AIAA JOURNAL
Vol. 36, No. 5, May 1998

Validation Issues for Engine–Airframe Integration

John A. Benek* and Edward M. Kraft†
Micro Craft, Inc., Tullahoma, Tennessee 37388-0370
and
Rodney F. Lauer‡
Sverdrup Technology, Inc., Arnold Air Force Base, Tennessee 37389-6001

The engine–airframe integration methodology is reviewed and some process deficiencies are identified. The use of computational fluid dynamics as an alternative simulation source is discussed. An approach toward combining computations with the existing integration methodology is suggested. Validation issues associated with the computational procedures are examined. Computational fluid dynamics as a source of test facility corrections is found to be well developed, but validation efforts remain ad hoc. The use of computational fluid dynamics to evaluate inlet drag increments and to generate high-resolution total pressure maps is discussed. Related validation issues include the simulation of unsteady flow with steady-state approximations.

Nomenclature

D = drag force
e = total measurement error
M = Mach number
m = density
P = general parameter
Re = Reynolds number
T = thrust force
α = angle of attack

Introduction

ENGINE–AIRFRAME integration in modern jet-powered fighter aircraft consists of defining and optimizing the external aerodynamics as they are affected by the operating engine, as well as defining and optimizing the engine operability and performance within the flight envelope. These interactions are captured in the vehicle performance model, an engineering model, in which the underlying physics are lumped into various model parameters. The values of these parameters are determined from databases generated by simulations. The databases consist of baseline data and increments that account for parameter variations, e.g., Mach number, Reynolds number, and attitude, from the baseline. Increments also can include corrections for simulation errors such as geometric infidelities or test facility effects. For example, a specific model parameter P is constructed as

$$P_{\text{model}} = P_{\text{baseline}}(M_0, Re_0, \alpha_0, \ldots) + \Delta P(M, Re, \alpha, \ldots)$$

where P_{baseline} is evaluated at the baseline conditions indicated by the subscript 0, and ΔP is the increment describing the change in P.

The fundamental complication in the use of simulation to develop the performance-model database is that the engine does not scale. Hence, full-scale engine testing is required to identify the engine performance. These tests do not account for the installed effects the engine experiences in flight. On the other hand, the aerodynamics scale according to well-established rules, and thus, scale-model testing in wind tunnels is used to obtain the basic aerodynamic performance, including control effectiveness, and trim properties. Wind-tunnel tests, which include inlet total pressure recovery, engine-face distortion, calibration of inlet variable-geometry component schedules, inlet drag, and exhaust jet effects, also characterize the airframe–propulsion-system integration. Thus, the vehicle performance-modeling process becomes a decomposition of effects, the propulsion system characterized by full-scale simulations and the aerodynamics characterized by scaled simulations. The integration process must combine these two sets of simulations and account for the mutual interactions of the inlet and the propulsion system with the external aerodynamics.

The separation of engine and aerodynamic performance introduces multiple sets of reference conditions. The engine test reference is the condition of the flow at the engine face. The engine performance usually is based on the freestream conditions adjusted for total pressure recovery of the inlet duct and other discrepancies between the freestream and the engine-face conditions. Unfortunately, the actual-flight engine-face conditions are unknown before completion of the inlet tests. The aerodynamic reference conditions are wind-tunnel freestream conditions, flow through aerodynamic model mass flow ratio, and nozzle pressure ratio (NPR). Operational flight reference conditions are chosen to be close to the actual flight conditions, which include mass flow ratio and NPR as functions of Mach number. The modeling process assumes that actual flight performance can be obtained from the performance at the reference operating conditions by the addition of small increments. These increments are representative of the differences between the reference operating conditions and the actual flight conditions. The multiplicity of reference conditions requires that suitable data be obtained that will allow the transformation of the various data sets to the operational conditions in a reasonable and accurate manner.

The key tool for the combination of the various data sets is the Force Accounting System[1] (FAS). The FAS separates the total forces and moments acting on the aircraft into components acting on specific parts of the configuration. Because the decomposition of forces is arbitrary, many accounting systems are possible. The basic assumption implicit in the FAS is that the components are independent and can be combined linearly to obtain the total forces acting on the configuration. Therefore, each component force must be included once and only once in the performance-model database. Because many types of models are used to develop the database, the same force component can be included redundantly in several sets of test data and must be removed appropriately.

There are three primary interactions of interest to engine–airframe integration: the inlet and engine airflow, the reaction of the engine to the inlet flow, and the afterbody and exhaust jet flow. As Fig. 1 indicates, the major considerations arising from these interactions are the engine performance, the inlet drag, the engine stability, and the nozzle/afterbody performance.

Presented as Paper 96-2031 at the AIAA 27th Fluid Dynamics Conference, New Orleans, LA, June 17–20, 1996; received Oct. 22, 1996; revision received Nov. 13, 1997; accepted for publication Nov. 16, 1997.

*Manager, Integrated Test and Evaluation. Associate Fellow AIAA.
†Group Vice President. Fellow AIAA.
‡Engineering Specialist.

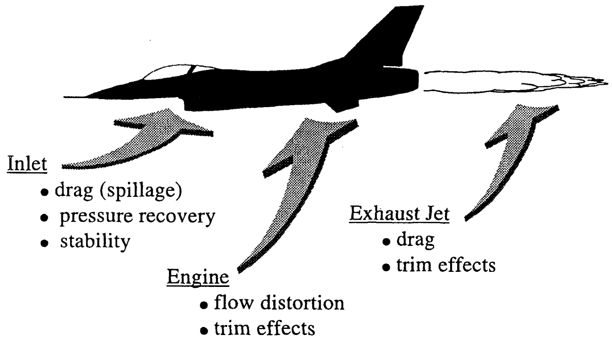

Fig. 1 Engine–airframe integration concerns.

The inlet and engine performance are closely related: The character of the flow at the compressor face largely determines the engine performance. One important measure of this performance is the total pressure recovery. The higher the recovery, the less flow energy lost to drag and the less energy the engine must supply to compress the air to a higher pressure. Another phenomenon of interest is inlet stability. Instability can occur whenever the engine speed decreases. The inlet airflow decreases with an attendant increase in static pressure. The resultant adverse pressure gradient in the inlet can become sufficiently large to cause a reversal of the inlet flow. This flow reversal can become extensive and further reduce the inlet airflow causing an increase in inlet flow spillage. If the freestream is supersonic with an external compression inlet, the inlet normal shock moves upstream from the inlet entrance. (A mixed compression inlet may unstart.) Because the engine continues to operate, the engine airflow is greater than the inlet airflow and causes the inlet duct static pressure to decrease. The inlet flow then reestablishes itself, and the process begins again. At subsonic freestream speed, this instability often is referred to as duct rumble and at supersonic speeds as inlet buzz. In either case, the planar pulse at the engine face may result in an engine compressor stall.

Inlet drag is that portion of the vehicle drag that changes with the inlet airflow. This drag component may be defined during the aerodynamic model test by varying the flow through the inlet ducts by means of various restrictions. For variable geometry or variable bleed inlets, this component must be defined in a separate test. The evaluation of inlet drag is discussed in more detail in the section on inlet testing.

Engine stability concerns the reaction of the engine to the flowfield at the compressor face. Velocity gradients as characterized by engine face, total pressure, and distortion indices are the primary parameters used to predict these conditions. Deviations from uniform, smooth flow are called distortions; various measures, i.e., distortion indices, have been developed to serve as a figure of merit for the inlet flow. They are used to define the engine-face nonuniformities at various test conditions. Because the engine responds to the flow distortions on a time scale related to the engine rotation speed, instantaneous values are also important. Engine stability problems frequently manifest themselves as compressor stall.

The engine exhaust is a hot, high-speed jet of combustion products whose presence alters the airflow about the aft portion of the aircraft. The jet changes the flow over the afterbody of the vehicle. The net result is that the jet changes the aircraft drag and also can affect the trim properties. The magnitude of these changes is a function of the nozzle pressure ratio and the nozzle geometry.

In the following sections, we examine the established engine–airframe integration methodologies in more detail and identify areas where computational simulations can supplement and improve the accuracy of the information obtained from the traditional experimental approach. We also discuss validation issues associated with the development of computational simulations and their use within the existing procedures.

Wind-Tunnel Simulation

The aircraft development process includes at least three separate types of wind-tunnel models to define the forces required by the FAS. Figure 2 indicates the three most common model types: aerodynamic, inlet, and jet effects. The aerodynamic model usually consists of a fixed inlet and nozzle geometry in a flow-through configuration. This model type typically is sting supported, and therefore, the aft geometry must be modified to accommodate the sting and force balance. The flow through the inlet and nozzle yields only one mass flow rate and NPR per test condition, i.e., M, Re, and model attitude. As previously noted, the inlet drag effects can be determined using this model for simple, zero-bleed inlets.

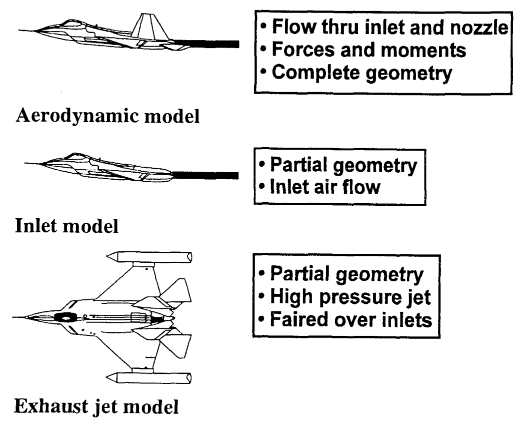

Fig. 2 Wind-tunnel model types.

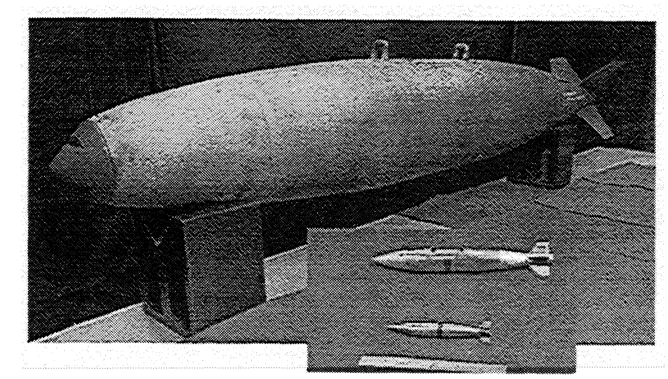

Fig. 3 Geometric fidelity with decreasing model scale.

The inlet model accurately represents the inlet geometry and bleeds but has simplified or incomplete external geometric fidelity. The geometric simplifications begin far enough aft of the inlets so as not to influence the inlet flow. This model type is used to determine the inlet duct pressure recovery and the engine face distortion over a range of inlet mass flows. The inlet drag depends on inlet mass flow and can be obtained from an integration of pressures measured at various locations on the inlet and adjacent surfaces. Frequently, the pressure measurement spacing is not sufficiently fine to resolve pressure gradients, and thus, the computed drag values are compromised. Alternatively, inlet drag can be determined from a specially designed inlet drag model.

The jet-effects model includes an accurate representation of the aircraft aft geometry including the engine nozzles. The exhaust jet is simulated by a high-pressure jet. In some models the simulated jet efflux can be heated to establish temperature effects, but normally, a cold, high-pressure air jet is used. In addition, this model type frequently is used to obtain sting support effect corrections for the aerodynamics model.

Testing Complications

The separation of propulsion and aerodynamic simulation introduces additional issues that must be balanced in the total modeling methodology. Tradeoffs that arise from the imperfect nature of ground simulation frequently are categorized as either model fidelity (geometric) or wind-tunnel interference effects, as well as test facility limitations. Geometry effects result from compromises in model fidelity because of the inability to include small features in the scale models: The smaller the model, the fewer geometric features that can be practically resolved. Figure 3 illustrates how geometric features are lost as model scale decreases, even for a simple geometric

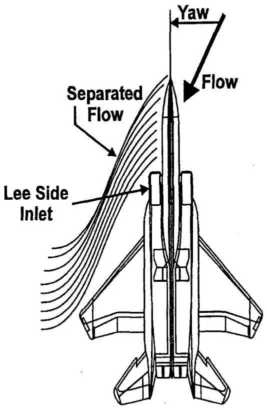

Fig. 4 Yawed aircraft inlet flowfield.

configuration. The figure shows a full-scale bomb, and the inset shows a 10% and a 5% model of the same geometry. Geometric details such as the mounting lugs and nose grooves are lost at the smaller scales. The requirement that a model be supported within the test facility imposes additional geometric compromises to accommodate the support structure. Interference effects result from alterations to the flowfield introduced by the presence of the model support structure and wind-tunnel walls. Interference effects become more predominant as the model size increases relative to the wind-tunnel dimensions and as the airflow speed increases from subsonic through transonic flow. Test-facility limitations arise from the inability to duplicate all values of the key simulation parameters. For example, few wind tunnels can match flight values of the Reynolds number, and extrapolation techniques must be employed to correct wind-tunnel data to operational reference conditions. Dynamic effects such as aircraft maneuver cannot be simulated experimentally.

Because the wind tunnel cannot duplicate the flight Reynolds number, additional compromises must be made in the experimental simulations. For example, consider the flow about configurations such as the F-15 with inlets on both sides of the fuselage. As indicated in Fig. 4 at yawed conditions, the flow on the leeward side of the fuselage is a strong function of the Reynolds number. Because of the mismatch in Reynolds number, the wind-tunnel simulation may present the inlet with a flow very different from that experienced in flight. The wind-tunnel Reynolds number is lower than the flight value, causing the model boundary layer to be thicker than at flight conditions. Thus, the scale-model, boundary-layer diverters will not remove the thicker layer unless the gutter height is increased. Unfortunately, increasing the gutter height only provides an adequate correction for a small portion of the entire test matrix. To properly simulate the gutter height, it would be necessary to make it vary as a function of α (angle of attack), β (side-slip angle), Mach number, and Reynolds number. This is clearly impractical. Additional issues associated with the state of the boundary layer also affect the simulation. For example, the difference between laminar and turbulent flow can significantly affect separation in the inlet, and the measured inlet performance by as much as 10%. The boundary-layer state also depends on the level and type of test-facility environmental disturbances, such as freestream turbulence and acoustics.

Inlet Testing

As we noted in the Introduction, there are many types of wind-tunnel simulations used in the engine–airframe integration process. We select inlet testing as representative and use it to illustrate some of the details of experimental simulation and the associated issues. The inlet model is used to determine inlet total pressure recovery and engine-compressor-face-distortion indices and may be used to determine inlet drag. The inlet model is frequently of larger scale than the aerodynamic model, and thus has greater fidelity in the geometry forward of the inlets as well as in the inlet and ducting. It has less external detail aft of the inlets. The increase in size helps to reduce the Reynolds number mismatch between wind-tunnel and flight conditions.

The inlet total pressure recovery and distortion indices are based on weighted averages of pressure measurements. It is not feasible to make these measurements at the engine compressor face, especially on full-scale or flight-test configurations. Instead, they are made slightly upstream at a standard location called the aerodynamic interface plane (AIP). The pressure measurements are necessarily sparse so that the pressure rakes have a minimum blockage and interference with the duct flow. The rakes usually consist of total pressure probes spaced radially, with eight such rakes installed at equal azimuthal angles. Some tests include flow angularity probes to characterize swirl. Because of the limited number of spatial locations, small but important features of the flow can escape detection. In addition, the distortion measured on the model may not scale to the flight vehicle because effects such as inlet flow separation may not be representative of flight conditions because of the Reynolds number mismatch between the wind-tunnel and flight conditions. The compressor can also affect the inlet flow and separation location; compressor effects frequently are not simulated in wind-tunnel tests.

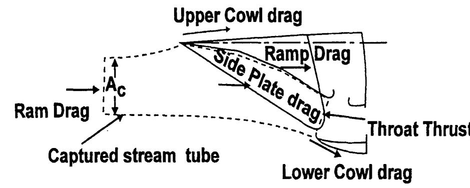

Fig. 5 Inlet drag force balance.

The measurements produce maps of the total pressure at the AIP and various weighted averages of the total pressures[2] are used to characterize the inlet recovery. The flow at the AIP is intrinsically nonuniform and unsteady. For the engine to respond, the nonuniformities must have spatial and temporal scales comparable to those of the engine. The deviations from uniform steady flow are called flow distortions. Because the number of total pressure maps that are obtained during an inlet test is large, distortion indices have been developed to simplify the analysis. Only flow conditions that produce indices larger than some empirically determined value are examined extensively. Because engine stability has been determined to be a function of the time-dependent distortion, the time-dependent distortion indices will be considered in more detail. Unsteady or dynamic distortion indices are obtained from a complex analysis of statistical properties of peak instantaneous values of the distortion indices over some time interval (typically, 5–10 s during model testing).

Because the FAS assigns the ram drag term to the propulsion system,[1] it must be removed from the inlet model data. For convenience, we denote the ram drag as

$$T_{\text{ram}} = (mV)_\infty$$

T_{ram} is computed from measured values of the inlet mass flow and the freestream Mach number. The inlet or spillage drag is an increment that accounts for the change in drag as a function of inlet mass flow and excludes the ram drag term. Referring to Fig. 5, it is computed as

$$\Delta D_{\text{inlet}} = D_{\text{inlet}} - (D_{\text{inlet}})_0$$

where

$$D_{\text{inlet}} = [D_{\text{ramps}} + T_{\text{throat}} - T_{\text{ram}}] + [D_{\text{upper}} + D_{\text{lower}} + D_{\text{side}}]_{\text{cowl}}$$

Whenever the pressure or curvature of the inlet geometry changes more rapidly than the spacing of the pressure orifices can resolve, the resulting estimate of the inlet drag can contain significant error.

Computational Fluid Dynamics Simulations

In the foregoing, we described the framework in which engine–airframe integration is performed. Some of the areas are noted in which the traditional experimentally based simulations are deficient. In the following sections, we suggest ways in which computational fluid dynamics (CFD) can be used within the existing, well-established methodology to complement wind-tunnel simulations. We also discuss some of the associated validation issues. Our focus is on procedural validation, in contrast to physics validation, which has an extensive literature, e.g., see Refs. 3–9. For the sake of argument, we assume that such efforts have established the domain of validity of CFD predictions of the basic fluid physics such

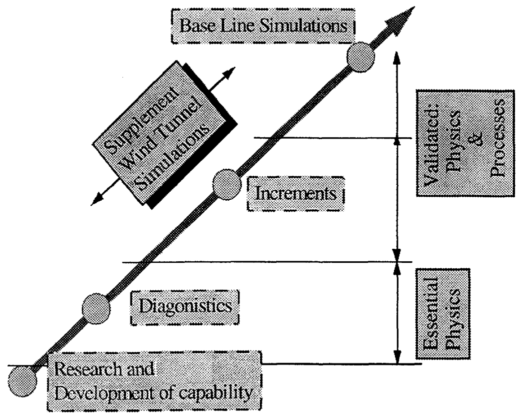

Fig. 6 Evolution of CFD applications.

as separation and reattachment, and vortex formation and bursting. Procedural validation issues are simulation compromises that neglect some of the nondominant physics in the interests of expediency. For example, inlet distortion indices consist of spatial and temporal averages of pressure data. It is common practice to use steady-state numerical simulations to represent the temporal averages. Other procedural issues associated with numerical simulation include model selection, e.g., inviscid, viscous, and turbulent; grid size; geometric resolution; and convergence criteria.

Figure 6 shows the three stages of the evolution of CFD applications: diagnostic, incremental effects, and baseline data generation for the performance-model database. CFD simulations have such a high information content that visualization tools[10] are required to make the results intelligible. Because these tools allow computed flow parameters to be presented in many combinations and to be seen from many viewpoints, CFD is an ideal diagnostic tool. It can be used to determine the origin of disturbances and flow instabilities, and the location, extent, and strength of flow structures, e.g., Refs. 2 and 11–13. The absolute accuracy requirement is not high; the basic requirement is that the predominant physics be reasonably represented. This modest accuracy requirement exists because the diagnostic results generally are verified by additional analysis or experimental simulation. Validation concerns are minimal. Because the diagnostic use of CFD does not contribute directly to the generation of databases, it is not considered further.

The ability of CFD to provide incremental data is already established in some areas relevant to engine–airframe integration. Note that highly accurate increments can be obtained from simulations that are less accurate than those required for baseline data. This is because portions of the error tend to cancel. For example, a typical increment is constructed from two terms:

$$\Delta P = (P + e)_2 - (P + e)_1$$

the error (biases and precision) and the true value of the parameter. For carefully crafted simulations, the bias part of the errors tends to cancel, leaving a smaller error term, so that

$$\Delta P = \Delta P_{\text{actual}} + \Delta e$$

Thus, the error in the increment is less than the error associated with the terms from which it is derived. This cancellation effect is well established for experiments; similar observations have been made for computational increments and are illustrated in the next section.

CFD and Facility Corrections

CFD-generated increments of interest to engine–airframe integration include the estimation of transonic wall interference. Sickles and Erickson[14] demonstrated excellent agreement between measured and computed corrections to an aircraft configuration. They demonstrated that data obtained from viscous simulations could be substituted for experimental data and used within the well-established correction methodology. (See Ref. 15 for a comprehensive review of transonic wall-interference correction methodology.) Martin et al.[16] applied the same computational methodology to the Space Shuttle launch configuration. Their calculations of normal force and pitching-moment coefficient increments were in good agreement with values obtained from measurements at several model scales.

Willhite et al.[17] computed sting interference corrections, including the effects of geometry alterations for the F/A-18E/F. These effects also were determined experimentally from a wing-tip-supported exhaust jet model. The model was altered with a dummy sting support and the associated afterbody modifications. A careful examination was made of measured and computed pressures, as well as computational and data reduction procedures. The computed and measured increments compared well over a wide range of flow conditions.

Hinkleman (unpublished results) compared drag increments, accounting for the mismatch between wind-tunnel and flight Reynolds number. His drag calculations for a transport configuration, which was tested in both a conventional and a high-Reynolds-number wind tunnel are in excellent agreement with the measured values.

The major stumbling block to validation of the wind-tunnel correction methodology is that the majority of data is either classified or proprietary, and therefore not generally available. If some of these data can be made available for review, a selection can be made and a validation database constructed at minimum cost. A common set of validation data is needed before a comprehensive assessment of simulation methodology can be made.

Establishment of a methodology for computational increments will make practicable the evaluation of increments that cannot be obtained readily by experiment. Such increments include corrections for geometric compromises such as oversized boundary-layer gutters or wind-tunnel support mechanisms. Validation of correction procedures for geometric effects will require careful study because different grids must be used for each geometry, and subtle resolution issues can introduce unsuspected biases. It also should be possible to make Reynolds-number corrections for inlet swirl and total pressure maps (Fig. 7).

CFD Inlet Simulations

The inlet drag increment (see Fig. 5) is computed from an integration of pressure forces measured on the inlet. Frequently, there is an insufficient number of measurements for an accurate estimate of this term. Improved estimates can be obtained by taking advantage of the higher resolution inherent in CFD simulations. In addition, the surfaces affected by the inlet flow variation may be larger than the measurement area; CFD simulations allow one to integrate over the entire vehicle so that no affected area is missed. In addition, CFD can provide an estimate of the skin-friction drag. The computed inlet drag increment can be used as either a correction to the measured increment or as a replacement for it. The choice depends upon the domain of validation of both phenomenology models and the correction methodology. A continual check can be obtained as a natural part of the process. Computed and measured pressures can be compared pointwise to assess the accuracy of the computed distributions. Similarly, values of the inlet drag obtained from measured and computed values can be used to check the methodology.[17]

Inlet total pressure maps are based on a limited number of measurements, which may allow small but important flow features to escape detection. Improved resolution maps can be obtained from CFD simulations. However, there are some interesting validation issues associated with the computation of these maps. One of these

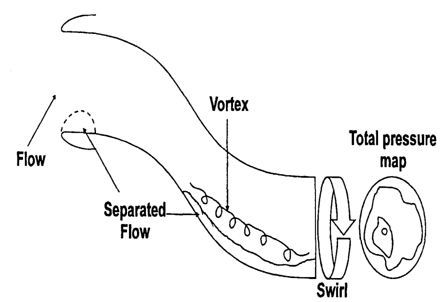

Fig. 7 CFD inlet simulation.

is the use of steady-state approximations to simulate time-averaged measurements of an intrinsically unsteady flowfield. Let us assume that the experimental data are averaged over a sufficiently long time interval to be time independent. Let us also assume that CFD can accurately compute the unsteady flow. If the steady-state approximation is used, will the same time-averaged quantities be predicted? This is a fundamental issue because steady-state solutions are currently employed to compute inlet total pressure maps. Although there is some evidence that the procedure works,[2,18] no systematic investigation to establish the validity of the procedure or to establish its domain of application has been undertaken. An additional complication is associated with the use of acceleration schemes based upon local time stepping.[19] It is assumed that such a procedure will produce the same converged solution as an unaccelerated method for steady-state flows. Is there an effect on the approximation of unsteady flow with this type of acceleration?

In addition, no systematic studies of the details of the computational procedures for inlet flow have been made. For example, curved inlet ducts with changing cross section can be expected to have regions of separated flow and swirl so that the flow is inherently unsteady. The unsteady features prevent the use of the usual steady-state convergence criteria that are based upon the reduction of the residual norms toward machine zero. Other indicators must be used. For example, Nichols, in a private communication regarding results reported in Ref. 20, used the iteration history of the variation of the distortion index. When the change in the values of the indices with increasing iteration fell below some arbitrary value, the simulation was assumed to have converged to a steady-state value. The applicability of this procedure to flow with greater unsteadiness has not been ascertained.

Time-accurate computations may be expected to be necessary for the simulation of unsteady (dynamic) distortion indices. This type of simulation presents a significant challenge to validation. First, from a phenomenological perspective, there is a paucity of carefully crafted experiments from which to establish a validation database. Because it is not practicable to attempt to reproduce the exact time history of unsteady flow, the analysis must concentrate on matching the statistical properties of the flow parameters. Which statistical properties are important to reproduce is an open area for investigation, and validation criteria need to be established by theoreticians, experimentalists, and the engineers who use the data.

Second, there is no established computational procedure for simulation of unsteady flow. For example, how long should the simulation be run before the starting computational transients no longer affect the solution? Suhs and Jordan[21] have studied the flow over cavities and have concluded that six time characteristics are sufficient to eliminate the computational transients for the configurations they investigated. They defined a time characteristic for their studies as the time it takes a fluid particle to travel the length of the cavity at the freestream velocity. Unfortunately, this criterion is not directly applicable to inlet flow. A related issue is the determination of the optimal initial conditions from which to start the simulation. Experiments by Suhs and Jordan examine several starting options for cavity flow. They conclude that the initial conditions for this type of flow are important in determining the rate at which the initial transients decay. A further examination of unsteady flow methodologies was conducted by Aboulmouna et al.[22] for simulation of the B-1B weapons bay. Their study includes comparisons of the forces and moments on stores located in the bay and an examination of the effects of various averaging strategies on the computed values. They examined the statistical properties of the flow computations to determine if stationary conditions were achieved. The comparisons with experiment were encouraging. In another effort, Aboulmouna et al.[23] investigated various computational procedures intended to reduce the time required to simulate the average aerodynamic loads on stores in the unsteady flow of a weapons bay. Their reduced-domain approach uses a time-averaged flowfield to provide boundary conditions for a simulation using only a subdomain of the original flow. They obtained good agreement between loads computed from time-accurate simulations and loads obtained using the reduced-domain methodology. Such efforts have only begun to define the issues associated with the computational simulation of unsteady flows. Much work remains to be done to identify the issues further and establish validation criteria for unsteady flow. A careful examination of the associated computational procedures also is needed before CFD can be used confidently to predict unsteady-flow properties and dynamic distortion in particular.

Finally, because CFD simulations are expensive in terms of computer resources and schedule, they must be used carefully to obtain the maximum benefit. A systematic study is needed to identify those parameters to which the performance model is most sensitive. Once these parameters are identified, the validation efforts can be concentrated on the most important issues.

Summary

We have reviewed the engine–airframe integration methodology and identified areas in which the traditional experimentally based process is deficient. We identified ways in which CFD can be used to supplement the existing methodology. We concentrated on validation issues associated with the computational procedures. Wall interference, support interference, and Reynolds-number corrections are well developed. However, a systematic approach toward validation is required. We discuss inlet testing in detail and suggest several ways in which computational simulation could augment and complement wind-tunnel testing. Specifically, CFD can be used to provide increased accuracy for inlet drag increments. A similar opportunity exists for inlet total pressure maps, but a major validation effort is required. This effort must address basic issues of approximating time-averaged measurements of unsteady flow with computations based on steady-state assumptions. Finally, we note that a systematic study of the process of performance-model construction should be made to identify the most critical areas so that validation efforts then can be focused on them.

Acknowledgments

We acknowledge M. Hinkleman, Boeing Defense and Space Group, Seattle, Washington, for a 1995 private communication and R. H. Nichols for a private communication regarding Ref. 20.

References

[1]Covert, E. (ed.), *Thrust and Drag: Its Prediction and Verification,* Vol. 98, Progress in Astronautics and Aeronautics, AIAA, New York, 1985.

[2]"Air Intakes for High Speed Vehicles," Fluid Dynamics Panel Working Group 13, AGARD AR 270, Sept. 1991.

[3]"Experimental Data Base for Computer Program Assessment," Fluid Dynamics Panel Working Group 04, AGARD AR-138, May 1979.

[4]"Applications of Computational Fluid Dynamics in Aeronautics," AGARD CP-412, Aix-en-Provence, France, April 1986.

[5]"Validation of Computational Fluid Dynamics," AGARD CP-437, Lisbon, Portugal, May 1988.

[6]Marvin, J. G., "Dryden Lectureship in Research, a Perspective on CFD Validation," AIAA Paper 93-0002, Jan. 1993.

[7]Martellucci, A., "The Challenging Process of Validating CFD Codes," AIAA Paper 90-1402, June 1990.

[8]Oberkampf, W. L., Blottner, F. G., and Aeschliman, D. P., "Methodology for Computational Fluid Dynamics Code Verification and Validation," AIAA Paper 95-2226, June 1995.

[9]Uenishi, K., Pearson, M. S., Lehnig, T. R., and Leon, R. M., "CFD Based 3D Turbofan Nacelle Design Tool," AIAA Paper 90-3081, Aug. 1990.

[10]Lasinski, T., Buning, P., Choi, D., Rogers, D., Bancroft, G., and Merrit, F., "Flow Visualization of CFD Using Graphics Work Stations," AIAA Paper 87-1180, June 1987.

[11]Risk, Y. M., and Gee, K., "Numerical Prediction of the Unsteady Flowfield Around the F-18 Aircraft at Large Incidence," AIAA Paper 91-0020, Jan. 1991.

[12]Degani, D., and Schiff, L. B., "Numerical Simulations of the Effect of Spatial Disturbances on Vortex Asymmetry," *AIAA Journal*, Vol. 29, No. 3, 1991, pp. 344–352.

[13]Kordulla, W., Vollmers, H., and Dallmann, U., "Simulation of Three-Dimensional Transonic Flow with Separation Past a Hemisphere-Cylinder Configuration," AGARD CP-412, Aix-en-Provence, France, April 1986 (Paper 31).

[14]Sickles, W. L., and Erickson, J. C., Jr., "Wall Interference Corrections for Three-Dimensional Flows," AIAA Paper 90-1408, June 1990.

[15]Erickson, J. C., Jr., "Adaptive Wind Tunnel Walls—Compendium of Final Report—AGARD FDP Working Group 12," AIAA Paper 90-1405, June 1990.

[16]Martin, F. W., Jr., Sickles, W. L., and Stanley, S. A., "Transonic Wind Tunnel Wall Interference Analysis for the Space Shuttle Vehicle," AIAA Paper 93-0420, Jan. 1993.

[17]Willhite, P., Jasper, D., and Romer, W., "A Critical Evaluation of CFD Predictions of Full Aircraft Drag Increments," AIAA Paper 95-2289, June 1995.

[18]Smith, C. F., and Podleski, S. D., "Installed F/A-18A Inlet Flow Calculations: A Grid Study," *Journal of Propulsion and Power*, Vol. 11, No. 6, 1995, pp. 1250–1256.

[19]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solution of the Euler Equations by Finite Volume Methods Using a Runge–Kutta Time Stepping Scheme," AIAA Paper 81-1269, June 1981.

[20]Fluid Dynamics Panel Working Group 13, "Test Case 3—Subsonic and Transonic Circular Intake," AGARD AR-270, Sept. 1991, Chap. 3.3.3.

[21]Suhs, N. E., and Jordan, J. K., "Three-Dimensional Cavity Flow Computations at Transonic Mach Numbers," Arnold Engineering Development Center, TR-88-30, Arnold AFB, TN, Feb. 1989.

[22]Aboulmouna, M. E., Jechura, M. C., Powell, E. S., Rist, F. L., Shope, F. L., Thoms, R. D., and Thomson, W. G., "Computational Fluid Dynamics Support to the B-1B Conventional Weapons Test Program at AEDC," Arnold Engineering Development Center, TR-95-25, Arnold AFB, TN, Dec. 1995.

[23]Aboulmouna, M. E., Bangasser, C. T., Denny, A. G., Hand, T. L., Jechura, M. C., Ratriff, Suhs, N. E., Thoms, R. D., and Thomson, W. G., "Status Report on Weapons Bay Flow Computations for Store Separation," Arnold Engineering Development Center, TR-95-31, Arnold AFB, TN, Jan. 1996.

P. R. Bandyopadhyay
Associate Editor