AIAA 2004-3933

Modeling and Simulation of Dynamic Inlet Flow Distortion Generation

K. P. Savage, G. A. Flandro and J. Majdalani
Advanced Theoretical Research Center
University of Tennessee Space Institute
Tullahoma, TN 37388

40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit
11–14 July 2004
Fort Lauderdale, FL

40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit
11-14 July 2004, Fort Lauderdale, Florida

AIAA 2004-3933

Modeling and Simulation of Dynamic Inlet Flow Distortion Generation

Keith P. Savage,* Gary A. Flandro,† and Joseph Majdalani‡
University of Tennessee Space Institute, Tullahoma, TN 37388

Integrating a jet engine into an airframe involves many design and testing steps. Furthermore, the development of high performance fighter aircraft has added new requirements to this process. Ground test facilities are used to simulate the effect of inlet shape on the performance of an engine for cruise flight or other fixed conditions. While a static simulation of the inlet shape is adequate for most circumstances, the testing requirements of military fighter craft are greater because of their high maneuverability, a feature that is accompanied by intense and brief alterations of the airflow. These transient distortions can have a significant impact on performance as well as having the potential of causing structural problems. In addition to airflow fluctuation caused by flight maneuvers, the launching of weapons into the airstream can introduce transient turbulence and exhaust gas contaminants. Currently, none of these transient effects are fully examined in wind tunnel ground tests. One solution that has been proposed to simulate this problem in gound test facilities is an array of individually controlled wedges placed in a wind tunnel ahead of the test engine. The wedges could be opened and closed, not only to simulate the given inlet shape, but also, to generate transient distortions. Coupled with a system to introduce contaminants into the flow, all manner of transient and static inlet flow distortions could be simulated in a controlled environment. This paper focuses on implementing numerical and analytical models to capture the free stream turbulence and flow distortion downstream of variable angle wedges. The numerical scheme involves application of the shear stress transport approach (using the *k*-*ω* turbulence model). The analytical approach is based on Prandtl's mixing length hypothesis and Görtler's technique for treating far wake flow behind a bluff body. The numerical and analytical solutions are compared to the experimental measurements. It is shown that while a full blown CFD simulation can aid the design of a dynamic distortion generator, a well posed analytical model may provide an expeditious alternative.

I. Introduction

JET engines go through many development and testing steps before actually being installed in an aircraft. Once a bare engine is optimized for ideal operating conditions, it is submitted to ground testing to evaluate its performance in a simulated airframe. Various methodologies are employed in wind tunnel laboratories to induce distorted flow upstream of the compressor face and, in the process, simulate a particular inlet shape. These inlet flow distortions, at present, only simulate a fixed inlet configuration over a range of flow velocities. This scenario appears to be adequate for conventional aircraft engines that will rarely, if ever, be exposed to extreme environmental and operability conditions. However, high performance systems such as those used in military fighter craft exhibit a much broader operating envelope. This is owing to fighter craft being necessarily more powerful and maneuverable than commercial systems; some of their latest generation (e.g., the Joint Strike Fighter) supplement control surfaces with vectored thrust to

*Graduate Assistant, Department of Mechanical, Aerospace and Biomedical Engineering. Member AIAA.
†Boling Chair Professor of Excellence in Propulsion, Department of Mechanical, Aerospace and Biomedical Engineering. Associate Fellow AIAA.
‡Jack D. Whitfield Professor of High Speed Flows, Department of Mechanical, Aerospace and Biomedical Engineering. Member AIAA.

augment their maneuverability. The attendant changes in vehicle orientation are capable of distorting the inlet flow briefly but drastically, thus disrupting engine performance and possibly damaging components. An additional concern is the transient disruptions that occur when weapons are released; their wake and exhaust gases may be ingested when crossing path with the engine's inlet stream. Currently, there is no single, comprehensive and robust methodology to simulate either transient distortions and/or contaminants in the airflow during wind tunnel testing. This situation could greatly benefit from extensive testing in light of modern experimental and theoretical approaches.

One possible approach to simulating these transient effects is the use of an array of individually controlled wedges placed in the wind tunnel ahead of the engine being tested. This array could supplement existing methods for static testing and allow for more complex testing as well. The wedge angles could be varied such that the distortion element is closed completely, causing no significant flow disruptions, then selectively opened and closed to simulate a transient distortion. Simulated contaminants could also be introduced into the stream, with the wedges providing mixing effects that are similar to those produced by v-gutters in an afterburner. Such a wedge array could supplement existing testing methodologies while standing to provide an immensely broader range of simulation possibilities.

Past studies of airflow around bluff bodies have provided a starting point for mathematical models of wedges; in this investigation, it will be shown that both wind tunnel experiments and in-house CFD models can be corroborated by wake flow theory. A solid analytical model will be shown to simplify both the design of such an array and its usability by providing a more expedient, albeit approximate, predictive tool. In what follows, both numerical and analytical methods will be described and compared to contemporaneous test data.

II. Previous Work

Current wind-tunnel testing relies predominantly on static screens mounted upstream of the compressor face to generate the turbulence levels required to simulate an inlet shape.[1] As an alternative, fixed arrays of upstream airjets can also be used (see Fig. 1), but in either case the distortion shape is fixed, with the screens or airjets arranged to provide a specific disruption pattern that would correspond to the optimal operating conditions.

One possible solution to the problem of generating a transient flow distortion in a wind tunnel is through the use of an array of wedges which could be controlled individually (see Fig. 2). This would allow a variety of configurations and could generate both transient and static patterns to simulate a broad range of conditions. Supplemented with a system to feed "contaminated" air, the wedges could not only simulate the turbulence of weapons release, but also the effects of weapon exhaust gases introduced into the inlet stream.

Fluid flow past wedges and other bluff bodies has been studied for decades and is well characterized.[2-4] Flow separation and vortex shedding are well understood, though difficult to model due to their time-dependent nature. Airflow around aircraft inlets has also been extensively studied[5,6] as has the chemical and erosive effects of weapon discharge gases on the turbo machinery parts.[7]

To verify the basic concept, a study at Virginia Polytechnic Institute developed mathematical models to represent the wake characteristics of airflow around a wedge, then verified them with wind tunnel tests. First, a splitter plate was used, then a series of individual solid wedges.[8] This was followed by a second study focusing on more complex arrangements, with additional measurements being taken closer to the wedge.[9] Using a modified door hinge for the wedges

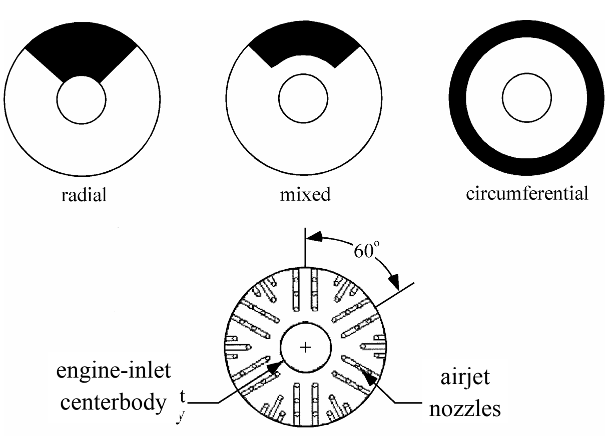


Fig. 1 Screen and airjet distortion generators.

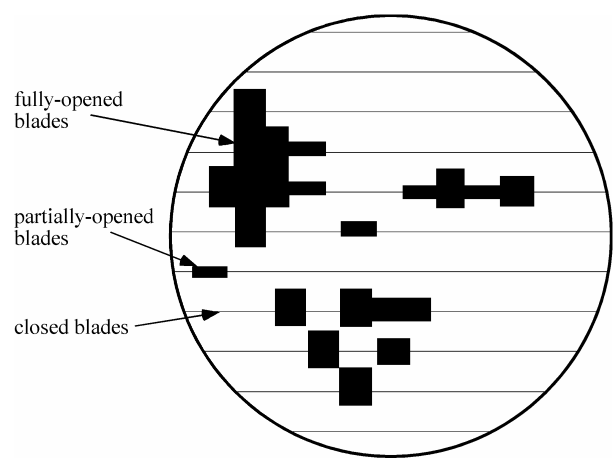


Fig. 2 Proposed array of actuated wedges.

(shown in Fig. 3), simple arrays were quickly set up and modified, ultimately demonstrating that wedges could be used to generate consistent and predictable results.

Rather than taking a trial and error approach to further develop an array of wedges, it seemed reasonable to try building a "virtual" one as a CFD model. To verify that a numerical solution would provide the same results as experimental methods, a simple CFD model was set up using the software *Wind* and a 2-D Cartesian grid to simulate the behavior of the wedges by Eddy.[9] To accomplish this, the grid was developed with wedges of the same geometry, albeit greatly simplified in the hinge area (see Fig. 4).

Once the grid was prepared, five test cases were executed and set to match the five single-wedge test cases corresponding to Eddy's experiments.[9] Wedge angles were set at half-angle openings of θ = 15°, 30°, 45°, 60° and 75° from the horizontal; the velocity value used in the wind-tunnel tests was 42 m/s, so a value of M = 0.125 was used in the simulation. After pre-processing the grid file and defining the boundary conditions, a script was used to capture solution data every 100 iterations. *Wind* was invoked with the two-equation "Shear Stress Transport" (SST) turbulence model, which includes the k-ω model parameters (where k is the specific kinetic energy of the turbulence flux and ω is the dissipation per unit of turbulent kinetic energy);[10] this arrangement was employed to best fit the unsteady nature of the flow around the wedge. Each test case was executed for a total of 10,000 iterations to ensure that enough samples were available, after convergence, for time-averaging of the results. Post-processing showed a favorable convergence rate.

The results were reassuring. Figure 5 shows typical

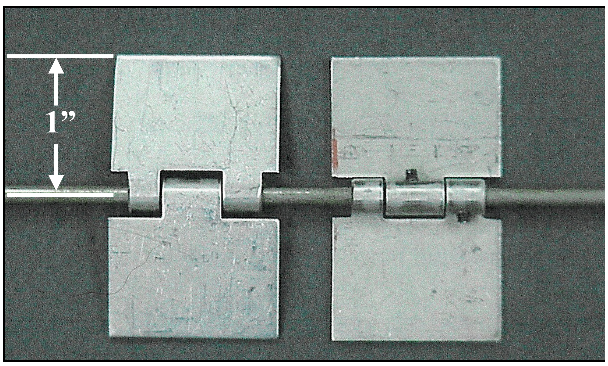


Fig. 3 Experimental wedge used by Eddy.[9]

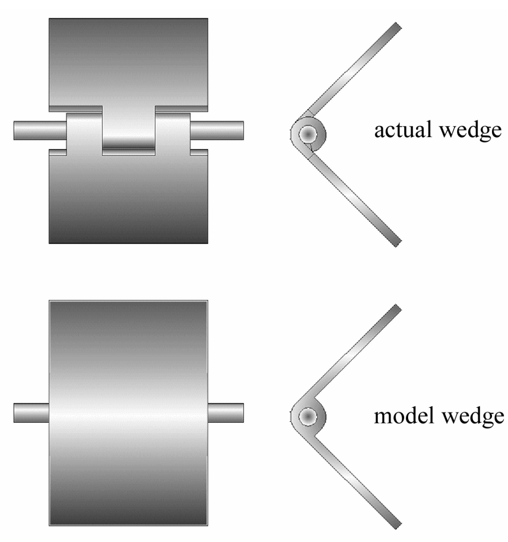


Fig. 4 Comparison of experimental and CFD wedge shapes. Note that the CFD model permits control of the wedge angle.

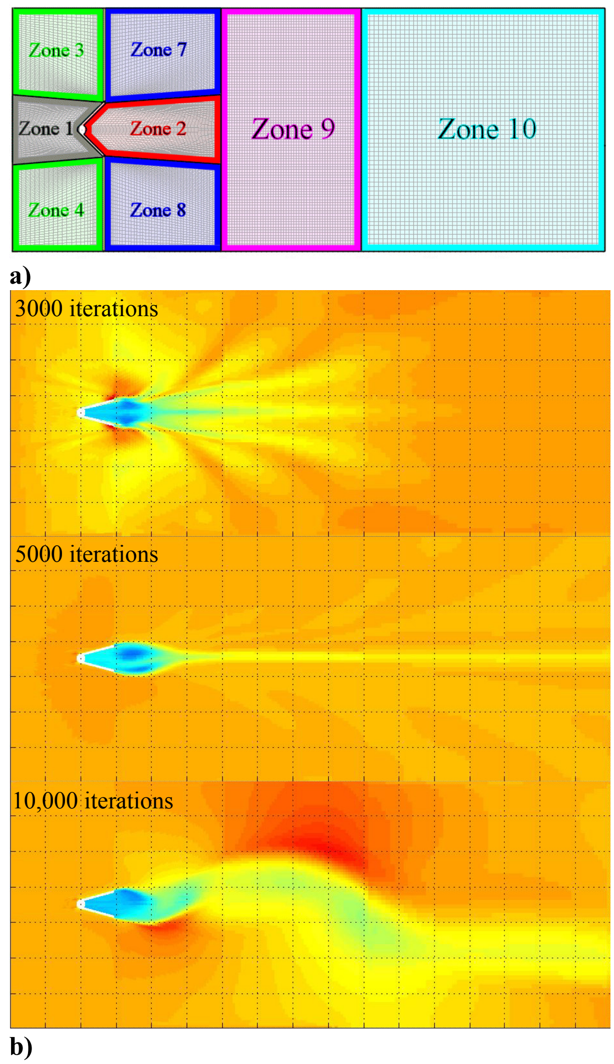


Fig. 5 Zone layout in a) followed in b) by the pressure evolution using *Wind*. Vortex shedding is shown as the solution develops.

development of the downstream flow for the 15° wedge test case. Note that Zones 1, 2, 5 and 6 define the boundaries of the wedge. Zones 3, 4, 7 and 8 fill in the corners while zones 9 and 10 extend the grid with increasing coarseness. (Zones 5 and 6 are too narrow to be seen in this resolution.) Initially the values were chaotic, but as the solution converged, distinct vortices began to form off the wedge tips. After further development, the vortices were seen to detach and begin alternating, thus giving rise to the expected bluff-body vortex shedding.

In order to measure pressure distortion, the pressure loss coefficient was calculated at axial positions that corresponded to the test planes in Eddy.[9] Figure 6 illustrates a comparison of data obtained from Eddy's work and our computations. The values appear to indicate a close match between experimental and numerical results that follow the 'dimpled' curve typical of flows past bluff bodies (see Yang, Tsai and Wang[2]). Overall, CFD predictions of pressure distortion levels seemed to faithfully represent the actual experimental measurements, especially as the wedge angle was increased. The preliminary success of the numerical model in reproducing the experimental test data has motivated the quest for project continuation in the hope of extending the model to more complex geometry.

However, it should be noted that a single 2-D slice of a single wedge at a fixed angle took approximately four hours to execute on a single 1.3 GHz processor with 384 MB of RAM and running Linux 2.4. While there are opportunities to increase efficiencies and reduce this time, and while *Wind* can be run on multiple-processor clusters, it is still reasonable to believe that modeling a full array of wedges would be challenging because of the time and processing power needed.

III. Analytical Approach

If CFD modeling is potentially too time consuming, then an alternative is to develop an analytical solution that can provide adequate predictive results in a more reasonable amount of time. A recent wind-tunnel test at Arnold Engineering Development Center (AEDC), using a radial array of porous wedges (to minimize shedding effects),[5,6] provided a wealth of measurements for a more complex flow to model (see Fig. 7).

Experimental data has shown the time-averaged flow at different planes across the flow, and CFD results have shown the "instantaneous" flow around and downstream of the wedge. In order to study the effects of various configurations, it must be realized that only the flow characteristics directly upstream of the

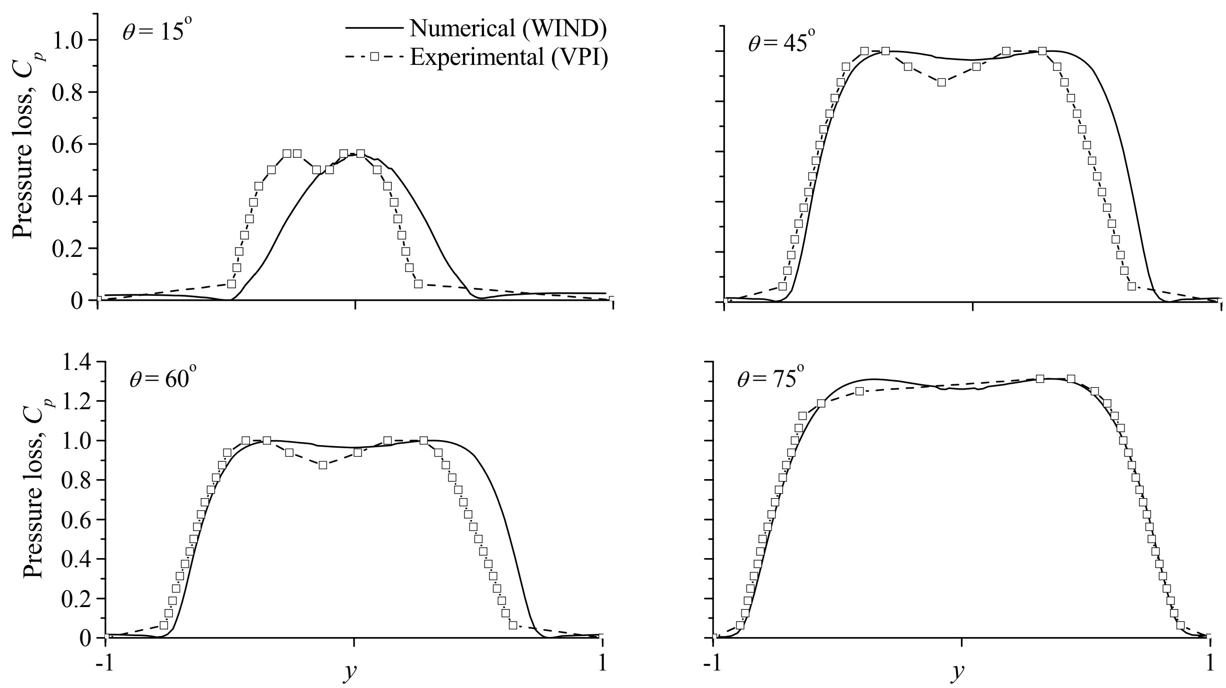
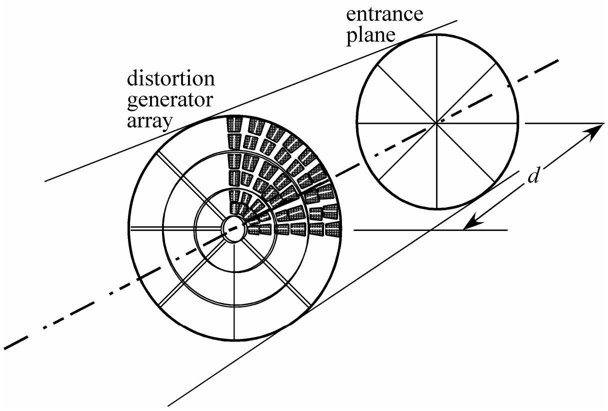
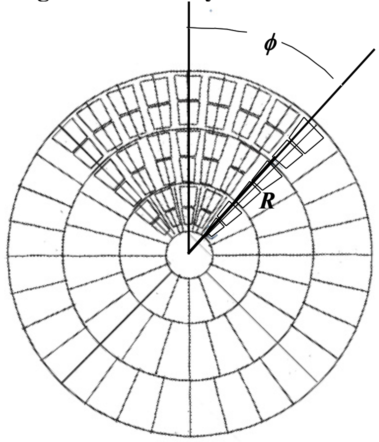


Fig. 6 Sample pressure distortion measurements showing the pressure loss coefficient at an arbitrary axial position. Here results are collected 3 inches downstream of a wedge with 1-inch flaps. The vertical coordinate y is normalized by 2 inches.



a) distortion generator array



b) detail of the radial array

Fig. 7 Radial distortion generator array.

compressor face are essential to examine. As shown in Fig. 8, the main focus should be to model the far-wake characteristics for a given distortion element. From the earlier studies, it is clear that turbulence will dominate the behavior of the flow. It can also be shown that the far wake velocity profile is independent of the body shape and that, instead, it depends only on the size and drag of the body.[11] If the array is placed far enough upstream of the compressor face for these "far-wake" assumptions to apply, then predicting the velocity and pressure profiles at the entrance plane becomes relatively simple.[10-12]

To start, we consider a single element suspended in a steady, uniform flow at velocities low enough to treat as incompressible (i.e. $M < 0.5$). With density constant, the continuity and momentum equations for the x-y plane can be expressed as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=\frac{1}{\rho}\frac{\partial \tau}{\partial y} \tag{2}$$

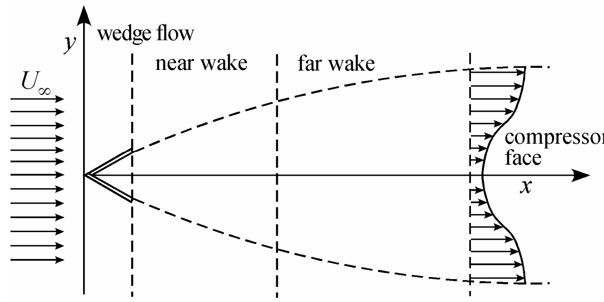


Fig. 8 Basic flow geometry.

The Prandtl mixing length (ℓ) can be used to estimate the shear stress via

$$\tau=\rho\ell^2\left|\frac{\partial u}{\partial y}\right|\frac{\partial u}{\partial y} \tag{3}$$

Now the velocity defect is introduced; (this only applies to large enough distances that the defect is small compared to the free stream velocity). Define this as

$$\hat{\boldsymbol{u}}=U_\infty \boldsymbol{i}-\boldsymbol{u} \tag{4}$$

Then, the wake flow can be characterized from the boundary value problem

$$U_\infty\frac{\partial \hat{u}}{\partial x}=-2\ell^2\frac{\partial \hat{u}}{\partial y}\frac{\partial^2 \hat{u}}{\partial y^2} \tag{5}$$

$$\hat{u}(x,y)\to 0;\ \ y\to\pm\infty \tag{6}$$

$$\frac{\partial \hat{u}}{\partial y}=0;\ \ y=0 \tag{7}$$

Next, the drag force can be determined by applying the momentum integral balance, namely,

$$\begin{aligned}D&=\tfrac{1}{2}\rho C_D(wh)U_\infty^2\\&=h\rho\int_{-\infty}^{\infty}\hat{u}(U_\infty-\hat{u})dy\approx h\rho U_\infty\int_{-\infty}^{\infty}\hat{u}dy\end{aligned} \tag{8}$$

Here, w is the fixed spanwise width and h is the streamwise height of the generator element; the latter varies with the opening angle of the wedge. Having

$$\int_{-\infty}^{\infty}\hat{u}dy=\tfrac{1}{2}\rho C_D w U_\infty \tag{9}$$

a simple similarity solution can be formulated using well-known techniques. By positing that

$$\hat{u}=U_\infty(c_D w/x)^{\frac{1}{2}}f(\eta) \tag{10}$$

One can set $b=B\sqrt{c_D wx}$ and $\eta=y/b$, where b is a measure of the width of the fully-developed wake far downstream of the generator.[11] These transformations reduce the problem to an ordinary differential equation

$$\tfrac{1}{2}(f+\eta f')=\left(2\beta^2/B\right)ff'' \tag{11}$$

where $\beta \equiv \ell / b$. Assuming that the mixing length is of the form $\ell = \alpha b(x)$. Then the solution for the velocity deficit can be deduced to be

$$\hat{u} = \tfrac{1}{18} U_\infty B \beta^{-2} \left(C_D w / x\right)^{\frac{1}{2}} \left[1 - \left(y / b\right)^{\frac{3}{2}}\right]^2 \qquad (12)$$

where $B = \sqrt{10}\ \beta$.

This formulation provides a solution for a single distortion element and can be enhanced with the application of the Prandtl-Glauert rule to compensate for the compressibility of higher velocity flow. By assuming that overlapping wakes of different elements have a very small impact on each other, an array of elements can be modeled by merging of individual wakes, with the largest values dominating.

Using data from an earlier experiment to estimate drag coefficient, this formulation was found to provide good agreement with experimental data (see Fig. 9).

IV. Concluding Remarks

A dynamic distortion generator comprised of an array of individually controlled wedges could be used to greatly extend the capabilities of ground test facilities. Such an array would allow the generation of the same static test flows currently used as well as a variety of dynamic distortions not practical with existing methodologies. A solid analytical model provides fast and reliable configurations to generate specific distortions, leaving the laboratory facility researcher with the ability to conduct further tests.

Acknowledgments

This work was sponsored partly by the University of Tennessee, UTSI, and partly by the Arnold Engineering Development Center, under Contract No. F40600-00-D-0001. The second author wishes to express appreciation for additional support from the Edward J. and Carolyn P. Boling Chair of Excellence in Advanced Propulsion, University of Tennessee.

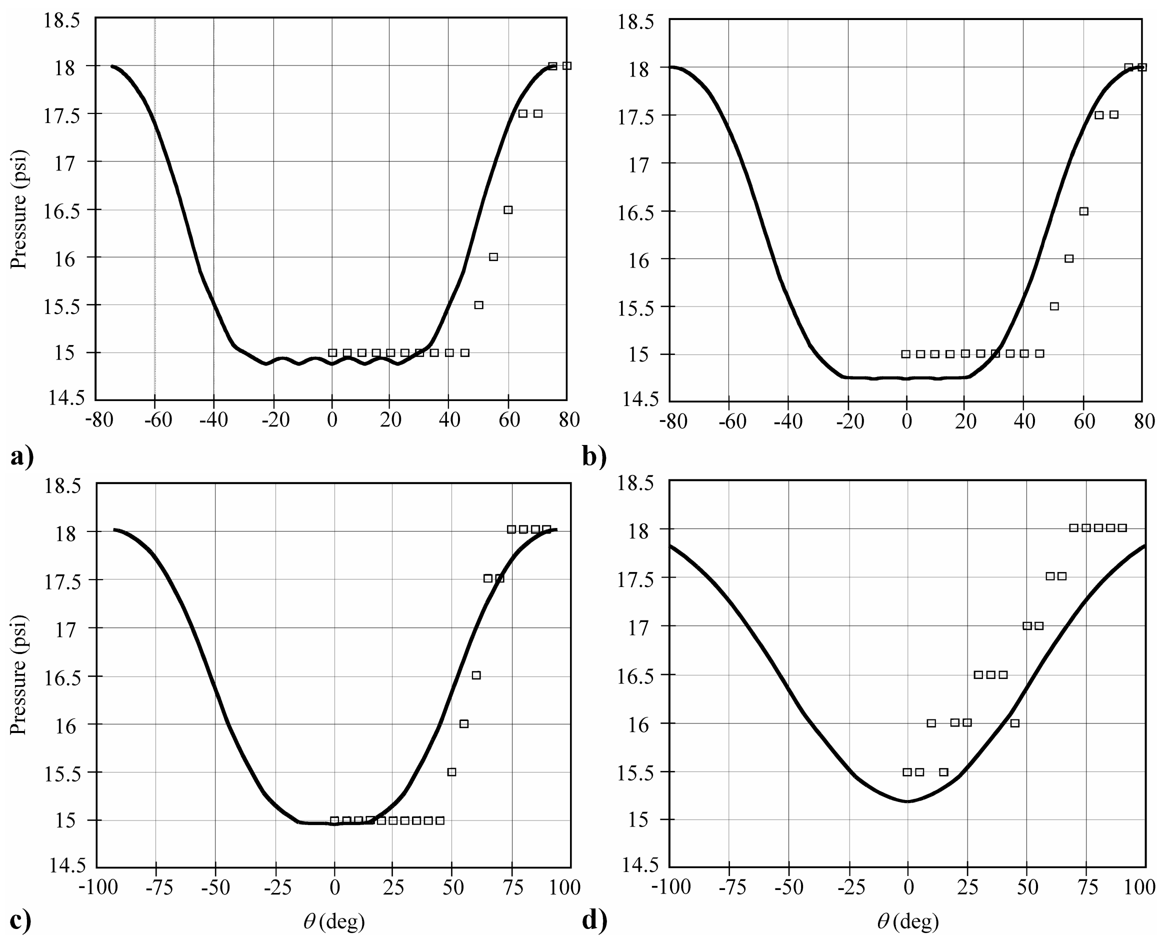


Fig. 9 Comparison of analytical results (solid line) to measured distortion (scatter line). Here the total inlet pressure is shown versus angular position for an entrance plane-to-array distance of 16 inches and a radius of a) 8, b) 6, c) 4, and d) 2 inches.

References

[1]Davis, M., Baker, W., Power, G., and Beale, D., "A Proposal for Integration of Wind Tunnel and Engine Test Programs for the Evaluation of Airframe-Propulsion Compatibility Using Numerical Simulations," *Journal of Engineering for Gas Turbines and Power,* Vol. 124, No. 3, 2002, pp. 447-458.

[2]Yang, J.-T., Tsai, G.-L., and Wang, W.-B., "Near-Wake Characteristics of Various V-Shaped Bluff Bodies," *Journal of Propulsion and Power,* Vol. 10, No. 1, 1994, pp. 47-55.

[3]Parkinson, G. V., and Jandali, T., "A Wake Source Model for Bluff Body Potential Flow," *Journal of Fluid Mechanics,* Vol. 40, No. 3, 1970, pp. 577-594.

[4]Gerrard, J. H., "The Mechanics of the Formation Region of Vortices Behind Bluff Bodies," *Journal of Fluid Mechanics,* Vol. 25, No. 2, 1966, pp. 401-413.

[5]Bruns, J. E., and Smith, C. F., "Installed F/A-18 Inlet Flow Calculations at High Angle of Attack," *Journal of Propulsion and Power,* Vol. 10, No. 1, 1994, pp. 110-115.

[6]Beale, D. K., "Improving Information Productivity and Quality in Turbine Engine Ground Testing," AIAA Paper 2001-0163, 2001.

[7]Hamed, A., "Influence of Secondary Flow on Turbine Erosion," *Journal of Turbomachinery,* Vol. 111, No. 3, 1989, pp. 310-314.

[8]Jumel, J., King, P. S., and O'brien, W. F., "Transient Total Pressure Distortion Generator Development," Arnold Engineering Development Center, Final Technical Rept. F40600-94-K-0018, Tullahoma, TN, July 1999.

[9]Eddy, G. L. J., "Study of Steady-State Wake Characteristics of Variable Angle Wedges," M.S. Dissertation, Virginia Polytechnic Institute and State University, 2001.

[10]Wilcox, D. C., *Turbulence Modeling for CFD*, DCW Industries Inc., La Cañada, CA, 1994.

[11]Schlichting, H., *Boundary-Layer Theory*, 7th ed., McGraw-Hill, New York, 1979.

[12]Flandro, G. A., Private Communication, 2004.