillates in similar way to a bluff-body wake to produce a von Kármán vortex street that extends downstream. However, the flow differs from the usual vortex street in several ways. First, the vortex shedding is not strictly periodic in time and is not highly correlated along the spanwise direction. Second, the equilibrium structure of the turbulence in the approaching boundary layers is dramatically modified as it passes through the highly sheared oscillating flow just behind the trailing edge and into the wake. There is a balance between additional turbulence being produced by the oscillating component of the flow, and the destruction of the coherent oscillations by the increased levels of turbulence. This unsteady balance is difficult to capture using conventional turbulence models, but can be treated naturally by direct numerical simulation techniques. The flow system has the characteristics of a forced oscillator with relatively strong coupling between the large-scale and small-scale motions.
Several experimental and numerical studies have been conducted recently to investigate the trailing edge flow problem. Gough and Hancock [1] carried out an experimental study using a pair of symmetric incoming turbulent boundary layers, each with $\delta_{99}$ at the trailing edge equal to 10 times the trailing edge thickness $h$ and a Reynolds number based on the boundary layer momentum thickness $\theta$ of 600. For this flow, it was found that a mean recirculation region existed behind the trailing edge and a coherent von Kármán vortex street existed in the near wake and extended downstream to about $10h$, after which it was scrambled by the presence of small scale turbulence. The same flow was investigated computationally by Gao et al. [2] using the large eddy simulation (LES) technique. Their calculations showed reasonable qualitative agreement with experimental data for the turbulence statistics, but quantitative differences remained which were attributed to the differences in the inflow boundary layers between the experimental and numerical arrangements. It is believed that the upstream incoming flow has a significant effect on the downstream flow field, and therefore, that an accurate simulation of the inflow turbulent boundary layer is a necessary requirement for performing an accurate trailing edge flow simulation.

The present DNS calculation is arranged as shown in Figure 1. A boundary layer (precursor) simulation is used to generate the inflow data for the following trailing edge (successor) simulation. By sampling the precursor simulation at widely separated times, two statistically independent inflow streams can be produced. The Reynolds number $Re = U_e h/\nu$ is set equal to 1000; here, $U_e$ denotes the free stream velocity, $h$ the plate thickness, and $\nu$ the kinematic viscosity.

In a previous paper [3], we have presented DNS results for this flow, including turbulence statistics and some energy budgets. In this contribution, we refine the analysis of the turbulence energetics and include a novel method of presenting turbulent flux information.

![Turbulent boundary layer simulation](image)

![Trailing edge simulation](image)

Figure 1. Definition sketch showing the arrangement of the turbulent boundary layer and trailing edge simulations.

2. NUMERICAL METHOD AND SIMULATIONS

The Navier-Stokes equations are discretized on a staggered grid using second-order finite-differences, and advanced in time by the projection method based on the second-order Adams-Bashforth scheme. The provisional velocity is projected using data from the current and previous time steps but not including the contribution from the pressure terms. The velocity at the new time step must satisfy continuity and requires a pressure correction. This is provided by the solution of a Poisson equation for the pressure.

The complete simulation has two separate parts. The boundary layer calculation runs first, followed by the trailing edge simulation. The turbulent boundary layer flow is computed using a spatial DNS code, using a second-order finite-volume method with a direct Poisson solver that makes use of fast Fourier transforms in two spatial directions. The technique of Lund et al. [4]
is used to rescale the turbulence from near the outflow and feed it back as the inflow condition. The method has been validated with laminar and turbulent boundary layer flows and results compare well with experiment and other simulations [3]. A time sequence of flow variables at one plane is stored for later use as the inflow boundary condition in the trailing edge flow simulation. The latter simulation is carried out with a complex-geometry code which uses the same finite-difference scheme as for the precursor simulation, but this time with a multigrid Poisson solver. The code is parallelized based on the concept of constructing the computational domain with an assembly of rectangular blocks with their faces mapped either to each other or to boundary conditions. It is implemented in C/C++ and uses MPI parallel message libraries. A parallel multigrid algorithm is used for solving the Poisson equation, details of which can be found in [5]. The code has been validated for different cases such as the backward facing step flow, open channel flow, and flow over a ground-mounted cube, all giving good agreement with published computational and experimental data.

The boundary conditions for the trailing edge DNS are defined as follows. At the inlet plane, we apply the inflow data interpolated from the boundary layer simulation sampled database. The upper and lower parts of the inflow are taken from a single simulation, and are thus, statistically identical, but with a sufficiently large lag to be uncorrelated in time. The outlet boundary conditions are treated with a convective condition, so that outgoing disturbances leave the domain smoothly. On the upper and lower surfaces, free-slip conditions are used and in the spanwise direction a periodic condition is adopted.

The computational box has size $20h \times 16h \times 6h$ in the streamwise ($x$), wall-normal ($y$), and spanwise ($z$) directions, respectively. The length in the streamwise direction $L_x$ includes $5h$ along the flat plate and $15h$ in the wake region behind the trailing edge to allow for the development of the incoming turbulent boundary layer and the wake. The computational domain is deliberately chosen to extend some distance upstream of the trailing edge, thus permitting the upstream boundary layer flow to be influenced by the motions in the downstream wake flow. Hence, it is possible to follow the natural progress from a boundary-layer flow to a wake flow. In the wall-normal direction the box length ($L_y$) is $16h$, with $8.5h$ in the upper and $7.5h$ in the lower part of the wake. Compared to the inflow turbulent boundary layer thickness $\delta_{99}$, which is $6.42h$ in the present simulation, the length is large enough to develop the incoming boundary layer along the trailing edge plate. The computational domain is decomposed in three directions, resulting in 256 blocks with four blocks along the trailing-edge geometry and 252 blocks in the flow field. The grid points are uniformly distributed with a grid of $256 \times 512 \times 64$. In each block, there are $64 \times 32 \times 16$ grid points and a four level multigrid sequence is constructed. This decomposition allowed 32 grid points to be placed across the thickness of the trailing edge. The simulations are carried out on a Cray T3E-1200 parallel computer.

The general features of the wake development can be determined from the mean flow field. A recirculation region extends to a point about $2h$ downstream of trailing edge and has a maximum reverse velocity of about 10% of the free-stream velocity ($U_e$). The mean velocity $U$ recovers to 50% of $U_e$ at about $10h$ downstream of the trailing edge and 55% of $U_e$ near the outlet plane.

In the boundary layer region ($2h$ upstream of the trailing edge), the grid spacing in wall units is $\Delta x^+ = 4.0$, $\Delta y^+ (1) = 1.6$, and $\Delta z^+ = 4.8$. Comparing with Spalart and Leonard [6] (who used a spectral method and $\Delta x^+ = 13$, $\Delta z^+ = 6.5$), the current simulation (with a finite difference method) has about three times as many grid points in the $x$ direction and about 40% more in $z$. In the wake region, the appropriate velocity scale is the mean velocity defect $\delta U_m$, rather than the wall shear velocity, and the grid resolution is represented in wake units defined by $\Delta \tilde{x}_i = \Delta x_i \delta U_m / \nu$, where $\Delta x_i$ is the grid spacing as before. Comparing against the plane far-wake DNS of Moser et al. [7], in which $\Delta \tilde{x} = 97.7$ and $\Delta \tilde{y} = 82.1$, we have $\Delta \tilde{x} = 83.3$ and $\Delta \tilde{y} = 33.5$ at locations in the recirculation region. Thus, the present resolution is comparable or better, depending on the direction, but that our finite-difference scheme is only second-order accurate. Overall, we believe that the simulations are numerically well converged, and this conclusion is
3. VORTEX SHEDDING

Visualizations of instantaneous flow structures in the near wake region reveal the basic structure of the flow. A plan view together with time series of five snapshots of the flow over a shedding cycle are shown on Figure 2a–2f. The darker colour marks a surface of constant pressure enclosing a low pressure region, while a light colour marks a surface of constant negative second invariant of velocity gradient $II$ defined by

$$II = \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i},$$

where $u_i$ denotes the instantaneous velocity along the $x_i$ coordinate. Plotting multiple surfaces in this manner helps to identify the different aspects of the flow structure. In particular, smaller scale structures are revealed by the second invariant plot. The interaction between low pressure structures and smaller scale strained vortices is shown quite clearly. The low-pressure structures are supported in the much more detailed examination of the effects of changing the domain size and grid resolution presented in [3].
are shed from the trailing-edge as quasi-two-dimensional structures and arrange themselves in a staggered manner similar to the von Kármán vortex street (the fluid rotates clockwise around the structures on the upper side of the wake and anti-clockwise for structures on the lower side). Incoming quasi-streamwise vortices, which are an inherent part of the boundary layer flow, are strained and deformed when they encounter these low pressure structures. The maximum straining occurs at the location of the (free) stagnation points positioned between successive low pressure cores; here the axial vortices are enhanced along the direction of maximum stretch. At the same time, the von Kármán vortices are perturbed and eventually break up due to the action of the strengthened small-scale structures.

Quantitative data concerning the vortex shedding behaviour has also been obtained. Figure 3a shows a time-separated autocorrelation of the integrated vertical force due to pressure imbalance on that part of the splitter plate which lies within the computational domain. A characteristic shedding period of about $10h/U_e$ is found. It should also be observed that the shedding is not completely regular since successive peaks in the autocorrelation become smaller for larger times. Figure 3b shows a power spectrum. The Strouhal number of the shedding is 0.118 with harmonics of this also present in the signal. This compares well with the value of 0.12 found in the [1] experiment.

Figure 3. Vortex shedding behaviour as indicated by the unsteady lift force: (a) Autocorrelation in time; (b) power spectrum (arbitrary units) showing a peak at a Strouhal number $St(fh/U_e) \approx 0.1$.

4. ANALYSIS OF TURBULENCE ENERGETICS

4.1. Kinetic Energy

Contours of the turbulence kinetic energy $K$, defined by

$$K = \frac{1}{2} u'_k u'_k \geq 0$$

are shown on Figure 4. Previous work [3] has presented conventional budgets for $K$ at different downstream positions in the flow. A particular feature identified was the spatial variability of the pressure transport term, which changes sign during the recirculation region and is of comparable importance to the largest terms in the budget in that region. In the following sections, we present a more complete analysis of the production, dissipation, and flux terms governing the turbulence energy.
4.2. Production and Dissipation

The turbulent production term $P_r$ gives the rate at which energy is extracted from the mean motion by work against the Reynolds stress.

$$P_r = -\bar{u}_i \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j},$$

(3)

It is often taken as axiomatic in the eddy viscosity modelling community to assume that any modelled Reynolds stress must lead to $P_r \geq 0$, similar to the Clausius-Duhelm inequality for the dissipation, so that the energy exchange between the mean flow and the turbulence is strictly one way. However, it can be shown that for certain classes of flows that regions with $P_r < 0$ must exist. A simple example occurs in the present configuration on the centreline just behind the trailing edge. Here it can be shown, using symmetry of the Reynolds stress on the centreline and the asymptotic variation of the fluctuating velocity near to a solid boundary, that

$$P_r = (-1/2) \frac{\partial \bar{u}_i}{\partial x_i} = \beta^2 x^2 \frac{\partial \bar{u}_i}{\partial x_i} + O(x^4),$$

(4)

where $\beta$ denotes a nonzero constant. We follow the usual convention that the fluctuating velocity $u_i$ has components $(u, v, w)$ along the streamwise ($x$), normal ($y$), and spanwise ($z$) directions, respectively. The reverse flow region imposes $\frac{\partial \bar{u}_i}{\partial x_i} < 0$, and hence, there must be at least some small region at the trailing edge where $P_r < 0$. In fact, it turns out that this region is more extensive than the above argument would suggest. Figure 5 shows the distribution of the $P_r$. There is a region of negative production immediately behind the training edge in the region of reverse flow, extending to about $x = 1.5h$ with a pair of local (negative) peaks at about $x = 0.9h$, $y = \pm 0.4h$. In this region, the Reynolds stress transfers kinetic energy from the turbulent fluctuations to the mean flow. Further downstream there are two large positive peaks at $x = 2.2h$, $y = \pm 0.5h$.

Figure 6 shows the distribution of dissipation $\varepsilon$, defined by

$$\varepsilon = \frac{1}{Re} \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \geq 0,$$

(5)

which is strictly positive for nonrigid body motion. It peaks locally on the solid boundary but with larger peaks just behind the trailing edge and also downstream at about $x = 2.5h$, close to where $K$ is a maximum. The structure of the dissipation is, however, significantly different from that of the production. For the remainder of the paper, we study the mechanisms by which energy is transported through the flow, passing from regions of production to regions of dissipation.
4.3. Kinetic Energy Transport

The transport terms differ from the production and dissipation terms because they move turbulent kinetic energy from one place to another and do not alter the total amount of $K$ in the system. Let $J_i$ denote the vector flux of $K$, so that conservation of turbulence kinetic energy requires that

$$\frac{\partial J_i}{\partial x_i} = P_r - \epsilon.$$  \hspace{1cm} (6)

The divergence of the flux $J_i$ thus gives the local imbalance between the production and dissipation of $K$.

The flux $J_i$ is composed of a mean convective flux $\bar{u}_i K$ plus partial fluxes due to velocity triple moments, pressure-velocity, and viscous diffusion, i.e.,

$$J_i = \bar{u}_i K + J_i^u + J_i^P + J_i^\nu.$$  \hspace{1cm} (7)

where the partial fluxes are given by

$$J_i^u = \frac{1}{2} \bar{u}_i \bar{u}_j \bar{u}_j, \quad J_i^P = p' \bar{u}_i, \quad J_i^\nu = -\frac{1}{Re} \left( \frac{\partial K}{\partial x_i} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} \right).$$  \hspace{1cm} (8)
By translational symmetry along $z$ (spanwise), $J_i$ has nonzero components only in the $(x,y)$ plane. There is also a mirror symmetry of the problem in the $y$-direction. Energy is transported through the flow along lines tangential to $J_i$. The topology of the flux lines may therefore be considered as central to the energetics of the turbulence.

A number of points concerning the interpretation of the flux lines follow immediately. There can be no flow of energy across the flux lines, and the total energy flow in the region between any pair of lines must be conserved, thus the amount of $K$ entering or leaving the region at its boundaries is balanced by the total $P_r - \varepsilon$ within the region. Here, the solid boundaries may be considered either as distributed critical points, where the flux vanishes, or as bounding flux lines on which the flux happens to vanish. If the flux lines are arranged to form a closed loop or circuit, or lines diverge from one nodal point and converge on another, then the region must contain regions of both creation $P_r > \varepsilon$ and destruction $P_r < \varepsilon$. If the lines diverge from a node, then the existence of a nonzero flux implies that a diverging node must be located in a region where $P_r > \varepsilon$. A similar argument shows that converging nodes must be located in regions where $P_r < \varepsilon$. In the absence of the combined $P_r - \varepsilon$ term, the total energy flux between a pair of flux lines must be constant along the lines, and thus the appearance of diverging or converging lines implies a corresponding decrease or increase in the local flux density.

With this physical picture in mind, we may now examine Figure 7, which shows the flux lines tangential to $J_i$, along which the turbulent kinetic energy $K$ flows, along with contours of the $P_r - \varepsilon$ imbalance. The topology, sketched in Figure 8, has been extracted from enlarged versions of Figure 7 and can be described as follows. The rear nodal point of the flux vector $J_i$ is a saddle-node $S_1$ located on the centreline at $x \approx 1.85h$, and slightly upstream of the recirculation nodal point at $x \approx 2h$. There is a diverging focus $F_1$ located at $(x, y) \approx (1.05h, \pm 0.4h)$ and a second saddle-node $S_2$ located at $(x, y) \approx (0.7h, \pm 0.4h)$. All the flux lines within the region enclosed by the flux line that meets $S_1$ terminate at either the origin or on the solid boundary (equivalent to nodal point $N_1$). It follows that as $K$ flows out of $F_1$, this point must be located in a region of positive $P_r - \varepsilon$, and as these lines terminate on the boundary, they must also pass through a region of negative $P_r - \varepsilon$. The largest values of $K$ occur at a double peak located at $(x, y) \approx (2.3h, \pm 0.4h)$ outside of the recirculation bubble and downstream of the region of maximum $K$ production.

The contributions to the total flux term can be analysed individually. It turns out that the viscous terms in equation (7) are significant only immediately adjacent to the solid boundary and are negligible elsewhere in the wake. In Figure 9, the structure of the convective term in equation (8) is shown. The flux lines for this case are simply the streamlines of the flow. Positive values of the divergence of this contribution to the total flux are located in the shear layers away from the centreline. Negative values are located on the centreline in the recirculation region.

![Figure 7: Transport of turbulent kinetic energy $K$ along the flux lines $J_i$ (shown with arrows). The superimposed contours show the difference between the local creation and destruction of $K$, indicated by the divergence of $J_i$, $(J_{i,i} \equiv P_r - \varepsilon)$. Contours at (positive) 0.001, 0.002, ...; (negative) -0.005, -0.001, ...](image)
Figure 8. Topology sketch.

Figure 9. Topology of the convection contribution to flux (arrowed lines). The superimposed contours show the divergence of the convection (contribution to total flux). Contours at (positive) 0.001, 0.002, ...; (negative) −0.0005, −0.001, ....

Figure 10. Topology of the velocity triple product contribution to flux (arrowed lines). The superimposed contours show the divergence of this flux (contribution to total flux). Contours at ±0.001, ±0.002, ....

Figure 11. Topology of the pressure-velocity contribution to flux (arrowed lines). The superimposed contours show the divergence of this flux (contribution to total flux). Contours at ±0.001, ±0.002, ....
Figures 10 and 11 show the same diagram for the triple moment and pressure-velocity correlation flux terms. The topology of these is very different. Compared to the pressure term, the triple moment has an additional focus at $x = 1.5h$, $y = 0.4h$. The pressure term can be seen to have a saddle structure upstream of the trailing edge. This supports the initial decision to include a significant portion of the splitter plate within the DNS calculation. It is interesting to compare the observed structure of these last two terms with the structure that would arise from the assumption of gradient diffusion, as is commonly carried out in turbulence modelling. In a gradient diffusion model, the flux lines would be everywhere normal to the $K$ contours shown in Figure 11 and have critical points at location where the $K$ gradient vanishes. Downstream of $x = 3h$, this is qualitatively correct. However, around the recirculation region, where large scale vortex motions dominate, large errors are apparent and there is evidence of counter-gradient diffusion. It may be helpful in the future to develop models for turbulence transport that are topologically consistent with the real flow.

5. CONCLUSIONS

An analysis of the energetics of turbulent flow past a blunt trailing-edge has revealed a number of key features. First, the production is found to have negative values over a significant portion of the recirculation region. The necessary existence of such a region has been demonstrated by a simple asymptotic analysis. A novel topological analysis of the turbulent flux vector has then been used to study energy flows through the complex flowfield. Energy flux lines show the direction of turbulent transport of energy through the flow. Energy is added or subtracted as the lines pass through regions with an imbalance of production and dissipation. The regions where modelling assumptions break down are the regions of strong activity of coherent vortex structure, illustrated by visualization of the instantaneous flow field.

REFERENCES