JOURNAL OF AIRCRAFT
Vol. 42, No. 1, January–February 2005

Fuel Jet Spread Modeling for Numerical of Aircraft Fuel Dumping Simulation

Tsze C. Tai*
U.S. Naval Surface Warfare Center, West Bethesda, Maryland 20817

An analytical jet spread model has been developed for determining the boundary of an axisymmetric jet ejecting into a two-fluid, coflowing stream. When the circumferential velocity of the jet is assumed to be small, the model can be coupled with the conventional computational fluid dynamics (CFD) method for simulation of the fuel dumping from an aircraft. Application of the model is demonstrated by considering the fuel dumping from a V-22 aircraft flying at Mach 0.345 and angles of attack from 7 and 16 deg. The model provides a first-order correction to the CFD simulated results with the jet dispersion boundaries to yield an integrated solution that represents a realistic aircraft fuel dumping environment.

Introduction

DUMPING excessive fuel from an aircraft before landing may be necessary due to safety concerns. During fuel dumping, massive fuel can be successfully dumped without any contact between the fuel jet and the aircraft surface. Fuel mist, which is caused by the thickening of the fuel jet boundary, may impinge onto the aircraft even though the main fuel stream does not touch the surface. The thickening of the jet boundary is commonly known as the jet dispersion, the jet spread, or the jet entrainment. It consists of particles of the surrounding medium carried along with it and particles of the jet itself that have been slowed down.

To avoid any fuel and/or fuel mist impingement onto any part of the aircraft surface during dumping, the geometry of the fuel jettison probe (FJP) must be properly designed and its location carefully determined. This is an important aerodynamic issue requiring resolution by combining the results from wind-tunnel tests, computational fluid dynamics (CFD) simulations, and flight tests.

The role of the CFD simulation is to provide an initial design study regarding the FJP geometry configuration, jet orientation, and its location on the aircraft, and also to gain insights into the complex flowfield about a complete aircraft with the FJP for fuel jettison. The problem is complicated not only by the complex geometry, but also the fuel jet characteristics due to jet dispersion. The jet dispersion, or the jet spread, is a subproblem that is embedded in the overall flowfield. It must be accounted for if the simulation truly represents a realistic aircraft fuel dumping environment. Unfortunately, the present capability of the conventional CFD method[1–4] lacks the ability to solve the whole problem in one formulation. The problem would have to be solved separately for the aircraft flowfield and for the embedded jet flow and then coupled together to yield an overall solution that depicts the physical problem.

In the present paper, an analytical jet spread model for the fuel dispersion is proposed and described. The model can be coupled with the conventional, Navier–Stokes solution-based CFD method for simulation of the aircraft fuel dumping. The coupling is performed by a first-order correction in which the jet boundary due to the jet spread is calculated based on the simulated flowfield, but the flowfield is not updated by the presence of the jet volume. Although the work is specially aimed at the V-22 aircraft fuel dumping, the model is general enough to be applicable to aircraft other than V-22.

Fuel Jet Spread Model

The problem of the fuel jet spread involves a coflowing stream of a jet being surrounded by the air stream in the flowfield. In the development of the model, the theory of an axisymmetric jet in a single-fluid, coflowing stream of Abramovich[5] is extended to a two-fluid, coflowing stream.

The schematic of a jet of two streams moving in the same direction (the coflowing jet) is shown in Fig. 1. The jet exits at the nozzle with an initial velocity u_0 into an external stream that is parallel to the jet with velocity u_H. The jet boundary thickens as it proceeds downstream. Assume the jet thickness along the jet axis increases in proportion to the distance from the mixing origin, that is,

$$\frac{\mathrm{d}r}{\mathrm{d}x} = \frac{c(u_m - u_H)}{u_m + u_H} \tag{1}$$

where c is the thickening coefficient of the submerged isothermal jet and u_m is the maximum velocity at the jet center. Because the velocity u_m is a variable, the slope for the jet growth is curved. Introducing the velocity ratio m, where $m = u_H/u_0$, Eq. (1) can be recast into the following form:

$$\frac{c\,\mathrm{d}x}{\mathrm{d}r} = 1 + \frac{2m(u_0 - u_H)}{(u_m - u_H)(1 - m)} \tag{2}$$

The variable u_m is a function of x (the longitudinal axis of the jet); its variation can be determined with the aid of the conservation of momentum equation along the axis of the jet,

$$\int (u - u_H)\,\mathrm{d}M = \int (u_0 - u_H)\,\mathrm{d}M_0 \tag{3}$$

where u is the jet velocity and M is the mass flux of the jet fluid. The elementary mass rate of the flow at an arbitrary cross section is $\mathrm{d}M = \rho u\,\mathrm{d}F$, and $\mathrm{d}M_0 = \rho_0 u_0\,\mathrm{d}F_0$ at the initial cross section. The subscript 0 denotes quantities at the initial location and $\mathrm{d}F$ is the area of the element; for an axisymmetric jet, $\mathrm{d}F = 2\pi y\,\mathrm{d}y$.

To integrate Eq. (3), expressions for the velocity and density profiles are needed. Numerous experiments in the past show that the velocity profiles in a turbulent jet spreading into an external stream of fluid flowing in the same direction as the jet are similar to the universal velocity profile of the Prandtl–Schlichting theory (see Ref. 6). This classical velocity profile is, therefore, used here, which was also employed by Abramovich,[5]

$$(u_0 - u)/(u_0 - u_H) = (1 - \xi^{1.5})^2 \tag{4}$$

For the variation of the density, a density profile for the two-fluid, coflowing flow in the following form is proposed:

$$\rho/\rho_{\mathrm{jet}} = 1/[1 + (a - 1)\xi^b] \tag{5}$$

Presented as Paper 2003-3929 at the AIAA 21st Applied Aerodynamics Conference, Orlando, FL, 23 June 2003; received 3 September 2003; revision received 4 January 2004; accepted for publication 19 December 2003.

*Senior Research Scientist, Marine and Aviation Department, Carderock Division; taitc@nswccd.navy.mil. Associate Fellow AIAA.

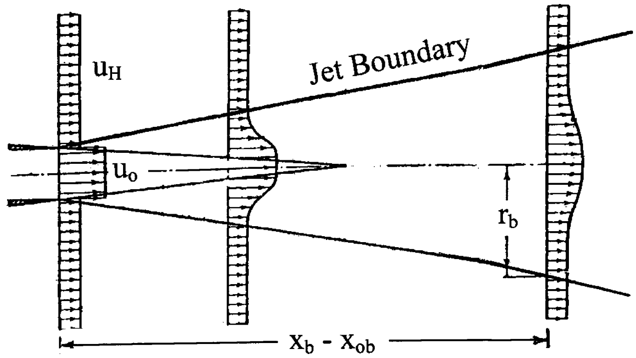


Fig. 1 Coflowing jet.

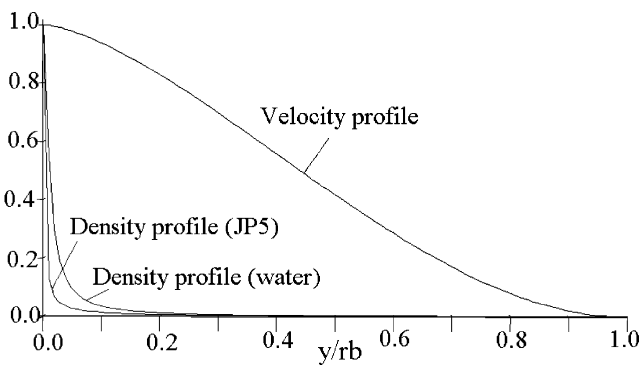


Fig. 2 Velocity and density profiles.

where $\xi = y/r$, a is the density ratio of the jet fluid to air where $a = 673$ for JP5 fuel or 827 for water, and $b = 0.5$–2. The profile satisfies the conditions at the jet center being the JP5 fuel used in most jet aircraft and air, which is the edge of the jet boundary. The parameter b is a constant that must be determined empirically by correlating the analytical results with experimental measurements. This has been performed by Tai and Woods[7] and was found to be $b = 1.0$ for JP5 fuel and 1.5 for water. The reason for using water in the wind-tunnel test is its safety. Figure 2 shows the velocity and density profiles at a given station using the empirically determined value for b.

With the aid of Eqs. (3–5), Eq. (2) can be integrated to yield the following form:

$$0.22(x_b - x_{0b}) = r_b + 1.33A_2\big/\left(A_1 g^2\right)\left[\left(r_b^2 + g^2\right)^{1.5} + r_b^3 - g^3\right] \quad (6)$$

where $g^2 = 4A_2(1 - m)/(mA_1)^2$ and $m = u_H/u_0$,

$$A_1 = 2f\{1/[1 + (a - 1)\xi^b]\}(1 - \xi^{1.5})^2\xi \, d\xi$$

$$A_2 = 2f\{1/[1 + (a - 1)\xi^b]\}(1 - \xi^{1.5})^4\xi \, d\xi$$

The constant c in Eq. (2) has been replaced by 0.22, which was determined by correlation with experimental data.[5] Also, $x_b - x_{0b}$ is the length of the jet from the jet exit and r_b is radius of the jet from center to the jet boundary. Equation (6) is similar to Eq. (5.91) in Ref. 5, except that here A_1 and A_2 contain the variation of density given by Eq. (5).

Three parameters govern the growth of the jet: the velocity ratio $m = u_H/u_0$, the density ratio $a = \rho_{\text{jet}}/\rho_{\text{air}}$, and the empirical constant b. Whereas parameters a and b are more or less fixed for a given jet fluid, the velocity ratio m is the only controllable variable that affects the growth of the jet boundary. Figure 3 shows the effect of the velocity ratio m in terms of the freestream Mach number on the growth of the jet boundary for both JP5 fuel and water, with the initial jet velocity in both cases set at $u_{\text{jet}} = 10$ ft/s. When the surrounding airstream velocity u_H is assumed to be the freestream velocity V_∞, the value for m in the case of $M_\infty = 0.1$ is determined

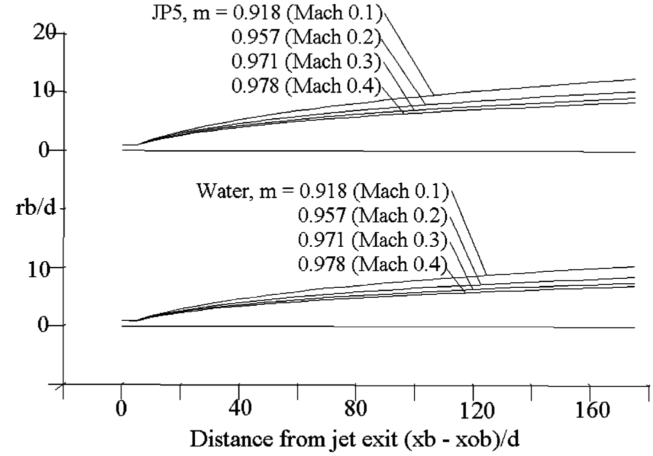


Fig. 3 Effect of velocity ratio on growth of jet boundary.

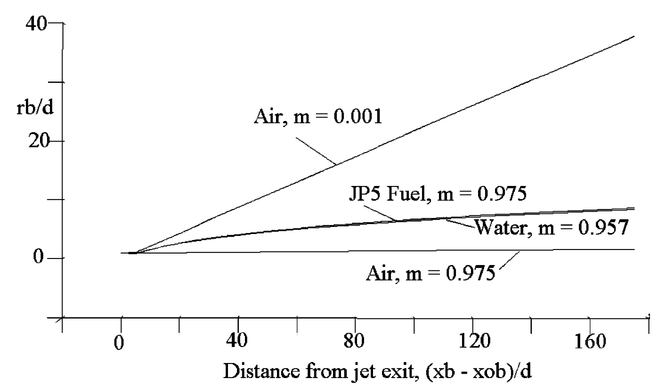


Fig. 4 Growth of jet boundary for JP5 fuel, water, and air.

as follows:

$$m = u_H/u_0 = V_\infty/(V_\infty + u_{\text{jet}})$$

$$= 0.1 \times 1117/(0.1 \times 1117 + 10) = 0.918 \quad (7)$$

As shown in Fig. 3, the lower the freestream Mach number is, the thicker the jet boundary or the fuller the jet spread. It implies that the aircraft would have less risk of being impinged by fuel mist by flying at a higher speed for fuel dumping.

In practice, we have two scenarios to consider in applying the present jet spread model: the flight test and the wind-tunnel test. In the flight-test scenario, the JP5 fuel is dumped from the aircraft flying at Mach 0.345 or $V_\infty = 385$ ft/s. Again, assume the surrounding airstream velocity u_H is the freestream velocity V_∞; then parameters of the jet spread model will be $a = 673$, $b = 1.0$, and $m = 0.975$. In the wind-tunnel test, water representing the jet fuel is dumped from the wind-tunnel model at Mach 0.212 or $V_\infty = 223$ ft/s; the model parameters are $a = 827$, $b = 1.5$, and $m = 957$. As shown in Fig. 4, the jet boundaries for both the flight test (JP5 fuel) and the wind-tunnel test (water) lie very close to one another. Coincidently, the closeness of the resulting jet boundary implies that the wind-tunnel test using water as the jet fluid (low m, high a) should simulate the flight-test condition (high m, low a) very closely. If the wind-tunnel test speed decreases slightly, the two curves should lie on top of one another. Thus, we have reason to believe that if there is no water impingement on the model in the wind-tunnel tests, there would be no fuel impingement onto the aircraft in flight tests. This statement is true only if the flow pattern of the fuel streamlines from the wind-tunnel test is similar to that of the flight test. Theoretically they could be slightly different, depending on the Mach and Reynolds numbers. The investigation of the effect of the Mach and Reynolds numbers on the fuel streamline pattern has not been performed in the present work.

If the jet fluid were air, the jet would remain extremely thin (Air, $m = 0.975$ in Fig. 4). This corresponds to the case for which the boundary of the jet in a coflowing flow is determined by using a constant density ($\rho/\rho_{\text{jet}} = 1$) as given by Eq. (5.91) of Abramovich.[5] When the velocity ratio reduces to nearly zero ($m = 0.001$), that is, the jet is spreading through a medium at rest (Air, $m = 0.001$ in Fig. 4), the jet is said to be submerged.

Coupling of Jet Spread Model to CFD Simulation

The CFD simulation of an aircraft configuration having a fuel jettison probe with an initial fuel jet has been carried out by Tai and Woods.[7] We will use the results of the simulation on one of the FJP configurations here to illustrate how the effect of jet spread can be embedded in the CFD simulation.

The coupling of the jet spread model to the CFD simulation can be performed in two ways: the first-order correction and the second-order coupling. In the first-order correction, the jet spread is calculated based on the CFD simulated flowfield and the resulting jet boundary is then superimposed onto the flowfield, but the flowfield is not updated by the presence of the jet volume. (The jet now has volume after its boundary is calculated.) In the second-order coupling, the same procedure of the first-order correction is pursued except that the flowfield is updated by the presence of the jet volume. In the present work, the coupling is performed in a first-order correction manner, which will be described in some detail.

The flowfield simulated by the CFD method contains flow properties such as density, velocity components, and energy at each mesh point so that other flow functions may be obtained by using the graphics software. The most important flow feature of interest is the flow pattern represented by the instantaneous streamlines in the field, including those emitted at the fuel jettison probe. The instantaneous streamlines originating from the FJP nozzle exit and its vicinity are identified as the fuel lines and stored as such. These curvilinear fuel lines, which are trajectories of particle paths with no volume, will serve as the centerlines of the fuel jet in applying the jet spread model. However, for the equation for determining the axisymmetric jet boundary to be valid along the curvilinear fuel lines, the model needs to be applied in compliance with the axisymmetric analog.

The axisymmetric analog allows the straight jet centerline of an axisymmetric jet to be replaced by the curvilinear fuel jet lines if the transverse velocities of the jet are small. This is sometimes referred to as the small crossflow assumption. Originally proposed by Cooke[8] and later refined by Bradley,[9] the axisymmetric analog simplifies a three-dimensional flow problem to an axisymmetric one by assuming the transverse components of the viscous flow to be small compared with the longitudinal ones. Use of the axisymmetric analog was fairly popular in the 1960s when the numerical solution to the Navier–Stokes equations in three dimensions was prohibitive. The analog provided a good tool to calculate the heat transfer over bodies at an angle of attack in hypersonic flow.[10]

A characteristic feature of the turbulent jets, as shown by theory and also by numerous experiments, is the smallness of the transverse velocity components in any section of the jet compared with the longitudinal velocity. The assumption that the axisymmetric analog is based on is, therefore, valid in case of a jet in a coflowing flow.

When the equation for determining the boundary of an axisymmetric jet is applied along curvilinear fuel lines, the boundary of the fuel jet is obtained. The contour of the boundary is curvilinear because of its curvilinear centerline. The fuel mist, which is more likely than the fuel itself to impinge onto the aircraft surface, is being enveloped inside the jet boundary. The contours of the fuel jet boundaries are then superimposed on the original fuel lines to show the effect of jet spread.

Results and Discussion of Results

Application of the jet spread model to the CFD simulated flowfield is demonstrated by using the flow over the V-22 aircraft configuration with an FJP. The steady-state calculations of the flow over the V-22 configuration with the FJP geometry on the sponson were performed by Tai and Woods.[7] The results for the case of a freestream Mach number at 0.345 and aircraft angles of attack of 7 and 16 deg are used here for demonstration purposes. These conditions yield a Reynolds number of 20.5×10^6 based on the wing chord length of

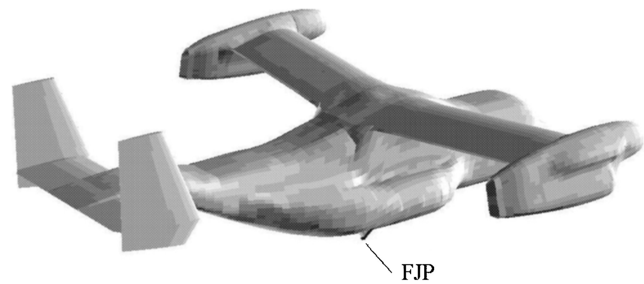
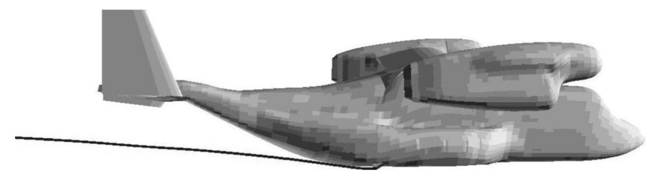
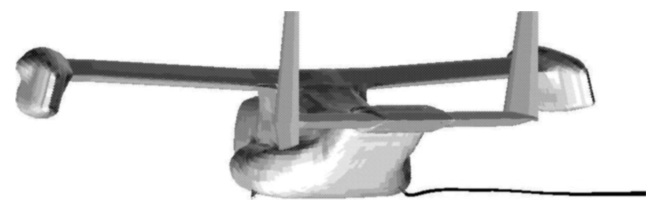


Fig. 5 FJP configuration installed on V-22 sponson.



a)



b)

Fig. 6 Particle traces over V-22 aircraft with a parallel fuel jet FJP at $M_\infty = 0.345$ and $\alpha = 7$ deg; fuel streamlines only: a) side view and b) rear view.

8.33 ft. The flow was assumed fully turbulent. In the simulation, the JP5 fuel was ejected at the FJP exit at 10 ft/s.

The results are presented in the form of particle traces that show the fuel streamlines along with instantaneous streamlines from upstream. The fuel streamlines are of primary interest to see whether they impinge on the aircraft surface. The FJP configuration considered here has a circular exit mounted at the midregion on the sponson, with the fuel being jettisoned in the direction parallel to the external airstream. The exit is about 7.5 in. from the surface of the sponson. Figure 5 shows the FJP configuration installed on the aircraft sponson.

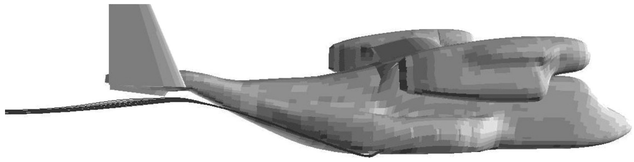
The particle traces of the CFD results computed for the configuration with FJP geometry at a freestream Mach number of 0.345 and an angle of attack of 7 deg (cruise flight) are shown in Fig. 6. The lower part of the vertical tail is not shown in Fig. 6 for easy viewing of the fuel jet position. Only streamlines emanating from the FJP exit area are shown. These, in fact, are fuel streamlines, which contain the primary part of the fuel dumped proceeding directly downstream. Shown in both side and rear views, the fuel lines behave coherently and proceed directly downstream at a distance away from the aircraft surface.

The distance between the core of the fuel lines and the tail varies with the angle of attack due to changes in the resulting flowfield: the higher the angle of attack, the shorter this distance. As the angle of attack increases to 16 deg, the core of the fuel lines approaches closer to the tail (Fig. 7a).

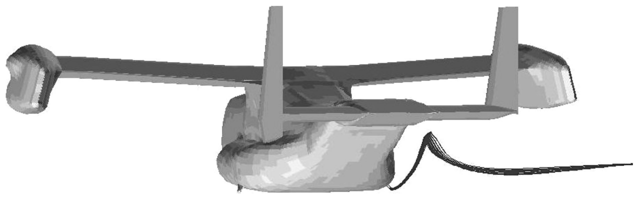
From the side view it looks as if some part of the fuel lines have already touched the tail surface, yet they remain a small distance away from the tail as shown in the rear view. Thus, there is no fuel impingement from the pure CFD simulation.

In reality, however, this small distance represents the distance between the center of the fuel jet and the aircraft surface, rather than the boundary of the jet and the aircraft surface. The boundary of the jet might have already touched the aircraft surface, although the center of the jet has not.

Now we apply the jet spread model to the CFD simulated results by adding the analytically determined fuel jet boundary to

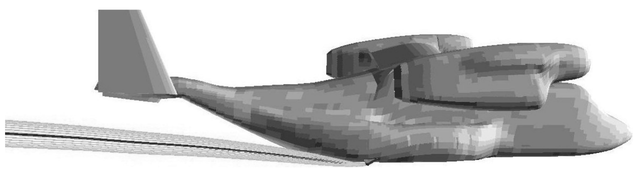


a)

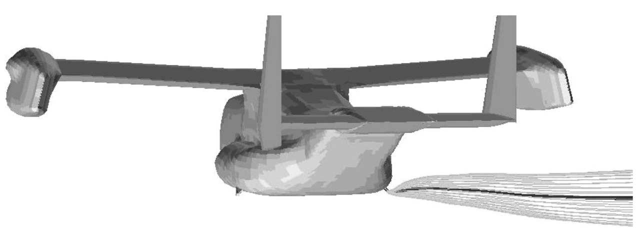


b)

Fig. 7 Particle traces over V-22 aircraft with a parallel fuel jet FJP at $M_\infty = 0.345$ and $\alpha = 16$ deg; fuel streamlines only: a) side view and b) rear view.

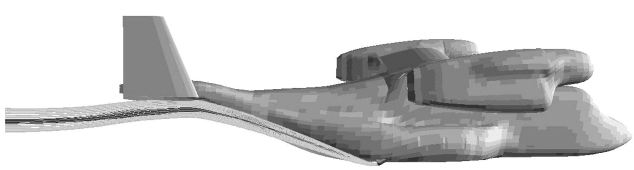


a)

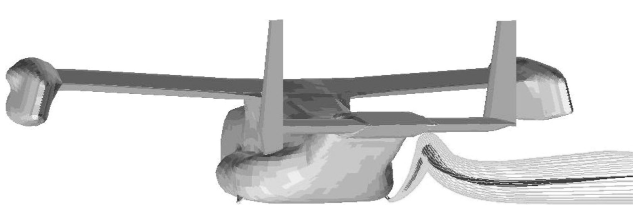


b)

Fig. 8 Particle traces over V-22 aircraft with a parallel fuel jet FJP at $M_\infty = 0.345$ and $\alpha = 7$ deg; fuel streamlines with fuel jet spread effect: a) side view and b) rear view.



a)



b)

Fig. 9 Particle traces over V-22 aircraft with a parallel fuel jet FJP at $M_\infty = 0.345$ and $\alpha = 16$ deg; fuel streamlines with fuel jet spread effect: a) side view and b) rear view.

the preceding fuel streamlines. The boundaries were determined by solving Eq. (6) using simple numerical integration methods[11] with variables set at $a = 673$ for JP5 fuel and $b = 1$. When the jet boundary is added, the fuel jet now possesses a volume filled with the fuel mist. The flow patterns of Figs. 6 and 7 are then updated to become those shown in Figs. 8 and 9, respectively. The original fuel lines are dark, whereas the jet boundaries are shown lighter.

Although it is clear that there is no fuel impingement on the aircraft at 7-deg angle of attack (Fig. 8), there is great possibility that impingement of the fuel mist would occur at angle of attack of 16 deg, where the fuel jet boundary barely touches the lower horizontal tail surface (Fig. 9). Thus, the modeling indicates that the fuel would not impinge on the aircraft at low angles of attack. However, impingement is indicated to occur for an angle of attack of 16 deg. In other words, to use this FJP configuration for fuel dumping, the aircraft must be flying at an angle of attack lower than 16 deg. Fuel dumping at high angles of attack would need to be avoided to preclude fuel impingement.

The present work is the first time that the CFD technique has been successfully used to include the effect of the fuel jet spread on the aircraft fuel dumping problem. Although the V-22 aircraft fuel dumping was considered, the jet spread model is also applicable to other types of jet aircraft.

Conclusions

An analytical jet spread model has been developed that can be coupled with a conventional CFD method for simulating the fuel dumping from aircraft. Based on the results obtained in the illustrative examples, the findings are as follows.

1) The jet spread model effectively determines the boundary of the fuel jet using the fuel streamlines as the jet centerlines.

2) The effect of the jet spread must be considered and included in the CFD simulation method to model aircraft fuel dumping realistically.

It is believed that the present CFD method coupled with the newly developed jet spread model may serve as a practical tool with reasonable accuracy and computer cost in providing a practical tool for aircraft fuel dumping.

Acknowledgments

The present work was supported by the Naval Air Systems Command (NAVAIR 4.3.2) under the cognizance of S. A. Woods. The Department of Defense High Performance Computing facilities located at Naval Oceanographic Center provided the Cray SV1 CPU time. The author wishes to thank S. A. Woods and D. Bein of NAVAIR, M. A. McVeigh and J. Liu of The Boeing Company–Philadelphia, and K. Tai of Massachusetts Institute of Technology for useful discussions and comments.

References

[1]Krist, S. L., Biedron, R. T., and Rumsey, C. L., "CFL3D User's Manual (Version 5.0)," NASA TM-1998-2089, NASA Langley Research Center, Jan. 1998.

[2]Buning, P. G., Jespersen, D. C., Pulliam, T. H., Klopfer, G. H., Chan, W. M., Slotnick, J. P., Krist, S. E., and Renze, K. J., *OVERFLOW User's Manual (Version 1.8s)*, 2000.

[3]Power, G. D., and Underwood, M. L., "WIND 2.0: Progress on an Applications-Oriented CFD Code," AIAA Paper 99-3212, June 1999.

[4]Strang, W. Z., Tomaro, R. F., and Grismer, M. J., "The Defining Methods of Cobalt: A Parallel, Implicit, Unstructured Euler/Navier–Stokes Flow Solver," AIAA Paper 99-0786, Jan. 1999.

[5]Abramovich, G. N., *The Theory of Turbulent Jets*, MIT Press, Cambridge, MA, 1963, Chap. 5.

[6]Schlichting, H., *Boundary Layer Theory*, translated by J. Kestin, 7th ed., McGraw–Hill, New York, 1987, pp. 729–757.

[7]Tai, T. C., and Woods, S. A., "Simulation and Analysis of V-22 Aircraft Fuel Dumping," Carderock Div., Rept. NSWCCD-50-TR-2003/054, Naval Surface Warfare Center, West Bethesda, MD, Oct. 2003.

[8]Cooke, J. C., "An Axially Symmetric Analogue for General Three-Dimensional Boundary Layers," Aeronautical Research Council R&M, No. 3200, London, June 1959.

[9]Bradley, R. G., "Approximate Solutions for Compressible Turbulent Boundary Layers in Three-Dimensional Flow," *AIAA Journal*, Vol. 6, No. 5, 1968, pp. 859–864.

[10]DeJarnette, F. R., and Tai, T. C., "A Method for Calculating Laminar and Turbulent Convective Heat Transfer over Bodies at an Angle of Attack," NASA CR 101678, March 1969.

[11]James, M. L., Smith, G. M., and Wolford, J. C., *Applied Numerical Methods for Digital Computation*, 3 ed., Harper and Row, New York, 1985, pp. 377–456.