

Techniques for Improved Blade Vortex Interaction Modeling," *Proceedings of the 51st Annual Forum of the American Helicopter Society* (Fort Worth, TX), 1995.

<sup>31</sup>Johnson, W., *Helicopter Theory*, Princeton Univ. Press, Princeton, NJ, 1980, Chap. 1.

<sup>32</sup>Miller, W., "Analytical/Numerical Matching and Periodic Inversion: Two Advances in Free Wake Analysis," Ph.D. Dissertation, Dept. of Mechanical Engineering, Duke Univ., Durham, NC, May 1990.

<sup>33</sup>Bliss, D. B., and Miller, W. O., "Efficient Free Wake Calculations Using Analytical/Numerical Matching," *Journal of the American Helicopter Society*, Vol. 38, No. 2, 1993, pp. 878, 879.

S. Glegg  
Associate Editor

## Two-Way Coupled Gas-Particle Systems in an Axisymmetric Ramjet Combustor

E. J. Chang\* and K. Kailasanath†  
U.S. Naval Research Laboratory,  
Washington, D.C. 20375

and  
S. K. Aggarwal‡  
University of Illinois at Chicago,  
Chicago, Illinois 60680

### Introduction

FOR a variety of reasons, in many multiphase flow applications, tracking the dispersed phase in a Lagrangian manner may be desirable. Here individual particle trajectories are calculated as they are carried by the gas phase in the system.<sup>1-4</sup> However, because of the high numerical cost of tracking large numbers of particles, these studies have necessarily been kept at relatively low seeding and mass fraction levels. To reach higher seeding levels, using virtual particle methods where each computational, or tracked, particle actually represents a group of particles of the same size has been necessary.<sup>5</sup> Calculation of energy and momentum transfer between phases is then based on the entire group instead of just the carrier particle. Using such a method allows a high mass loading to be obtained without the necessary computational expense of tracking every particle in the system. However, the accuracy of the results obtained using this technique is a matter of concern.<sup>6</sup>

This Note extends our previous efforts on the simulation of particle dynamics in the confined geometry of an idealized ramjet combustion system.<sup>3,4</sup> In that work we neglected the effect of the particles on the fluid flow. We now take up the issue of return momentum and energy effects from the particles to the gas phase of the flow. The inclusion of such return effects is critical to the development of reliable predictive tools for transient, multiphase flows in propulsion systems such as ramjets. In a situation such as that considered here, with particles injected as a narrow stream, the use of virtual particles may be called into question. Artificially high mass loadings may arise within a computational cell as the true distribution of particles in space is neglected and replaced by a carrier particle. Thus, return effects that may be distributed over several computational cells as

particles are dispersed in the flow may, in fact, be represented as a single point force when using the virtual particle method. Results are presented from simulations with and without virtual particles at the same mass loading levels in an effort to determine how accurately the use of virtual particles represents the coupled gas-particle system.

### Numerical Model

The flow into a central dump ramjet combustor is computed by solving the compressible, time-dependent, conservation equations for mass, momentum, and energy in an axisymmetric geometry. These equations are given as

$$\frac{\partial \rho}{\partial t} = -\nabla \cdot (\rho \mathbf{U}) \quad (1)$$

$$\frac{\partial (\rho \mathbf{U})}{\partial t} = -\nabla \cdot (\rho \mathbf{U} \mathbf{U}) - \nabla p + S_v \quad (2)$$

$$\frac{\partial E}{\partial t} = -\nabla \cdot (E \mathbf{U}) - \nabla \cdot p \mathbf{U} + S_E \quad (3)$$

where  $E = e + \frac{1}{2} \rho \mathbf{U}^2$  and  $e = p/(\gamma - 1)$ ;  $\rho$ ,  $p$ ,  $e$ ,  $\mathbf{U}$ , and  $\gamma$  are the density, pressure, internal energy, velocity, and specific heat ratio, respectively.  $S_v$  and  $S_E$  are momentum and energy source terms from the particle, or dispersed, phase, which accounts for mass loading effects. To solve these equations, the flux-corrected transport (FCT) algorithm,<sup>7</sup> a conservative, monotonic algorithm with fourth-order phase accuracy, is employed. Further details on the numerical approach used can be found in Ref. 3.

The geometric configuration used is shown in Fig. 1 and is representative of a generic ramjet propulsion system. A cylindrical jet with a prescribed mean velocity of 100 m/s flows through an inlet of diameter  $D = 6.35$  cm into a cylindrical combustion chamber of larger diameter  $2D$ . An annular exit nozzle at the end of the chamber is modeled to produce choked flow and forces the flow to become sonic at the exit throat. The mass inflow rate is 0.78 kg/s, and the initial chamber pressure is 188 kPa. A fixed  $60 \times 120$  computational grid is used with fine zones near the entrance to the combustor in both the radial and axial directions. The cell sizes gradually increase away from the dump plane. The grid resolution is the same as that used in our previous studies, where it was found to be sufficient to produce accurate results.<sup>3</sup> The time step used is  $\Delta t = 3.76 \times 10^{-7}$  s. For these flow conditions the characteristic shedding frequency of vortex structures at the combustor step is given by  $\sigma_s = 1380$  Hz. A vortex merging frequency of 690 Hz and an inlet acoustic mode of 145 Hz are additional characteristic frequencies observed in the system.

A Lagrangian approach is used to track each particle. Under the assumption that the density of the particle  $\rho_p$  is much larger than the density of the surrounding gas phase, the equations of motion for a particle of diameter  $d_p$  in the absence of gravity reduce to

$$\frac{d\mathbf{V}(t)}{dt} = \frac{[\mathbf{U}(\mathbf{Y}, t) - \mathbf{V}(t)]f(Re_p)}{\tau_p} \quad (4)$$

$$\frac{d\mathbf{Y}(t)}{dt} = \mathbf{V}(t) \quad (5)$$

where  $\mathbf{Y}$  is the particle position,  $\tau_p = \rho_p d_p^2 / 18 \mu_{\text{gas}}$  is the particle response time, and  $\mu_{\text{gas}}$  is the dynamic viscosity of the gas phase. The coefficient  $f$  is a scalar function of the particle Reynolds number  $Re_p$ , with  $f(Re_p) = 1.0 + Re_p^{0.687}$ . A fourth-order, predictor-corrector method is used to obtain particle velocity and position in time, and a sixth-order Lagrangian interpolation scheme is used to interpolate flow properties from the computational grid to the particle positions. Particle-particle interactions are neglected.

To take into account return effects from the dispersed particles to the gas phase, the particle momentum and energy are calculated for each particle location, and a linear weighting scheme based on cell volume is used to find the corresponding source terms used in Eqs. (2) and (3) for the gas-phase calculation at the surrounding

Presented as Paper 95-2561 at the AIAA/ASME/SAE/ASEE 31st Joint Propulsion Conference, San Diego, CA, July 10-12, 1995; received March 16, 1998; revision received Nov. 6, 1998; accepted for publication Nov. 19, 1998. This paper is declared a work of the U.S. Government and is not subject to copyright protection in the United States.

\*Mathematician, Laboratory for Computational Physics and Fluid Dynamics, Code 6410, Member AIAA.

†Head, Center for Reactive Flow and Dynamical Systems, Laboratory for Computational Physics and Fluid Dynamics, Associate Fellow AIAA.

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

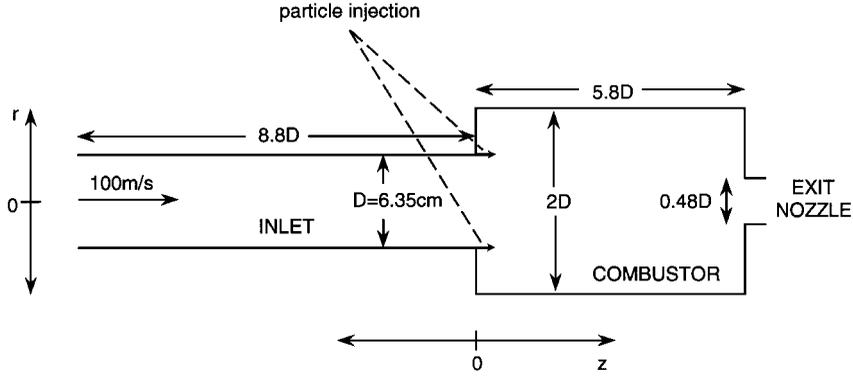


Fig. 1 Schematic of flow configuration. Particles are injected near the combustor step in the direction of the inflowing gas phase.

grid locations. For each particle in the system, these source terms are given as

$$S_v = -N_g N_\theta m_p \left[ \frac{dV}{dt} \right], \quad S_E = -N_g N_\theta m_p \left[ \frac{dV}{dt} \right] \cdot V \quad (6)$$

where  $m_p$  is the mass of a single particle. To account for volumetric effects, an azimuthal distribution must be specified, and an even distribution of  $N_\theta = 1024$  azimuthal particle injection locations is assumed.  $N_g$  represents the number of particles in a group consisting of 1 carrier particle and  $N_g - 1$  virtual particles. For the case with no virtual particles,  $N_g = 1$ .

## Results

The multiphase flow simulations are started after a fully developed unseeded flow has been established. Particles are injected into the ramjet combustor every  $10\Delta t$  near the corner of the combustor step with zero radial velocity and streamwise velocity  $v_{inj} = 50$  m/s. To obtain a mass loading (particle mass flux in/gas-phase mass flux in) of 10%, a particle diameter of  $d_p = 32.4 \mu\text{m}$  is used, corresponding to a Stokes number of  $S = \tau_p \cdot \sigma_s = 4.53$  with  $\rho_p = \rho_{\text{water}}$  and  $\mu_{\text{gas}} = \mu_{\text{air}}$ . Because reflective wall boundary conditions for the dispersed phase yield qualitatively similar results,<sup>3</sup> we present only results from simulations with absorptive wall boundary conditions.

Two different techniques are employed to achieve significant particle mass loadings. For the first method, which we shall refer to as the direct simulation method, we use  $N_g = 1$  but employ 16 distinct radial injection locations per azimuthal position. The first radial injection location is at the combustor step ( $r = 3.175$  cm,  $z = 0$ ), and each subsequent location is  $194.4 \mu\text{m}$  from the previous location, inward toward the combustor center, forming a stream of particles with a radial width of  $0.291$  cm. This spacing provides a 12-particle radius separation between injection locations and is on the same order in size as the smallest computational cell. We note that, although a Lagrangian approach for the dispersed phase is typically preferred when the resolution of the phenomena of interest is on a scale much smaller than the average droplet spacing,<sup>8</sup> it is deemed preferable in the present simulations as it provides a natural methodology for calculating particle dispersion in the presence of large-scale structures. Moreover, the spacing between droplet streams gets larger downstream because of the dispersion process.

The second technique involves the use of virtual particles. Here each particle in the simulation acts as a marker or carrier for a group of particles with a center of mass located at the simulation particle position. Each of the virtual particles has the same size and mass of the tracked particle, but its velocity and location are not explicitly calculated. The momentum and energy from the virtual particles are included in the coupling feedback source terms at the location of the associated carrier particle. In our simulations we use 15 virtual particles for each simulation particle ( $N_g = 16$ ) and inject either at the step corner  $r_{inj} = 3.175$  cm, at  $r_{inj} = 3.01$  cm, or at  $r_{inj} = 2.88$  cm. These positions are equivalent to the outermost, central, and innermost injection locations used to form the particle stream in the direct simulation method, respectively.

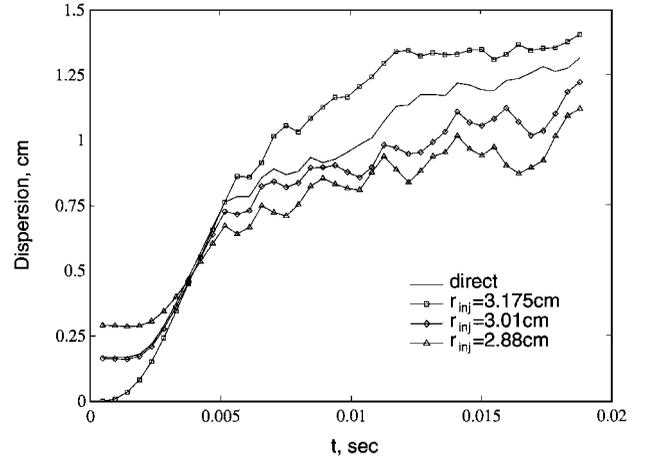
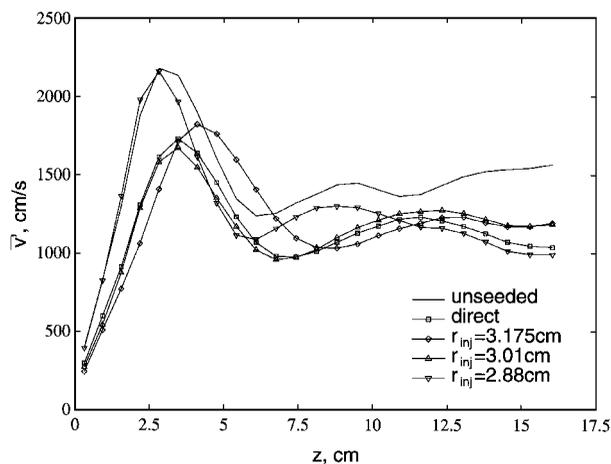


Fig. 2 Dispersion vs time using both direct and virtual particle simulation methods.

Figure 2 shows the time-dependent dispersion, defined here as the rms radial deviation of particle positions as measured from the height of the combustor step, for both the direct and virtual particle simulations. Dispersion is highest for the virtual particle simulation with  $r_{inj} = 3.175$  cm and overestimates that from the direct simulation. This overestimation is because of a higher particle ejection rate from the shear layer as dispersion is maximized when injection occurs at the step, corresponding to the core region of the shear layer where centrifugal effects are greatest.<sup>1,3</sup> For the virtual particle simulations with  $r_{inj} = 2.88$  and  $3.01$  cm, injection occurs outside the core region, and dispersion is significantly lower, underestimating that from the direct simulation.

When heavy particles are introduced into a flowfield, their inertia will act to attenuate or dampen fluctuations in the flow.<sup>9</sup> The rms velocity fluctuations in the shear layer (at  $r = 3.175$  cm) for all cases are shown in Fig. 3 as a function of axial location. As expected, in the direct simulation attenuation of fluctuations is observed at all downstream locations. The virtual particle simulation that best approximates this direct simulation result is that which uses the intermediate injection location at  $r_{inj} = 3.01$  cm. The other two virtual particle simulations provide mixed results. When the innermost location ( $r_{inj} = 2.88$  cm) is used, particles are injected outside of the shear layer, and little attenuation is observed near the step with the velocity fluctuations nearly the same as in the unseeded case. Farther downstream as particles begin to interact with the expanding shear layer, attenuation at a level similar to that of the other cases is achieved. Conversely, in the  $r_{inj} = 3.175$  cm case, particles are injected directly into the core region of the shear layer. Streamwise particle momentum and inertia act to stabilize the shear layer near the combustor step, delaying vortex development and shifting the location of peak fluctuations downstream. Also, note that an increase in the peak value of the fluctuations as the higher dispersion rate, and consequently lower particle concentrations, leads to lower attenuation levels in the shear layer downstream of the injection location.



**Fig. 3 Time-averaged rms fluctuating velocity in the shear layer as a function of downstream distance  $z$  for the unseeded flow, the direct simulation, and virtual particle simulations.**

Finally, a Fourier analysis of velocity fluctuations in the shear layer indicates that in both the  $r_{inj} = 3.01$  and  $2.88$  cm virtual particle simulations a downward shift in the vortex shedding frequency is observed with  $\sigma_s = 1355$  and  $1350$  Hz, respectively. This shift is due to a thickening of the shear layer caused by the presence of a high concentration of particles injected at a lower velocity than the carrier fluid in the high-speed side of the shear layer. No such frequency shift was observed in either the direct simulation where particles are distributed uniformly across the shear layer or the virtual simulation with  $r_{inj} = 3.175$  cm where the injection velocity is approximately the same as the gas phase in the core region of the shear layer.

### Conclusions

Our investigation of a two-way coupled gas-particle system indicates that the shear-layer dynamics in a ramjet combustor can be modulated by controlling the dispersed-phase injection parameters, especially the particle injection location, size, velocity, and mass loading ratio. Both direct and virtual particle simulation techniques were employed to simulate the stream of particles injected into the shear-layer region. The effect of particle mass loading on the flow is seen as an attenuation of velocity fluctuations in the shear layer. In the direct simulations this attenuation is observed at all locations downstream of particle injection. When modeling the particle stream using the virtual particle method, attenuation is found to depend on injection location. The injection location corresponding to the central injection location of the particle stream, at  $r_{inj} = 3.01$  cm, provides the best approximation to attenuation levels achieved in the direct case. When injecting at this location, though, dispersion was slightly underestimated because of reduced centrifugal effects. In addition, the choice of injection location was seen to alter slightly the characteristic vortex shedding frequency as the shear-layer thickness may be increased effectively by the presence of the particles. When taken in context, though, these results indicate that, when compared to the direct simulations, the virtual particle method can provide acceptable results and suggest that, with certain caveats, this technique can be used with confidence in situations where it is not practical to track every particle in the flow.

### Acknowledgments

This work has been supported by the Office of Naval Research through the Mechanics and Energy Conversion Division and the U.S. Naval Research Laboratory. A grant of High Performance Computing (HPC) time from the Department of Defense HPC Shared Resource Center, U.S. Army Corps of Engineers Waterways Experiment Station is also gratefully acknowledged.

### References

<sup>1</sup>Wang, L. P., and Maxey, M. R., "Settling Velocity and Concentration Distribution of Heavy Particles in Homogeneous Isotropic Turbulence," *Journal of Fluid Mechanics*, Vol. 256, 1993, pp. 27–68.

<sup>2</sup>Uthuppan, J., Aggarwal, S. K., Grinstein, F. F., and Kailasanath, K., "Particle Dispersion in a Transitional Axisymmetric Jet: A Numerical Simulation," *AIAA Journal*, Vol. 32, No. 10, 1994, pp. 2040–2048.

<sup>3</sup>Chang, E. J., and Kailasanath, K., "Simulations of Particle Dynamics in a Confined Shear Flow," *AIAA Journal*, Vol. 34, No. 6, 1996, pp. 1160–1166.

<sup>4</sup>Chang, E. J., and Kailasanath, K., "Behavior of Heavy Particles in an Acoustically Forced Confined Shear Flow," *AIAA Journal*, Vol. 34, No. 11, 1996, pp. 2429–2431.

<sup>5</sup>Raju, M. S., and Sirignano, W. A., "Spray Computations in a Centerbody Combustor," *Journal of Propulsion and Power*, Vol. 6, No. 2, 1990, pp. 97–105.

<sup>6</sup>Elghobashi, S. E., "On Predicting Particle-Laden Turbulent Flows," *Applied Scientific Research*, Vol. 52, No. 4, 1994, pp. 309–329.

<sup>7</sup>Boris, J. P., and Book, D. L., "Flux-Corrected Transport I: SHASTA—A Fluid Transport Algorithm That Works," *Journal of Computational Physics*, Vol. 11, 1973, pp. 38–69.

<sup>8</sup>Aggarwal, S. K., Fix, G., Lee, D. N., and Sirignano, W. A., "Numerical Optimization Studies of Axisymmetric Unsteady Sprays," *Journal of Computational Physics*, Vol. 50, No. 1, 1983, pp. 101–115.

<sup>9</sup>Squires, K. D., and Eaton, J. K., "Particle Response and Turbulence Modification in Isotropic Turbulence," *Physics of Fluids A*, Vol. 2, No. 7, 1990, pp. 1191–1203.

G. M. Faeth  
Editor-in-Chief

## Nonsingular Eigenvectors of the Flux Jacobian Matrix for Reactive Flow Problems

Mark A. Busby\*

U.S. Naval Surface Warfare Center,  
Bethesda, Maryland 20817

and

Pasquale Cinnella†

Mississippi State University,  
Mississippi State, Mississippi 39762

### I. Introduction

THE development of numerical tools for the simulation of finite-rate chemistry flows has received increasing attention over the past decade. Reacting flows play an important role in a wide range of scientific fields, such as combustion, hypersonic flight, and explosion analysis. The purpose of this study is to describe a step that can be taken to develop accurate and efficient solutions for such flows: The eigensystem of the Euler equations for a flow in chemical nonequilibrium is developed, including a well-designed set of eigenvectors for higher-order upwind applications, which do not contain potentially ill-defined terms. This work results in an extension of the methods developed by Whitfield and Janus<sup>1</sup> for a perfect gas and their later extensions.<sup>2</sup>

Several researchers have developed upwind, implicit, finite-rate chemistry algorithms for very general flow situations. References 3–5 are representative of the currently available technology. A common thread among these solvers is the development and use of eigenvalues and eigenvectors of the inviscid fluxes, which are necessary for the design of the numerical schemes. The choice of eigenvectors is somewhat arbitrary because of the presence of repeated eigenvalues. The eigensystem proposed by Cinnella,<sup>6</sup> although ideally suited when used with higher-order spatially accurate MUSCL schemes, can prove inadequate when used in conjunction with flux-extrapolation schemes. (The MUSCL approach achieves high-order

Received July 25, 1998; revision received Nov. 16, 1998; accepted for publication Nov. 25, 1998. Copyright © 1999 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

\*Mathematician, Carderock Division. Member AIAA.

†Associate Professor, Department of Aerospace Engineering. Senior Member AIAA.