[2]Newman, P. A., Hou, G. J.-W., Jones, H. E., Taylor, A. C., III, and Korivi, V. M., "Observations on Computational Methodologies for Use in Large-Scale Gradient-Based Multidisciplinary Design," *Proceedings of the Fourth AIAA/USAF/NASA/OAI Symposium on Multidisciplinary Analysis and Optimization* (Cleveland OH), AIAA, Washington, DC, 1992, pp. 531–542 (AIAA Paper 92-4753).

[3]Newsome, R. W., Walters, R. W., and Thomas, J. L., "An Efficient Strategy for Upwind Relaxation Solutions to the Thin-Layer Navier-Stokes Equations," *AIAA Journal*, Vol. 27, No. 9, 1989, pp. 1165, 1166; also AIAA Paper 87-1113, June 1987.

[4]Barger, R. L., and Adams, M. S., "Automatic Computation of Wing-Fuselage Intersection Lines and Fillet Inserts with Fixed-Area Constraint," NASA TM 4406, March 1993.

Swirl Control in an S-Duct at High Angle of Attack

P. F. Weng*
Shanghai Jiao Tong University, Shanghai 200030, People's Republic of China
and
R. W. Guo†
Nanjing University of Aeronautics and Astronautics, Nanjing 210016, People's Republic of China

Introduction

A DIFFICULT design problem occurs with military aircraft where the inlet is located in an offset position. The problem of angular swirl flow in an S-shaped inlet without guide vanes has come to the fore in the past several years.[1,2] Many possible modifications of the inlet geometry have been used in the reduction of swirl. Prior work by Guo and Seddon[2,3] has shown that a large vortex occurs because of flow separation on the bottom wall near the duct throat which produces a large swirl at the engine face. In the prior work, two devices for swirl control were used; one is the solid spoiler, the other is auxiliary inflow by blowing (auxiliary inflow was found to have adverse effects on the flow and pressure recovery characteristics at lower angles of attack). Seddon[4] tested the effect of bottom wall and sidewall fences of various sizes and combinations in reducing the swirl of an S-duct at a 30-deg angle of attack. Tests of Vakili et al.[5] explored the improvement of the secondary flow in an S-duct at 0-deg incidence by using a vortex generator or a flow control rail. The present authors[6] proposed an automatically adjustable blade (AAB) method. The AAB was located inside the duct, and the function relating angle of attack and the optimized angle of the fix (where the duct swirl disappears) was determined. The measures reviewed in the preceding would be invalid or would produce a large total pressure loss when an S-shaped inlet is at a high angle of attack. The flow distortion and duct swirl increase with increasing the angle of attack (up to 80 deg or 115 deg).

This paper describes an improvement of a severely distorted swirl flow in an S-shaped duct at very high angle of attack using a variable lip technique.

Received Jan. 30, 1993; revision received Dec. 15, 1993; accepted for publication Dec. 20, 1993.

*Postdoctoral Fellow, Department of Engineering Mechanics. Senior Member AIAA.

†Professor, Vice President, Department of Aerospace Engineering.

Experimental Description

The S-shaped diffuser investigated is shown in Fig. 1 and consisted of five parts: lip, first bend, straight midsection, second bend, and straight rear section. The diffuser area ratio (exit area to throat area ratio) was 1.3095. Figure 1 also shows the shape and the size of the variable lip device and its location on the model. The lip is 124 mm long, 50 mm wide, and 10 mm thick with a trailing angle of 45 deg. The variable lip is the same shape as those on other walls. It was located at an axial station near the entry where flow separation will occur. The angle of the lip β is 0 deg if the variable lip is close to the wall. The variable lip may change its position by rotation to adjust to the entrance air flow direction. Experimental measurements, including static pressures along the model walls, the cross flow velocities and the total pressure distribution at the diffuser exit were made at α = 0-, 15-, 30-, 45-, 60-, 70-, and 80-deg angles of attack, and compared with test results without the AAB. The tests were performed in a mixed wind tunnel powered by both a vacuum pump and a blower at the Inlet Aerodynamics Laboratory of Nanjing University of Aeronautics and Astronautics. Freestream velocity is about 41.0 m/s. Details of the data processing are described in Ref. 6.

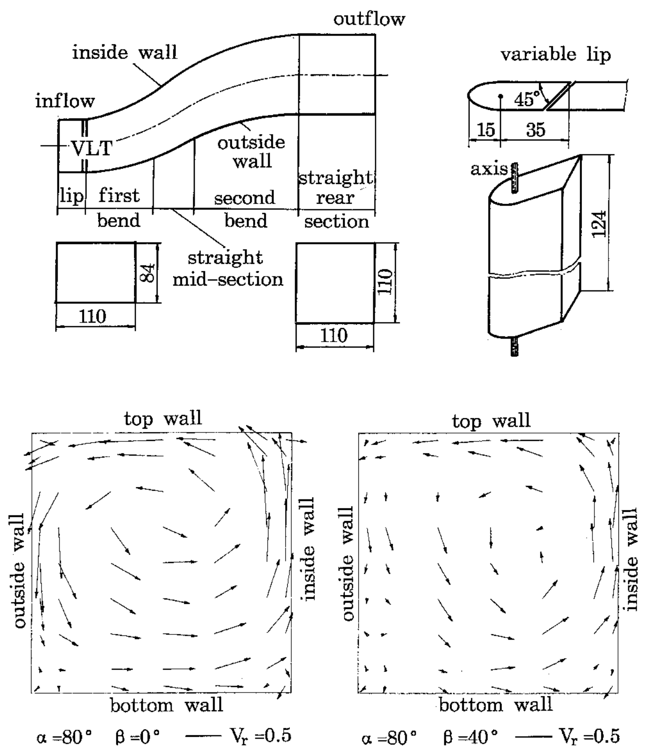

Fig. 1 Experimental model and exit velocity field.

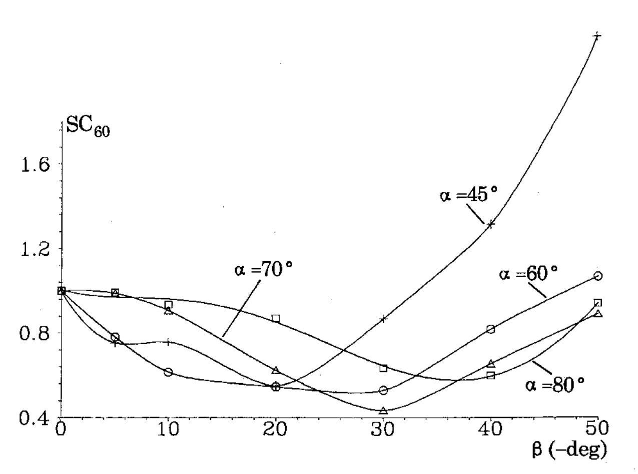

Fig. 2 Variation of SC_{60} with β.

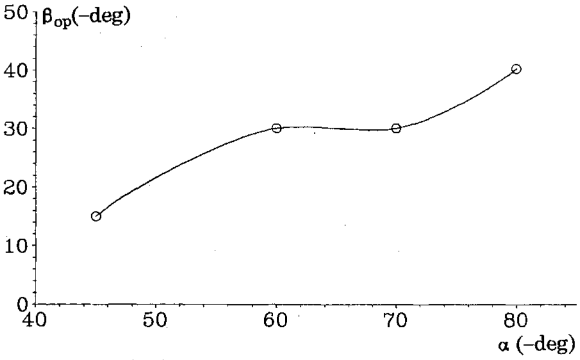

Fig. 3 Relationship of α and β_{op}.

Results and Discussion

Figure 1 gives the cross flow velocities at the duct exit at 80-deg angle of attack. It is observed that the cross flow velocity becomes larger as the angle of attack α increases up to 80 deg. The flowfield is badly distorted, and a strong vortex exists inside the inlet. The use of the variable lip affects the entrance air flow. With a gradual increase of β, two effects occur: 1) the angle of attack seems to reduce, and 2) a door with the same effect as an auxiliary air intake door (AAID) exists behind this lip because of its rotation, and allows air flow with a high kinetic energy into the diffuser. Figure 1 also shows the reduction of the swirl flow at 80-deg angle of attack when β is 40 deg. The test results show the variations of swirl coefficient SC_{60}, entrance air flow flux Q, average total pressure coefficient η_σ, and pressure distortion index DC_{60} with the angle of the lip β at 45-, 60-, 70-, and 80-deg angles of attack. From Fig. 2 it is observed that there is a minimum value of SC_{60} for every curve. The SC_{60} can be reduced by about 40% when α is 80 deg and β is 40 deg. An increase in the entrance air flow is obtained at high angles of attack. Q is increased by about 22% at α = 80 deg and β = 40 deg. With the variable lip, the total pressure loss and its distortion are reduced considerably. For an α of 80 deg and a β of 40 deg, η_σ increases by 40% and DC_{60} decreases by almost 43%. There is an optimum angle of the variable lip β_{op} for every angle of attack where a maximum improvement in flow performance is obtained. β_{op} is a function of α, as shown in Fig. 3. Although more work is needed for compressible flow, the relationship of α and β_{op} is of great value for automatic control of intake swirl.

Conclusions

The main conclusions drawn from the preceding are the following. 1) The variable lip technique improves flow properties in an S-shaped duct at the high angle of attack. A valuable relationship between α and β_{op} is established in the paper. 2) The variable lip has two effects: a) a reduction of the effective angle of attack, and b) an effect similar to that of an AAID.

References

[1]Stocks, C. P., and Bissinger, N. C., "The Design and the Development of the Tornado Engine Air Intake," AGARD Symposium on Aerodynamics of Power Plant Installation, Paper 10, May 1981.

[2]Guo, R. W., and Seddon, J., "The Swirl in an S-Duct of Typical Air Intake Proportions," *Aeronautical Quarterly*, Vol. 32, Pt. 2, May 1983, pp. 99–129.

[3]Guo, R. W., and Seddon, J., "Swirl Characteristics of an S-shaped Air Intake with both Horizontal and Vertical Offsets," *Aeronautical Quarterly*, Vol. 32, Pt. 2, May 1983, pp. 130–146.

[4]Seddon, J., "Understanding and Countering the Swirl in an S-Duct, Testing on the Sensitivity of Swirl to Fence," *Aeronautical Journal*, Vol. 88, No. 874, 1984, pp. 117–127.

[5]Vakili, A. D., Wu, J. M., Liver, P., and Brat, M. K., "Flow Control in a Diffusing S-Duct," AIAA Paper 85-0524, March 1985.

[6]Weng, P. F., and Guo, R. W., "New Method of Swirl Control in a Diffusing S-Duct," *AIAA Journal*, Vol. 30, No. 7, 1992, pp. 1918, 1919.

Relationship Between Stokes Number and Intrinsic Frequencies in Particle-Laden Flows

S. K. Aggarwal*
University of Illinois at Chicago, Chicago, Illinois 60680

Introduction

A KEY parameter characterizing particle dispersion in turbulent flows is the Stokes number, defined as the ratio of the particle response time (t_p) to the appropriate time scale of turbulence or flow time (t_f). While the particle response time is generally taken as $t_p = \rho_p d_p^2/(18\,\mu)$, assuming the Stokes drag law, the choice of a proper flow time is somewhat ambiguous because of the existence of several time scales in a turbulent shear flow. Results from both laboratory experiments[1–4] and numerical simulations[5,6] indicate that the flow dynamics as well as the particle dispersion behavior in the near region of a shear flow are controlled by large-scale coherent structures. Naturally, a proper flow time for defining the Stokes number should be based on a flow time associated with the dynamics of large structures. The latter time is generally defined from the global length and velocity scales of the structures. However, this definition is somewhat ambiguous in situations when the shear flow under consideration lacks an intrinsic length scale. For example, using the initial momentum thickness in a developing planar shear layer would not yield the correct flow time scale. In such cases, one may be able to use the intrinsic time scale of the unsteady flow for defining the Stokes number. The objective of this note is to propose, based on the results of numerical simulations, that using the dominant frequency of the shear flow can yield the proper flow time scale for defining the particle Stokes number. Results presented on particle dispersion indicate that the dominant frequency is the fundamental frequency for a planar shear layer, and the preferred mode frequency for an axisymmetric jet flow. Several experimental results from the literature[1–4] are used to further substantiate the hypothesis.

It is recognized that relating the flow time scales to the dominant frequencies in a shear flow is not a novel concept. In fact, there exists a large body of experimental and theoretical work[9,10] focusing entirely on identifying the dominant frequencies in different shear flows. However, none of the studies concerning particle dispersion in large coherent structures has used the dominant frequency to define the particle Stokes number.

Results and Discussion

The dynamics of large-scale vortex structures in the developing planar shear layer and axisymmetric jet are simulated by employing a finite-difference method based on the monotone integrated large eddy simulation (MILES) algorithm.[7] The dynamics of solid particles injected into the shear flows is simulated by solving the standard particle equations by a Runge-Kutta procedure. Further details are provided in a separate study.[8] The gas-phase algorithm employs a fourth-order accurate, flux-corrected transport method with a directional time-splitting technique with appropriate inflow and outflow boundary conditions, and is shown to reproduce the large-scale features of a variety of flows that are observed in the laboratory experiments. Although no explicit subgrid model is used, the algorithm is demonstrated[7] to incorporate a built-in subgrid model. Moreover, because the present study considers the influence of large coherent structures on particle dispersion in the initial region of two shear flows, the algorithm is expected to be sufficiently accurate for the stated objective.

Received March 23, 1993; revision received Nov. 16, 1993; accepted for publication Dec. 18, 1993.

*Associate Professor, Department of Mechanical Engineering. Associate Fellow AIAA.