[8]Couaillier, V., "Numerical Simulation of Separated Turbulent Flows Based on the Solution of RANS/Low Reynolds Two-Equation Model," AIAA Paper 99-0154, Jan. 1999.

[9]Menter, F., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.

[10]Jones, W., and Launder, B., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," *International Journal of Heat and Mass Transfer*, Vol. 15, 1972, pp. 301–314.

[11]Cousteix, J., Houdeville, R., and Javelle, J., "Response of a Turbulent Boundary Layer to a Pulsation of the External Flow with and Without Adverse Pressure Gradient," IUTAM Symposium, International Union of Theoretical and Applied Mechanics, Toulouse, France, May 1981.

[12]Benoit, B., and Legrain, I., "Buffeting Prediction for Transport Aircraft Applications Based on Unsteady Pressure Measurements," AIAA Paper 87-2356, Aug. 1987.

R. M. C. So
Associate Editor

Onset of Condensation in Vortical Flow over Sharp-Edged Delta Wing

Satoru Yamamoto*
Tohoku University, Sendai 980-8579, Japan

Introduction

THE actual atmosphere of the Earth includes a finite amount of water vapor, which plays an important role in weather conditions and in the Earth's various environments. Water vapor also plays an interesting role in the flight of airplanes. Water vapor may occasionally condense around the airplane. The phase change may be generally dominated by heterogeneous nucleation of water vapor, because small particulates such as soot or aerosols may behave as a nucleus of condensation.

The condensation around an airplane is the so-called vapor trail. Vapor trails are occasionally observed in the takeoff, landing, and transonic cruising of airplanes. A vapor trail appears as a white cloud over the wing or extending from the wing tip. Campbell et al.[1] summarized a number of photographs of natural condensation.

Because time and space scales in a wind tunnel are fundamentally smaller than those in flight, actual condensation in flight may not be recreated by a flow around a small-scaled wing in the wind tunnel. In atmospheric wind-tunnel conditions, the supersaturation ratio S of water vapor may quickly go beyond the saturation ratio $S = 1$ due to homogeneous nucleation through the inlet of the wind tunnel; it reaches $S \gg 1$ rapidly around the wing without condensation. After this process, a huge amount of water droplets is nucleated in a highly supersaturated condition. This process is dominated by nonequilibrium condensation. As a result, condensation increases temperature and pressure due to the release of the latent heat of water vapor. In atmospheric flight conditions, on the other hand, the chord length of the wing is much longer than that of wind-tunnel models. Furthermore, small particulates, in the atmosphere, such as dust, soot, ions, and aerosols, may play the role of the nucleus for heterogeneous processes. Therefore, actual condensation in flight may be dominated by almost equilibrium condensation.

Recently, Schnerr and Dohrmann[2] studied two-dimensional transonic flows of moist air around an airfoil in atmospheric wind-tunnel conditions experimentally and numerically. Transonic viscous flows of moist air around a NACA0012 airfoil at 2-deg angle of attack in atmospheric wind-tunnel conditions were studied by Iriya et al.[3] Three-dimensional transonic turbulent flows around the ONERA M6 wing were calculated by Yamamoto et al.[4] assuming atmospheric wind tunnel conditions. In this study, the computational code is applied to the calculation of condensation in a streamwise vortex produced over a single delta wing in atmospheric wind-tunnel and flight conditions.

Low-speed flows over a delta wing or a double delta wing at high angle of attack have been widely studied by Hsu et al.,[5] Fujii and Schiff,[6] and Gordnier and Visbal.[7] It is known that a strong streamwise vortex is produced from the apex of the delta wing. All delta-wing studies that we know of have assumed ideal gas without water vapor (zero humidity). However, condensate streamwise vortices are occasionally observed around the delta wing in actual flight.[1] Although condensation is known as quite an important phenomenon affecting wing performance, the physics of condensation in the streamwise vortex over the delta wing is still unknown. Therefore, a study to understand the mechanism underlying the phenomenon would be quite valuable even if it is done at a preliminary level. We provide a preliminary numerical result for the condensation in a streamwise vortex around a delta wing in this Note to further the study of condensate flows in atmospheric flight conditions. As a numerical example, three-dimensional subsonic laminar flows over the 76-deg, sharp-edged single delta wing without thickness at 20.5-deg inclination are calculated under varying inlet or freestream Mach numbers and atmospheric flow conditions, and the onset and the rate of condensation in each flow condition are compared.

Fundamental Equations

The fundamental equations for three-dimensional compressible laminar flows of moist air consist of conservation laws of the total density, the momentums, the total energy, the density of water vapor, the density of liquid water, and the number density of water droplets. This set of equations has been well evaluated in previous studies.[3,4] Flow is supposed to be homogeneous in this study from the assumption that water droplets occurring due to condensation are sufficiently small. Then, velocity slips among air, water vapor, and liquid water are neglected. The equations of state and the speed of sound in this study are those introduced by Ishizaka et al.,[8] assuming that the mass fraction of liquid water, β is sufficiently small ($\beta < 0.1$). These equations are given as follows:

$$p = \rho RT(1-\beta) = \frac{(1-\beta)R}{C_{pm}-(1-\beta)R}\left(e - \frac{1}{2}\rho u_i u_i - \rho h_{0m}\right) \quad (1)$$

$$c = \left[\frac{C_{pm}}{C_{pm}-(1-\beta)R}\frac{p}{\rho}\right]^{\frac{1}{2}} \quad (2)$$

where

$$R = (\rho_a R_u/\rho_g M_a + \rho_v R_u/\rho_g M_v)$$

ρ_a and ρ_g are the densities of dry air and mixed gas; M_a and M_v are the molecular weights of dry air and water vapor, respectively; R_u is the universal gas constant; C_{pm} is defined by the linear combination of the specific heat at constant pressure between gas phase and liquid phase using β; and h_{0m} is the heat of formation.

Condensation Model

In wind-tunnel experiments, onset of condensation may be dominated by homogeneous nucleation,[2] whereas natural condensation observed in the flight of airplanes is governed by heterogeneous nucleation and almost equilibrium condensation. Homogeneous nucleation in this study is based on the classical condensation theory

Received 31 May 2001; presented as Paper 2001-2651 at the AIAA 15th Computational Fluid Dynamics Conference, Anaheim, CA, 11–14 June 2001; revision received 23 April 2003; accepted for publication 12 May 2003.

*Associate Professor, Department of Aeronautics and Space Engineering. Member AIAA.

defined by Frenkel.[9] The surface tension used in the present condensation model was modeled by Schnerr and Dohrmann.[2] Heterogeneous nucleation in this study is approximately calculated by setting a constant radius and a constant number of particulates in atmosphere, which is discussed later.

The mass generation rate Γ of water droplets consists of the mass generation rate of critical-sized nuclei and the growth rate of water droplets. In this study, the approximated model derived by Ishizaka et al.[8] is also employed. The growth rate of a water droplet dr/dt is described by the Hertz–Knudsen law, which assumes that the droplet radius is much smaller than the mean free path of a vapor molecule. A simplified model of the growth rate of a water droplet modeled by Schnerr and Dohrmann[2] is employed. The saturation pressure for a water droplet at radius r is given by the Kelvin–Helmholtz equation.

Numerical Method

A high-resolution finite difference method developed by our research group is used for solving the system of fundamental equations.[4] In this method, the fourth-order compact MUSCL extrapolation[10] satisfying the total variation diminishing condition is implemented for the approximation of primitive variables. The modified Roe's Riemann solver[11] is also employed for space discretization of convection terms. The viscosity term is calculated using the second-order central difference scheme. The explicit second-order Runge–Kutta method is used for the time integration. The eight equations[4] are solved simultaneously at each time step.

Results

Three-dimensional laminar flows of moist air over a 76-deg, sharp-edged single delta wing without thickness at 20.5-deg inclination are calculated for a preliminary understanding of the mechanism of condensation in a streamwise vortex, the so-called vapor trail. The present delta wing is known as an experimental model for the study of off-surface flows. Flow conditions are as follows: the uniform Mach number is $M_\infty = 0.30$, the uniform temperature is $T_\infty = 293.15$ K, the uniform pressure is $p_\infty = 1.24 \times 10^5$ Pa, and the Reynolds number is $Re = 0.9 \times 10^6$, which corresponds to 0.1 m in chord length. The computational grid employed is an H-type grid that has 121, 101, and 101 grid points in x, y, and z directions corresponding to streamwise, spanwise, and top-to-bottom directions, respectively. Sixty grid points are generated on the wing surface in the spanwise and streamwise directions. A symmetric boundary condition is specified at the right boundary surface in the spanwise direction. The inlet boundary is located at two times the wing chord length ahead of the wing front. The top and bottom boundaries are each located at three times the chord length from the wing surface. The left boundary surface in the spanwise direction is located at three times the chord length from the wing tip. The outlet boundary is located at four times the chord length after the wing trailing edge. Freestream boundary conditions are specified at these boundaries. Laminar flows of moist air are calculated assuming atmospheric wind-tunnel and flight conditions. We focus only on a numerical prediction, especially in the onset of condensation in the streamwise vortex generated over the single delta wing, because no experiments have yet been provided. The calculated results are compared to those of different atmospheric flow conditions.

A set of atmospheric wind-tunnel conditions is first taken into consideration. The nucleus of liquid water is supposed to be homogeneously produced. In this case, the stagnation temperature, the stagnation pressure, and the stagnation relative humidity are fixed at $T_{01} = 293.15$ K, $p_{01} = 1.24 \times 10^5$ Pa, and $\Phi_{01} = 90\%$. The Reynolds number is $Re = 0.9 \times 10^6$ (at $M = 0.3$ and chord length of 0.1 m). The inlet Mach number M is changed from $M = 0.3$ to $0.8 (\Delta M = 0.1)$ for the investigation of the effect of inlet flow speed on the onset of condensation. No condensation was found in the case of $M = 0.3$, and the calculated result of $M = 0.4$ also presents a quite trivial condensation, which can be neglected. The maximum condensate mass fraction β_{max} picked up from the values in each flow cross section over the wing is distributed along the streamwise direction in Fig. 1. The location where β is a maximum may be

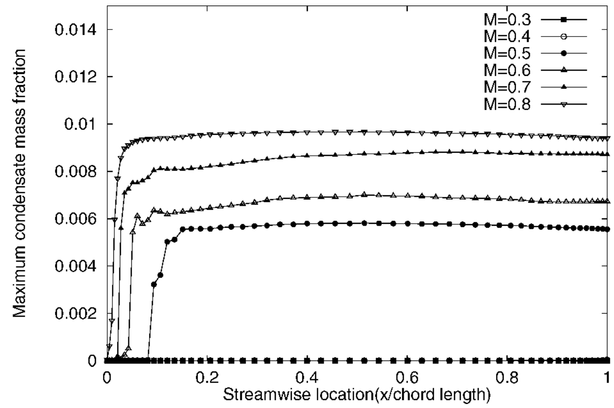
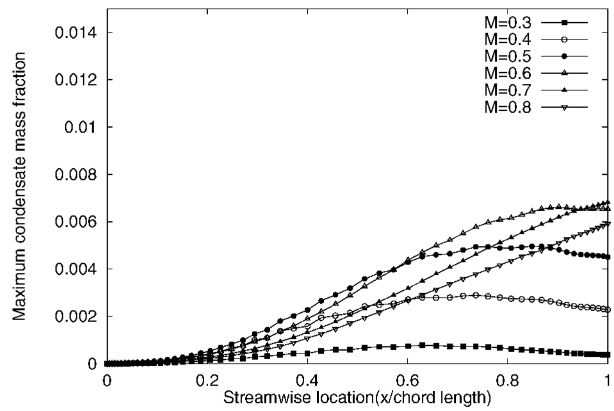
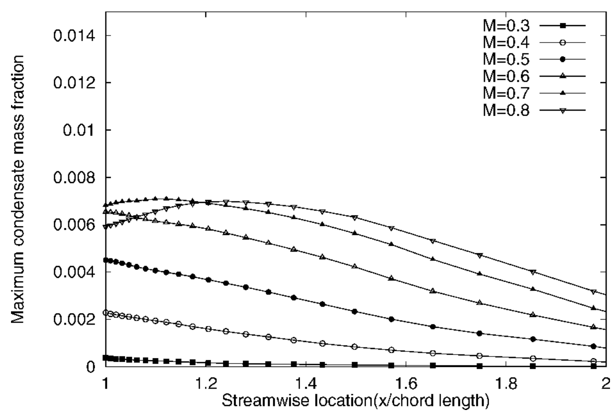


Fig. 1 Maximum condensate mass fraction distributions (atmospheric wind-tunnel conditions).



a) Over the wing

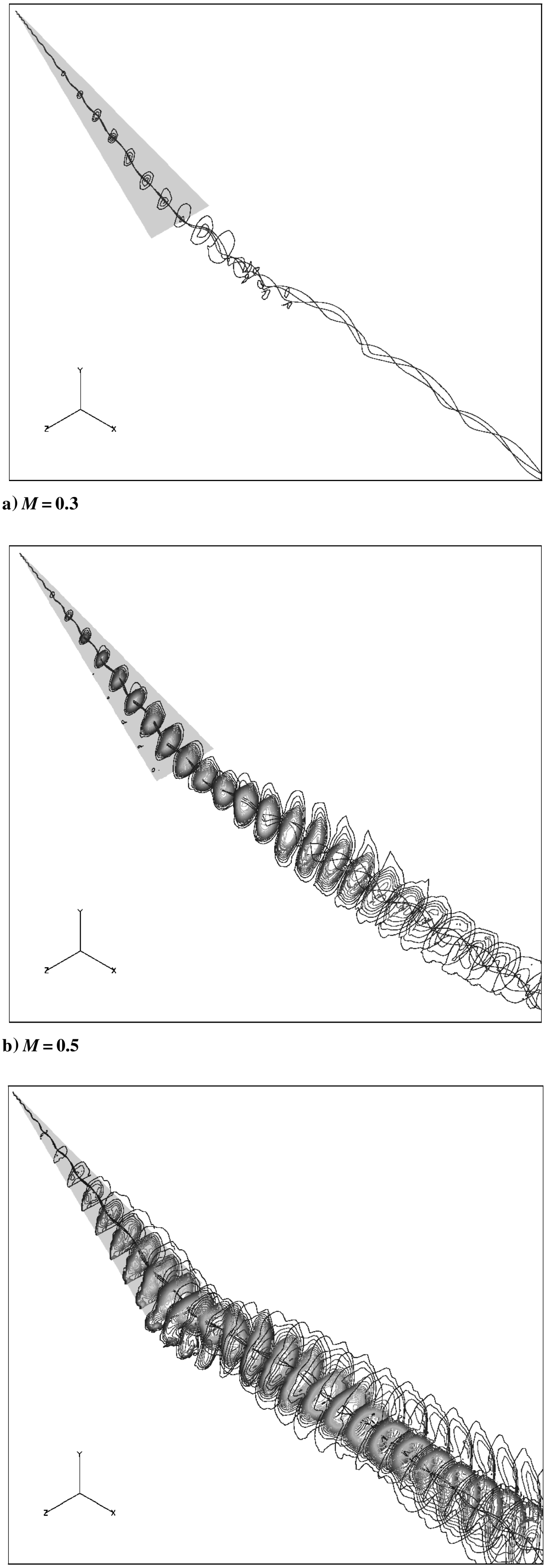


b) Behind the wing

Fig. 2 Maximum condensate mass fraction distributions (atmospheric flight conditions).

closely equivalent to the core of the strake vortex. In all cases beyond the inlet Mach number $M = 0.4$, the onset of condensation is located close to the wing front. The condensation starts rapidly and the value of the maximum condensate mass fraction soon reaches a maximum flat value. This result indicates that water droplets do not grow after β reaches the flat value. Condensation starts from a point closer to the wing front as the inlet Mach number increases.

In actual atmospheric flight conditions, on the other hand, the wing chord length of airplanes is sufficiently long and flow may be essentially a turbulent flow. Furthermore, some dust, soot, ions, or aerosols may play a role as a nuclei of water droplets. However, because the correlation between turbulence and condensation is still unknown, laminar flows with heterogeneous nucleation in atmospheric flight conditions are only calculated herein as a preliminary calculation. A simple approximation for heterogeneous



a) $M = 0.3$

b) $M = 0.5$

c) $M = 0.8$

Fig. 3 Density contours of liquid water and streamlines.

nucleation is employed by assuming that a constant number of spherical particulates with constant radius is already contained in whole flowfield. The radius and the number density are fixed at 0.1 μm and 10^{12} m^{-3} in this study. The chord length of the delta wing is specified as 10 times longer than the previous homogeneous cases: 1.0 m in length. The other conditions are specified as atmospheric flight conditions: the uniform static temperature, the uniform static pressure, and the uniform relative humidity are fixed at $T_\infty = 293.15$ K, $p_\infty = 1.24 \times 10^5$ Pa, and $\Phi_\infty = 90\%$, respectively. The uniform Mach number M is changed from $M = 0.3$ to $0.8(\Delta M = 0.1)$. Condensation is found in all cases. The maximum condensate mass fraction $\beta_{\max}$ picked up from the values in each flow cross section over the wing and behind the wing is distributed along the streamwise direction in Figs. 2a and 2b. Compared with the calculated results in atmospheric wind-tunnel conditions, condensation starts from the wing front in all cases, while the growth of water droplets is relatively slow. Because a constant number density of particulates exists in whole flowfield, condensation starts quickly. However, because the number of particulates is limited compared with that in homogeneous cases, the total mass of water droplets may increase slowly in the strake vortex and the amount is smaller than that in atmospheric wind-tunnel conditions when compared with Fig. 1. Especially in the cases of $M = 0.7$ and 0.8, the maximum peak of the maximum condensate mass fraction is located in the downstream region behind the wing as illustrated in Fig. 2b.

The calculated density contours $\rho\beta$ of liquid water at each flow cross section and the streamlines indicating the strake vortex are visualized in Figs. 3a–3c from the calculated result for the uniform Mach number at 0.3, 0.5, and 0.8, respectively. The plotted maximum value $(\rho\beta)_{\max}$ is 0.01 and $\Delta(\rho\beta) = 2 \times 10^{-4}$. As the uniform Mach number increases, the region with liquid water spreads wider at each flow cross section and the region maintains itself longer downstream along the strake vortex. Condensation also occurs at the apex of the wing in the case of $M = 0.8$.

Conclusions

Three-dimensional laminar flows of moist air over the 76-deg, sharp-edged single delta wing without thickness at 20.5-deg inclination in atmospheric flow conditions were calculated, and condensation in the streamwise vortex was numerically investigated. The calculated results indicate that the condensation deeply depends on atmospheric flow conditions. Onset of condensation was very sensitive to the atmospheric wind-tunnel conditions assuming homogeneous nucleation. A large number of water droplets was produced after the onset of condensation and the amount quickly reached almost a flat value for these conditions. Condensation could not be observed in the case with inlet Mach number $M < 0.5$. On the other hand, water droplets grew slowly after the onset of condensation in the atmospheric flight conditions assuming heterogeneous nucleation; condensation was found in the case at lower uniform Mach numbers compared with those in the atmospheric wind-tunnel conditions. Consequently, these results suggest that the assumption of atmospheric flight conditions should be necessary to obtain a real condensation phenomenon in a streamwise vortex over a delta wing. The present study may, however, be in a preliminary level; further modifications to the present method are also needed.

References

[1]Campbell, J. F., Chambers, J. R., and Rumsey, C. L., "Observation of Airplane Flowfields by Natural Condensation Effects," *Journal of Aircraft*, Vol. 26, No. 7, 1989, pp. 593–604.

[2]Schnerr, G. H., and Dohrmann, U., "Transonic Flow Around Airfoils with Relaxation and Energy Supply by Homogeneous Condensation," *AIAA Journal*, Vol. 28, No. 7, 1990, pp. 1187–1193.

[3]Iriya, A., Yamamoto, S., and Daiguji, H., "Numerical Method for Transonic Viscous Flow Considering Humidity," *Transactions of the Japan Society of Mechanical Engineers, Series B*, Vol. 62, No. 603, 1996, pp. 100–105 (in Japanese).

[4]Yamamoto, S., Hagari, H., and Murayama, M., "Numerical Simulation of Condensation Around the 3-D Wing," *Transactions of the Japan Society for Aeronautical and Space Sciences*, Vol. 42, No. 138, 2000, pp. 182–189.

[5]Hsu, C.-H., Hartwich, P.-M., and Liu, C. H.,"Incompressible Navier–Stokes Computations for a Rounded-Edged Double-Delta Wing," *Journal of Aircraft*, Vol. 25, No. 8, 1988, pp. 675, 676.

[6]Fujii, K., and Schiff, L. B., "Numerical Simulation of Vortical Flows over Strake-Delta Wing," *Journal of Aircraft*, Vol. 27, No. 9, 1989, pp. 1153–1162.

[7]Gordnier, R. E., and Visbal, M. R., "Unsteady Vortex Structure over a Delta Wing," *Journal of Aircraft*, Vol. 31, No. 2, 1994, pp. 243–248.

[8]Ishizaka, K., Ikohagi, T., and Daiguji, H., "A High-Resolution Finite Difference Scheme for Supersonic Wet-Stream Flows," *Transactions of the Japan Society of Mechanical Engineers, Series B*, Vol. 60, No. 579, 1994, pp. 3887–3892 (in Japanese).

[9]Frenkel, J., *Kinetic Theory of Liquids*, Dover, New York, 1955, Chap. 7.

[10]Yamamoto, S., and Daiguji, H., "Higher-Order-Accurate Upwind Schemes for Solving the Compressible Euler and Navier–Stokes Equations," *Computers and Fluids*, Vol. 22, No. 2/3, 1993, pp. 259–270.

[11]Yuan, X., Yamamoto, S., and Daiguji, H., "A Higher-Resolution Shock-Capturing Scheme for Simulating Unsteady Three-Dimensional Transonic Flows in Turbomachinery," AIAA Paper 94-3199, July 1994.

A. Plotkin
Associate Editor

Trailing Vortices from a Wing with a Notched Lift Distribution

W. R. Graham,* S.-W. Park,† and T. B. Nickels‡
University of Cambridge,
Cambridge, England CB2 1PZ, United Kingdom

Nomenclature

c	=	wing chord
c_l	=	section lift coefficient
R	=	chordwise radius of curvature
s	=	span of half-wing
U	=	towing speed
y, z	=	spanwise and normal coordinates
α_{local}	=	section incidence angle
Γ	=	section circulation

Introduction

INCREASING demands on airport capacity and the imminent arrival of the Airbus A380 super-Jumbo have refocused research and regulatory attention on the hazard associated with an aircraft's trailing vortices. Thus, from a manufacturer's viewpoint, configuration modifications that alleviate these vortices are potentially desirable.

In a recent paper,[1] Graham used two-dimensional vortex method calculations to show that significant reductions in the rolling moment induced on a following aircraft were predicted for notched wing lift distributions. Here a region of counter-rotating vorticity is shed between the flap outboard and wing tip vortices, which prevents their merger. This region is unlike the counter-rotating vortex typically generated by the aircraft's tail, in that it never rolls up into a distinct axisymmetric structure, instead becoming wrapped around the tip vortex. For a follower with wingspan 50% of that of the generator, the predicted reductions in rolling moment were around 20–25%.

One possible interpretation of these gains is that they fall short of the improvement required. This argument is based on results described in Rossow's recent review paper,[2] where a "vortex dissipater" mounted on a Convair 990 reduced the roll accelerations induced on a following Lear jet from 4.4 to 2.4 rad/s^2 without altering the pilot-perceived hazard. This, however, was because the lower acceleration still comfortably exceeded the Lear jet's roll authority, a feature that is clearly dependent on the size of both the generator and the follower. One could equally well envisage a situation where a 25% reduction in rolling moment brought it down to a level within the follower's roll authority. Thus, an alternative interpretation is that such a gain would alleviate the hazard associated with a 360-t landing weight aircraft to a level comparable with a 270-t B747-400. Additionally, the corresponding four-vortex system would be potentially susceptible to the accelerated instabilities reported by Crouch et al.[3]

The potential benefits of notched lift distributions were, therefore, deemed sufficient to seek experimental verification of the numerical predictions. This Note describes initial, proof-of-concept results from the experimental program.

Apparatus and Method

To follow the wake evolution as far as possible, the tests were carried out in a towing tank. The tank is 17.5 m long and 0.59 m wide and was filled to a depth of 0.6 m.

Two model half-wings, designed to exhibit conventional and notched high-lift distributions, were tested. (The use of half-wings, with root located at the water surface, increases the allowable model size and eliminates support interference.) Each has a (semi-) span of 300 mm and a mean chord of 100 mm, resulting in an effective aspect ratio of 6. Their cross sections have a cambered-plate geometry (to avoid the low-Reynolds-number problems associated with conventional airfoil sections), with thickness of 2 mm, radius of curvature of 210 mm, and nose radius of 1 mm. The leading edges are straight, and the required loading variations are achieved by varying the chord length. The conventional wing has a rectangular flap planform, with chord of 106 mm over the inner 183 mm of span and 90 mm over the remainder. The notched wing has constant chord sections of 105 and 102 mm over the inner 135 mm and outer 75 mm, with a smoothly varying notch of minimum chord 77 mm in between.

The wing planforms, along with lift distributions evaluated using lifting line theory,[4] are shown in Fig. 1. Section properties for this calculation, estimated on the basis of Schmitz's experimental results for the Göttingen 417a cambered plate aerofoil,[5] were

$$c_l = 2.75\pi\alpha_{\text{local}} + 1.16c/R \tag{1}$$

The notched distribution closely matches the optimum identified in Sec. III of Ref. 1. The conventional loading has the same overall lift coefficient, 0.43, and almost identical root circulation (which determines the overall circulation shed into the wake). The induced drag factors (relative to elliptic loading) are 1.12 and 1.04, respectively.

The tests were conducted at a towing speed of 0.534 m/s, corresponding to a (mean-chord-based) Reynolds number of 5.34×10^4. The wings were impulsively set in motion from 3.7 m down the tank, and the associated wake velocities were measured using digital particle image velocimetry (DPIV) at a crossflow plane 2.1 m farther downstream. The PIV system is a commercial model produced by Dantec and consists of a New Wave Research Gemini PIV 15 Nd: YAG laser, an MASD, Inc., Megaplus Model ES 4.0 digital camera, and a data acquisition/control unit linked to a personal computer. The seeding particles used have a diameter of 100 μm and relative density 1.0 ± 0.02. After seeding, the PIV system was used to check that the resulting fluid motions had decayed before a run was started.

Velocity fields were derived from the raw data using Dantec's proprietary cross-correlation software on pairs of images separated

Received 2 October 2002; revision received 2 April 2003; accepted for publication 7 April 2003.

*Senior Lecturer, Department of Engineering, Fluid Mechanics Group, Trumpington Street; wrg@eng.cam.ac.uk.

†Research Student, Department of Engineering, Fluid Mechanics Group, Trumpington Street.

‡Lecturer, Department of Engineering, Fluid Mechanics Group, Trumpington Street.