Technical Notes

TECHNICAL NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Mean Value Measurements of a Turbulent Swirling-Jet

M. Samet* and S. Einav†
Tel-Aviv University, Ramat-Aviv, Israel

Introduction

THE turbulent swirling-jet in a coflowing stream is a typical example of a complex flow that arouses particular interest, since it incorporates characteristics of both swirling-motion and coaxial-mixing. The main goal of most related research is to enhance and control the mixing process by simple yet reliable methods.

Several experimental studies of the turbulent swirling-jet published by Rose,[1] Chigier and Chervinsk,[2] Pratte and Keffer,[3] Hosel and Rodi,[4] and Young and Rao[5] showed that the swirl can readily modify the overall turbulence structure in the mixing region. These studies also indicated that certain combinations of axial- and angular-momentums could produce local recirculations of the mean-flow, stabilize curved, turbulent shear-flow, whereas, in other cases, swirl was shown to destabilize the entire flow.

Our knowledge of the swirling-jet in a coflowing stream is quite limited. Only one experimental investigation of the swirling-jet in a coflowing stream has recently been reported in the literature: the study presented by Morse,[6] which was based on one velocity-ratio and one degree-of-swirl. The present study is an attempt to provide sufficiently complete and reliable information on swirling-jets in still and moving surroundings.

Description of the Experimental Apparatus

An open-circuit wind tunnel (Fig. 1) was used to provide a uniform coflowing stream of air. The tunnel consisted of a conventional arrangement of an intake, working section, diffuser, transition piece, axial fan, and calming chamber. The bell-mouthed-type intake was designed to have a square cross section at each point along the main axis, and a contraction ratio of 10.3. The working section (cross-sectional area, 500 mm × 500 mm) was relatively short (1000 mm); this minimized any interaction between the swirling-flow and the boundary layers along the walls. The air speed in the test section could be varied between a minimum of 2.5 m/s and a maximum of 18 m/s.

The swirl generator essentially followed the design used by Chigier and Chervinsky.[2] It consisted of an inner and outer concentric section with three inclined slots. Through these slots the outer, or tangential, airstream could enter the inner one (axial flow), thereby forcing the inner-stream to rotate. The swirling-flow then passed through a high-quality nozzle and was discharged at the opening of a 25.4 mm straight portion. Such a combination provided a uniform axial-velocity distribution and a well-defined swirl-pattern at the exit.

The flow parameters were measured using a specially designed, directional, 5-hole pressure probe. Description of this probe, its calibration, and data reduction processes are covered in detail by Samet and Einav.[7]

Experimental Results and Discussion

During the experimental study, 20 different cases were examined. Each case was defined by two parameters: the velocity ratio μ and the degree of swirl S.

The velocity ratio is defined as

$$\mu = \frac{U_\infty}{U_{mo}} \tag{1}$$

and the degree-of-swirl as

$$S = \frac{W_{mo}}{U_{mo}} \tag{2}$$

where

U_∞ = velocity of the uniform stream
U_{mo} = maximal axial velocity at the nozzle exit
W_{mo} = maximum angular velocity at the nozzle exit

During the experimentation four velocity ratios (0, 0.08, 0.20, and 0.30) and five values of S (0, 0.12, 0.31, 0.40, and 0.49) were used.

Mean Velocity Vector

The mean-velocity vector $\bar{V}$ is determined by the directions ψ and α and by the resultant mean velocity Q. Typical distributions of the yaw angle ψ are presented in Fig. 2. As expected, one can find for $S = 0$ that close to the exit plane there is a steep gradient of $d\psi/dr$. By changing the degree-of-swirl from $S = 0$ to $S = 0.31$, no apparent changes are observed in the distributions of ψ. The swirling-flow tends to increase its radial-spread while decreasing local values of ψ, whereas, in the nonswirling-flow, the radial motion is quite restrained. The most pronounced difference between two swirling-jets occurs

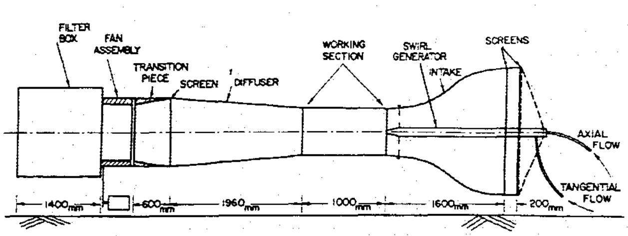
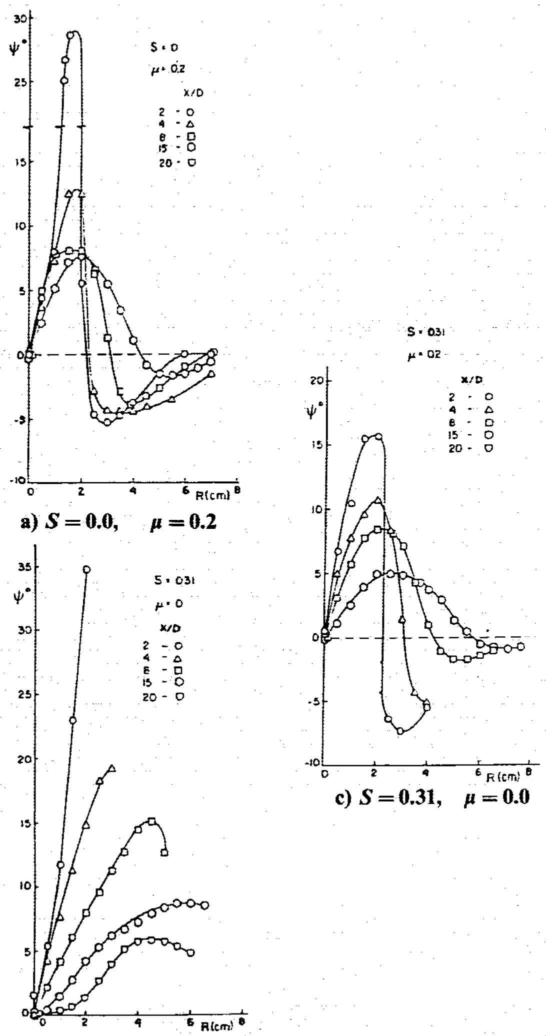


Fig. 1 The experimental apparatus. Coflowing air is driven by the fan assembly, whereas the axial and swirl jet stream are fed by the swirl generator.

Received Aug. 19, 1985; revision received May 20, 1987. Copyright © American Institute of Aeronautics and Astronautics, Inc., 1987. All rights reserved.

*Currently at the Department of Mechanical Engineering, Marquette University, Milwaukee, Wisconsin.

†Faculty of Engineering, Department of Fluid Mechanics and Heat Transfer.



a) $S = 0.0, \quad \mu = 0.2$

b) $S = 0.31, \quad \mu = 0.2$

c) $S = 0.31, \quad \mu = 0.0$

Fig. 2 Radial distribution of the yaw angle (3 cases).

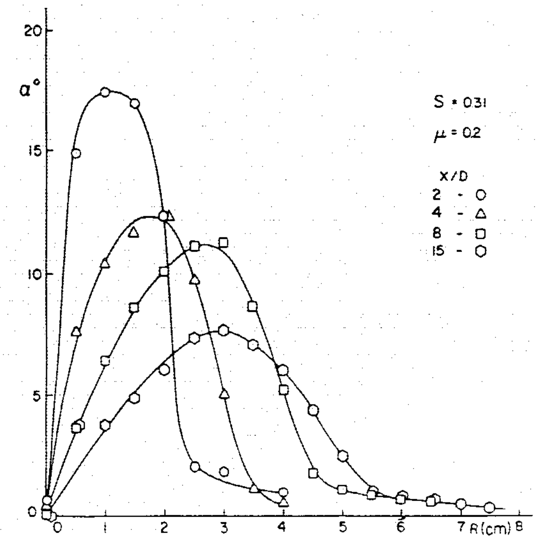
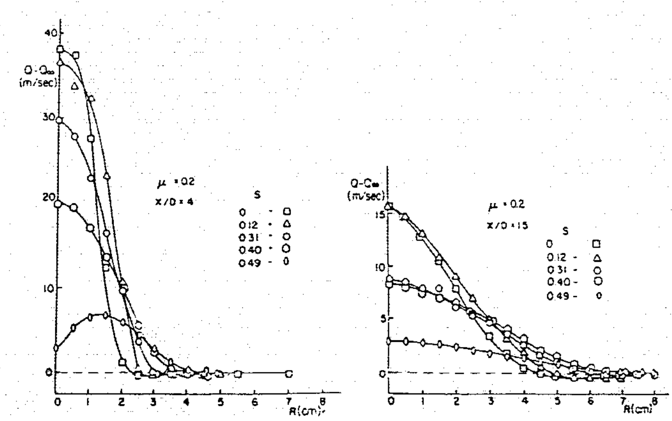


Fig. 3 Radial distribution of the pitch angle.



a) $X/D = 4$ b) $X/D = 15$

Fig. 4 Radial distribution of the velocity excess (2 cases).

when the velocity ratio in one of them is reduced to its minimal value $\mu = 0$. Such an extreme case is shown in Fig. 2c. As expected, the jet exhibits the high spreading rates of its boundaries, as well as extremely high local values for the yaw angle.

The pitch angle α of the mean-flow reflects the swirling motion of the fluid. Figure 3 shows typical distributions of α when $\mu = 0.2$ and $S = 0.31$. Unlike the distribution of ψ, no pronounced differences in distributions for α were found when μ or S were changed.

The resultant mean velocity Q is defined as

$$(Q = (\bar{V}^2)^{\frac{1}{2}}) \tag{3}$$

where $\bar{V}$ is the mean velocity vector. For a jet in coflowing stream, it is preferable to discuss flow characteristics in terms of velocity excess rather than in absolute values. Figure 4 presents radial distributions of $Q - Q_\infty$ at two cross sections, $X/D = 4$ and $X/D = 15$. The steep velocity gradients inherent to the flow pattern at the exit tend to be moderated further downstream due to the rapid decay of velocity excess (nearly 50% on the average).

From the above it follows that by increasing swirling motion, one can diminish the influence of the secondary stream and vice versa. Figure 4a shows that while the coflowing velocity and Re number were kept approximately the same, the resultant mean-velocity differs significantly for each value of S. When the mixing process takes place, the influence of the secondary stream becomes quite important, since it tends to smooth down differences in jets that vary but slightly in their degree-of-swirl. This is illustrated in Fig. 4b where the slightly swirled and unswirled jets exhibit virtually identical mean profiles. The cases in which $S = 0.31$ and $S = 0.4$ also support this idea. By decreasing μ to 0.08, the same results were obtained, but much closer to the exit plane. Increasing μ to 0.3 caused the same phenomena to occur further downstream.

In conclusion, the data obtained during the present study were restricted to mean values in the swirling jet. The data showed that the longitudinal and radial distributions of mean-velocity components were strongly affected by the degree-of-swirl and velocity ratio.

Acknowledgment

This work was supported in part by U.S. Air Force Grant AFOSR 84-0278.

References

[1]Rose, W. G., Ph.D. Thesis, The Johns Hopkins University, Baltimore, MD, 1962.

[2]Chigier, N. and Chervinsky, A., "Experimental Investigation of Swirling Vortex Motion in Jets," *ASME Journal of Applied Mechanics*, Vol. 34, 1967, pp. 443–451.

[3]Pratte, B. D. and Keffer, J. F., "The Swirling Turbulent Jet," *ASME Journal of Basic Engineering*, Vol. 94, 1972, pp. 739–748.

[4]Hosel, W. and Rodi, W., "Drallstrahelmessunger mit einem Laser Doppler Anemometer," Univ. of Karlsruhe, 1977, Rept. SFB 80/E/106.

[5]Young, A. D. and Rao, K. N., "Some Low Speed Results on the Effects of Swirl and Velocity Distribution on an Axisymmetric Jet," *Aerospace Quarterly*, Vol. 29, 1978, p. 4.

[6]Morse, A. P., "Some Experiments on the Turbulent Swirling Jet with and without External Stream," Ph.D. Thesis, Imperial College, London, 1979.

[7]Samet, M. and Einav, S., "The Directional Pressure Probe," *Review of Scientific Instruments*, 1984, pp. 582–588.

Vortex Flow Model for the Blade-Vortex Interaction Problem

Andrew T. Hsu* and James C. Wu†
Georgia Institute of Technology, Atlanta, Georgia

Introduction

IN the present paper, a highly efficient vortex flow model is developed to simulate the attached two-dimensional blade-vortex interaction (wherein boundary layers remain attached throughout the interaction process). Discrete vortex dynamics have been previously employed by other researchers in studying inviscid incompressible flows.[1,2] The present work differs from the previous studies in three major points. 1) The present work utilizes a viscous theory of aerodynamics to evaluate the aerodynamic load. 2) The work combines a theoretical solution for the boundary vortex sheet with a numerical procedure for the wake vortices. 3) In the present analysis, an accurate procedure is established for the simulation of the vortex shedding process, which is crucial to the accuracy of solutions of all inviscid unsteady flows.

Recent experimental observations by Poling and Telionis[3] indicate that the present trailing-edge flow model is physically realistic. The results from this model tend to the steady-flow solution asymptotically as the change in circulation vanishes, and the model allows vorticity to be shed from both sides of the trailing edge. In the present model, the location and the strength of the nascent vortex are determined separately. The usual difficulty that the strength of the nascent vortex is oversensitive to its location is thus eliminated.

Closed-form solutions for the unsteady lift, drag, and moment experienced by an airfoil encountering a vortex passing near it are obtained by applying a general theorem of aerodynamics.[4] These closed-form solutions allow the evaluation of vortex-induced unsteady loads without tedious calculations. Furthermore, these solutions allow the contributions of the various vortical regions coexisting in the interaction flowfield, such as the wake and the boundary layer, to be evaluated individually. The relative importance of each vortical region can then be assessed. Because of the simplicity of these formulations, they can be easily extended to the study of the noise induced by blade-vortex interaction.[5] Selected results obtained by the present vortex flow model are presented and compared with other numerical solutions in this paper.

Presented as Paper 86-1094 at the AIAA/ASME 4th Fluid Mechanics, Plasma Dynamics, and Lasers Conference, Atlanta, GA, May 12–14, 1986; received July 1, 1986; revision received Jan. 4, 1988.
Copyright © American Institute of Aeronautics and Astronautics, Inc., 1988. All rights reserved.

*Postdoctoral Fellow, School of Aerospace Engineering. Currently Research Engineer, Sverdrup Technology, Inc., Lewis Research Center, Cleveland, OH. Member AIAA.

†Professor of Aerospace Engineering. Associate Fellow AIAA.

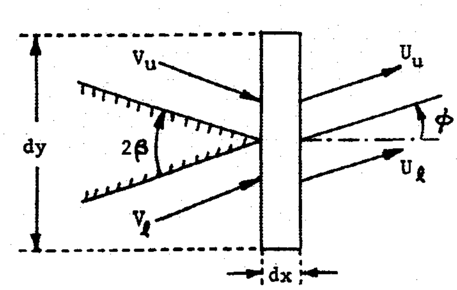
Vortex Flow Model for Blade-Vortex Interaction

In high-Reynolds number external flows, the vorticity is generally concentrated in regions near the solid body. Under some circumstances, simplified models can be utilized in analyzing the flow. For an attached blade-vortex interaction problem, the vortical region is composed of three distinct components: the passing vortex, i.e., the oncoming vortex that interacts with the blade; the boundary layer at the blade surface; and the wake trailing the blade. An accurate solution of the flowfield depends on the correct modeling of these three vortical components.

In the present study, the passing vortex Ω is represented by a point vortex, which is an accurate representation provided that the vortex is outside of the boundary layer during the interaction. In high-Reynolds number flows, the boundary layer is very thin compared to the characteristic length of the solid body. The vorticity in the boundary layer can be represented by a vortex sheet γ. Using ω to denote the vorticity field, the vortex sheet strength can be defined as $\gamma = \int_o^\delta \omega \mathrm{d}n$, where δ is the boundary-layer thickness. The wake is a continuation of the boundary layer. That is, the vorticity in the boundary layer is continuously transported downstream and shed into the wake. In the present study, the wake vorticity is represented by a series of point vortices, Γ_i, $i = 1,2,\ldots,N$, successively shed from the airfoil's trailing edge. Both the passing vortex and the wake vortices are allowed to convect with the local fluid velocities v_i, which are determined by the Biot-Savart law. Except for the nascent vortex, the positions of vortices are determined at each step by simple numerical integration.

Nascent Wake Vortex

In order to determine the strength and the position of a nascent vortex, three flow quantities at the airfoil's trailing edge are needed. These are 1) the vorticity shedding rate, 2) the flow velocity, and 3) the flow direction. At the trailing edge of the airfoil, in the absence of flow separation, the boundary layers on the two sides of the airfoil merge into a single wake layer. Since in the present model the boundary layers of finite thickness are represented by vortex sheets of zero thickness, the smoothly turning streamlines at the outer edges of the boundary layers are replaced by two streamlines that have finite turning angles at the trailing edge. In an unsteady real fluid flow, the wake vortex sheet is in general not parallel to either the upper or the lower surface of the airfoil trailing edge. The direction of the wake layer can be determined by applying the momentum theorem to an infinitesimal control volume at the airfoil trailing edge, as shown in the sketch.



For a sufficiently small control volume, the momentum equation can be written as

$$\int_{cs} \rho v(v \cdot \mathrm{d}s) = 0 \tag{1}$$

where v is the fluid velocity and cs is the control surface. The given vector expression can be written as two scalar equations:

$$(V_u^2 + V_\ell^2)\cos^2\beta = -(U_u^2 + U_\ell^2)\cos^2\phi \tag{2}$$