Readers' Forum

Comment on "Numerical Simulation of Cold Flow in an Axisymmetric Centerbody Combustor"

M.S. Raju,* M.J. Creed,† and L. Krishnamurthy‡
University of Dayton Research Institute
Dayton, Ohio

THE authors' main conclusions[1] would seem to be that their preliminary results have successfully established a numerical simulation of the unsteady flow in the near-wake recirculating region of a ducted bluff-body combustor; that the computational procedure can easily address the inclusion of a central fuel jet through the bluff body; and all that remains is the addition of chemistry to the computer program and the investigation of the unsteady combustion process. Neither the results of Ref. 1 nor the subsequent investigations reported by Krishnamurthy et al.[2] seem to justify the above conclusions. A comprehensive examination of the inflow and outflow boundary conditions to model the flowfield described in Ref. 2 with the same computer code of Ref. 1 appears to indicate that the authors' demonstration of the feasibility of modeling the self-excited oscillation due to the vortex shedding is far from conclusive. While the details are available in Ref. 2, herein we will point out some of the apparent deficiencies in Ref. 1.

1) In view of the recent experimental evidence[3] that strongly suggests the presence of three-dimensional effects in the interior of the near wake, the axisymmetric formulation may be inadequate to correctly simulate the flowfield.

2) The authors' estimate of a vortex shedding frequency of 70 Hz (corresponding to their anticipated Strouhal number range of 0.18-0.22) is based on a value of $u_\infty \sim 50$ m/s. The characteristic time L/u_∞ for the value of L of their computational domain will then be 9.1×10^{-3} s and not their quoted value of 5.2×10^{-3} s. It turns out that their quoted value is very close to the characteristic time obtained for a value of $L = 0.8481$ ft, which is used as a reference length in their code. Somewhat surprisingly, this length is precisely equal to the distance of 25.85 cm between the screen and the centerbody face in their Fig. 1. What is the relevance of this length and the characteristic time obtained therefrom to their analysis?

3) The computations in Refs. 1 and 2 cannot be representative of the so-called large-eddy simulation (LES). Correct formulation of LES requires a three-dimensional simulation and a subgrid-scale modeling of turbulence. These features are not implemented in Refs. 1 and 2, and the calculations are laminar-like, except for the explicitly introduced numerical damping and the inherently present numerical diffusion. The nature and extent to which these artificial viscous effects mimic turbulence, however, are unknown at best. While it is instructive to regard these time-dependent computations with the full Navier-Stokes equations employing the laminar viscosity as a special case of the Reynolds-averaged formulation with zero turbulence, its relevance to the high-Reynolds-number flows of current interest is questionable.

4) The value of 0.2 obtained for the Strouhal number (S_t) in Eq. (18) is based on a value of 50 m/s for u (corresponding to a combustor mass flow of 2 kg/s). We note that Fig. 4 indicates that the average mass flow is of the order of 4 kg/s, which corresponds to a mean annulus inflow velocity u of approximately 100 m/s. The Strouhal number for 70 Hz then is on the order of 0.1. While the authors' reliance on a frequency of 70 Hz and a mass-flow range of 3-5 kg/s receives further confirmation at the end of the paragraph following Eq. (18), the predictive success in terms of the Strouhal number becomes less persuasive. We add, parenthetically, that the experiments of Ko and Chan[4] report $0.25 \leq S_t \leq 0.26$ inside the recirculation region and $0.30 \leq S_t \leq 0.32$ inside the inner mixing region for the unconfined flowfield past an axisymmetric cylindrical bluff body (which is very similar to the present centerbody).

5) We believe that the probable explanation of the shedding-like behavior noted in Ref. 1 is along the following lines. Figures 1 and 2 correspond to the calculations subsequently[2] carried out with an identical set of inflow and outflow boundary conditions as in Ref. 1. As evidenced in Fig. 1a, the flowfield does exhibit shedding-like behavior in the earlier stages of the numerical computations. However, if the computations are carried out for a long enough time, the initial transients are wiped out before the flowfield establishes a single recirculation region (Figs. 1b and 1c). It is also observed that after the recirculation region has been established, it extends in the axial direction at a rate of 3 m/s. If the calculations are carried out further, this process continues until the reverse flow reaches the exit boundary, thereby rendering the exit-boundary conditions incompatible. Calculations performed with a wide range of different inflow and outflow boundary conditions suggest that the improper specification of outflow boundary conditions to handle reverse flow causes the flowfield to become nonphysical, breaks up the recirculation region, and exhibits the shedding-like phenomenon as longitudinal anharmonic oscillations in the mass-flow rate begin to appear with a dominant frequency $\omega_1 = c/4L$ cor-

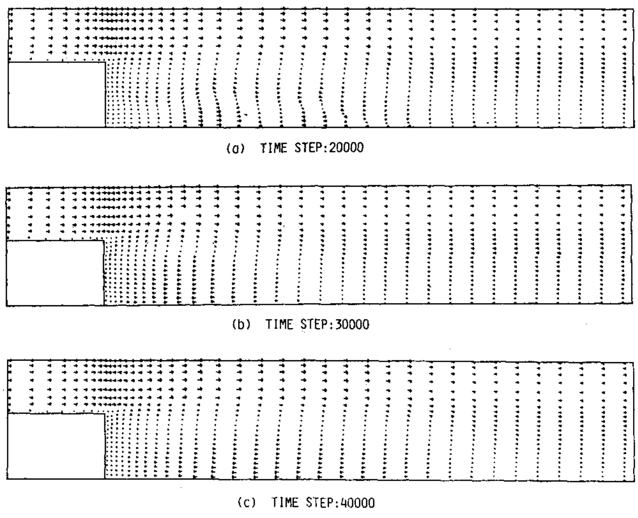

Fig. 1 Computed velocity-vector plots at different time steps in a centerbody combustor configuration.

Received July 6, 1985.

*Associate Research Engineer, Fluid Mechanics Group.

†Undergraduate, Chemical Engineering, Fluid Mechanics Group.

‡Senior Research Engineer, Fluid Mechanics Group. Associate Fellow AIAA.

responding to a quarter-wave resonator. Our results[2] indicate that these oscillations eventually attenuate before the flowfield re-establishes a single recirculation zone. The whole process repeats again with the axial migration of recirculation zone as the calculations proceed further. The foregoing appears to indicate that the shedding-like behavior observed[1] is neither self-sustaining nor physical, but strictly a numerical artifact. Indeed, continuation of the earlier computation[1] without the numerical damping was found to fail.[2]

Another undesirable feature of the numerical computations[1] (noted subsequently in Ref. 2) is the large temperature fluctuations. These were found to range from 4000 to 250°R and to be restricted to a narrow region near the centerline toward the reattachment point of the recirculation zone. A closer examination in Ref. 2 revealed that these large temperature fluctuations resulted from a discrepancy in Ref. 1 in the evaluation of the time step based on the CFL criterion. By correcting this error, the fluctuations were reduced to the range of 1500-300°R. Nevertheless, these temperature fluctuations are indications of nonphysical behavior of the solution as computations are performed for nonreacting and low subsonic flows. Furthermore, it becomes extremely difficult for the system to attain the desired mass-flow rate, since the boundary conditions of Ref. 1 do not have any control over the needed pressure drop through the combustor.

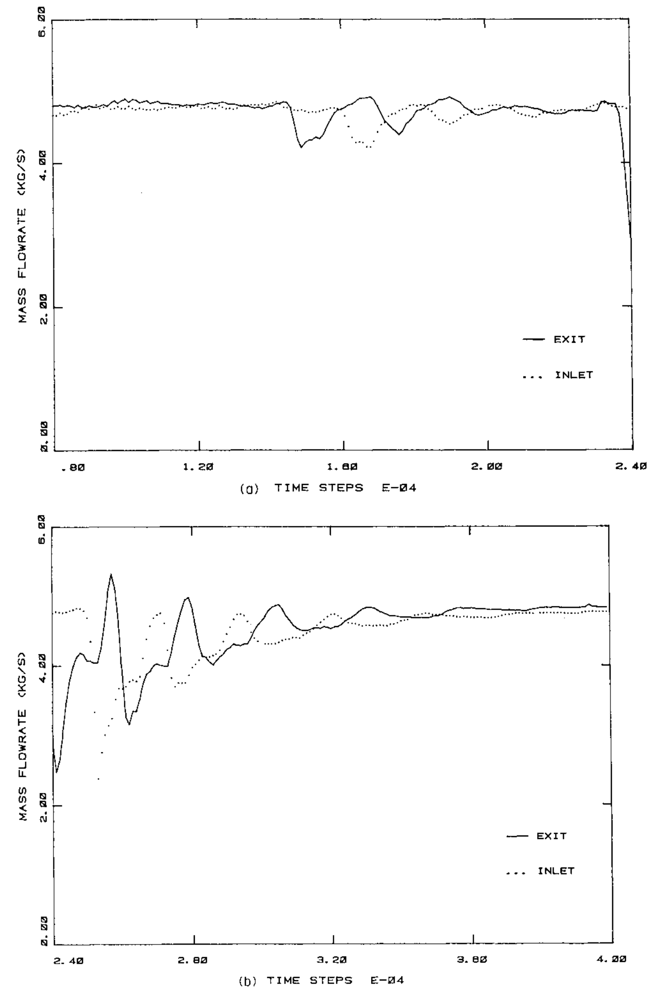

Fig. 2 Temporal variation of combustor mass flow.

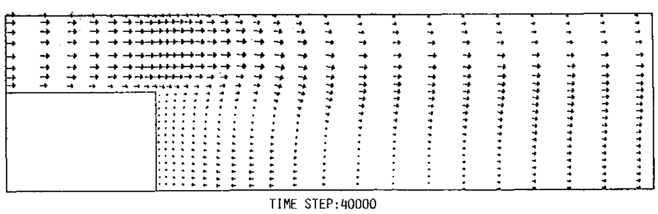

Fig. 3 Computed steady-state velocity-vector field in a centerbody combustor configuration.

6) In our recent calculations involving the time-dependent, axisymmetric, compressible Navier-Stokes equations, solutions have been obtained for the subsonic flowfield in the identical configuration with the identical computational grid of Ref. 1. Unlike the calculations reported in Refs. 1 and 2, turbulence is described by a two-parameter (k-ϵ) model of Jones and Launder.[5-7] The equations of motion are advanced in time by employing MacCormack's explicit unsplit algorithm. The computer code of Shang,[8] which was used in Refs. 1 and 2, is further modified to solve the two additional equations for k and ϵ. The equations for k and ϵ are solved in the same spirit of MacCormack's algorithm involving a forward-predictor, backward-corrector sequence at each time step. For a discussion of the boundary conditions and other details, the reader is referred to Ref. 9.

The calculations[9] indicate that the flowfield reaches an asymptotically steady-state solution after 40,000 time steps. Figure 3 shows the velocity-vector plot corresponding to this steady-state solution. Furthermore, the characteristics of the recirculation zone and the values of the flow variables obtained from these calculations correspond very closely to the experimentally observed vlaues.[9] Recent calculations[10] for the dump-combustor configuration involving the time-dependent equations also indicate that the flowfield reaches a steady state. These calculations further demonstrate that the time-dependent computations lacking a turbulence model to properly account for the dissipation inherently present at the high-Reynolds-number flows will lead to physically unrealistic solutions at best.

7) Both experimental and theoretical evidence on the bluff-body flame stabilizers suggest that during combustion the flow structure undergoes significant changes as compared to an isothermal stream.[3,11,12] Heat liberation is found to reduce the large-scale vortex shedding as well as the energy available to turbulent velocity fluctuations, because the dilatation due to heat release competes with or dominates the turbulence energy production by shear stresses in the vicinity of the recirculation zone. For Reynolds numbers on the order of 10^4, the flowfield behind the bluff body is observed to become stationary due to the introduction of combustion, in contrast to the flowfield characterized by large-scale unsteady motion observed in nonreacting flows. On the other hand, combustion in confined systems can and does lead to acoustic oscillations. Therefore, we believe it is a gross misrepresentation to compare reactive and nonreactive flowfields, as is done in Ref. 1.

References

[1]Scott, J.N. and Hankey, W.L. Jr.,"Numerical Simulation of Cold Flow in an Axisymmetric Centerbody Combustor," *AIAA Journal,* Vol. 23, May 1985, pp. 641-649.

[2]Krishnamurthy, L., Raju, M.S., Creed, M.J., and Memering, J.N., "Time-Averaged and Time-Dependent Computations of Isothermal Flowfields in a Centerbody Combustor," AFWAL-TR-84-2081, Dec. 1984.

[3]Roquemore, W.M., Tankin, R.S., Chiu, H.H., and Lottes, S.A., "The Role of Vortex-Shedding in a Bluff-Body Combustor," ASME Winter Annual Meeting, New Orleans, LA, Dec. 1984.

[4]Ko, N.W.M. and Chan, W.T., "The Inner Regions of Annular Jets," *Journal of Fluid Mechanics,* Vol. 93, 1979, pp. 549-584.

[5]Jones, W.P. and Launder, B.E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," *International Journal of Heat and Mass Transfer,* Vol. 15, 1972, p. 301.

[6]Jones, W.P. and Launder, B.E., "The Calculation of Low-Reynolds-Number Phenomena with a Two-Equation Model of Turbulence," *International Journal of Heat and Mass Transfer,* Vol. 16, 1973, p. 1119.

[7]Hanjalic, K. and Launder, B.E., "Contribution Towards a Reynolds-Stress Closure for Low-Reynolds-Number Turbulence,"

Journal of Fluid Mechanics, Vol. 74, 1976, p. 593.

[8]Shang, J.S., "Oscillatory Compressible Flow Around a Cylinder," AIAA Paper 82-1011, June 1982.

[9]Raju, M.S. and Krishnamurthy, L., "Computational Fluid Dynamic Studies of Certain Complex Turbulent Flows Relevant to Turbojet Combustors. Part I: Time-Dependent Calculations with the k-ϵ Turbulence Model for an Existing Centerbody Combustor," University of Dayton Research Insititute, Dayton, OH, UDR-TR-85-82, Oct. 1985.

[10]Drummond, J.P., "Numerical Study of a Ramjet Dump Combustor Flowfield," *AIAA Journal,* Vol. 23, April 1985, pp. 604-611.

[11]Fujii, S. and Eguchi, K., "A Comparison of Cold and Reacting Flows Around a Bluff-Body Flame Stabilizer," *Journal of Fluids Engineering,* Vol. 103, 1981, pp. 328-334.

[12]Lyashenko, V.P. and Yagodkin, V.I., "Analysis of Reacting Gas Flow Behind a Flame Stabilizer in a Plane Channel," *Combustion, Explosion and Shock Waves,* Vol. 19, 1984, pp. 583-587.

Reply by Authors to M.S. Raju, M.J. Creed, and L. Krishnamurthy

J.N. Scott*
University of Dayton Research Institute
Dayton, Ohio
and
W.L. Hankey Jr.†
Air Force Wright Aeronautical Laboratory
Wright-Patterson Air Force Base, Ohio

We appreciate the commentors' interest in our work and their diligence in examining the critical issues they raised. As they have noted, it is indeed not a trivial task to achieve a numerical simulation of self-sustained oscillations in fluid flow. However, it is apparent that the commentors misunderstood the conclusions of the work presented in Ref. 1. Evidently, they overlooked our statements which emphasized that the reported work was the first-phase solution of a difficult problem and that additional work is needed to simulate such complex flows accurately. In order to avoid any further confusion or misinterpretation we would like to reiterate the statements made in the conclusions of our original paper. "As the first phase of a numerical computation of flow in a combustor, this paper demonstrates the feasibility of modeling the self-excited oscillation involving vortex shedding in axisymmetric flow. It should be emphasized that the computed results are for cold flow only and that certain questions remain unanswered concerning the formulation of the boundary conditions. Although the numerical results are preliminary in nature, they provide insight into the mixing process and can help guide further research in the combustion field." Furthermore, as stated elsewhere in the text of the paper, "caution must be exercised in examining the numerical vorticity data because certain features may not truly represent the physical combustion flow"; and "it must be emphasized that the present numerical simulation does not completely model the combustor experiment since the computational configuration does not include the flow from the central jet, combustion, nor identical air flow rate." In view of these statements, it is most perplexing how the commentors determined that the cold-flow model presented in the paper was intended to represent the reactive flowfield of an operating combustor, and thereby state that the comparison made in the paper was a "gross misrepresentation."

The issues of boundary conditions, turbulence modeling, and extension to three dimensions are areas that have long since been identified as pacing items in advancing the computational capability for complicated flow situations.[2-4] Current computer storage and speed limitations impose restrictions that do not permit realistic three-dimensional computations. However, the computer capability to compute three-dimensional flows will be available with the introduction of the CRAY 2 computer, which will be operational sometime in late 1985.

In their investigation,[5] the commentors employed the k and ϵ turbulence model of Jones and Launder in their attempt to simulate an unsteady flow. It has been reported in the literature[6-8] that the use of a steady-state turbulence model accounts for part of the Reynolds stresses twice in a time-dependent calculation and destroys the basic physical process. The currently accepted procedure is to include only a portion of the turbulence model (only for the high frequencies) and compute the self-sustained oscillation by first principles (see Ref. 9).

A major goal in the advancement of computational fluid dynamics is the development of an unsteady turbulence model.[10] Since this does not exist, it is recommended that one first attempt to compute a known self-sustained oscillation without a turbulence model using the necessary time step and grid size, which will capture the axial waves and represent the transverse eigenfunction adequately. These values may be deduced from linear stability theory.[11-13] After successfully demonstrating the existence of a self-sustained oscillation, a small portion of the turbulence spectrum may be added; recalling, however, that numerical dissipation is already present. Existing circumstantial evidence shows that this approach can be made successful and indicates that the turbulent dissipation apparently is not an essential ingredient in a violent self-sustained oscillation.[14-20]

In order to improve the present state of the art in the area, the following investigations are required in order to advance the capability to numerically simulate self-sustained oscillations.

1) Conduct controlled experiments of oscillating flow in short test sections isolated by choke plates or with well-defined end conditions so that the entire length may be simulated efficiently.

2) Obtain experimental measurements of Reynolds stresses over a wide frequency range to permit the development of an "unsteady" turbulence model.

3) Perform a series of numerical simulations of these experiments and investigate the influence of boundary conditions and numerical resolution.

We are in the process of conducting investigations along these lines and hope that in this manner we may further add to the impressive work in self-sustained oscillations that already exists. We would encourage the commentors to continue their efforts in this area with the hope that, eventually, they too may realize the success others have achieved in simulating complex unsteady flow phenomena.[1,7,15-22]

Received Sept. 16, 1985.

*Research Engineer, Aerospace Mechanics Division. Member AIAA.

†Senior Scientist. Associate Fellow AIAA.

References

[1]Scott, J.N. and Hankey, W.L., "Numerical Simulation of Cold Flow in an Axisymmetric Centerbody Combustor," *AIAA Journal,* Vol. 23, May 1985, pp. 641-649.

[2]Chapman, D.R., "Computational Aerodynamics Development and Outlook," *AIAA Journal,* Vol. 17, 1979, pp. 1293-1313.

[3]Kutler, P., "A Perspective of Theoretical and Applied Computational Fluid Dynamics," AIAA Paper 83-0037, Jan. 1983.

[4]Scott, J.N., "Numerical Solution of the Navier-Stokes Equations for 3-D Internal Flow—An Emerging Capability," *3-D Computation Techniques Applied to Internal Flows in Propulsion Systems,* AGARD Lecture Series No. 140, June 1985.