DOT/FAA/AM-04/7

Office of Aerospace Medicine
Washington, DC 20591

CFD Validation for Contaminant Transport in Aircraft Cabin Ventilation Flow Fields

Robert P. Garner
Federal Aviation Administration
Civil Aerospace Medical Institute
Oklahoma City, Oklahoma 73125

Kwai L. Wong
Shawn C. Ericson
UT Joint Institute for Computational Sciences
Oak Ridge National Laboratory
Oak Ridge, Tennessee 37831

A.J. Baker
Joseph A. Orzechowski
University of Tennessee
Mechanical, Aerospace, and Engineering Sciences Department
Knoxville, Tennessee 37996

April 2004

Final Report

This document is available to the public
through the National Technical Information
Service, Springfield, Virginia 22161.

U.S. Department
of Transportation

Federal Aviation
Administration

NOTICE

This document is disseminated under the sponsorship of the U.S. Department of Transportation in the interest of information exchange. The United States Government assumes no liability for the contents thereof.

Technical Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
DOT/FAA/AM-04/7		

4. Title and Subtitle	5. Report Date
CFD Validation for Contaminant Transport in Aircraft Cabin Ventilation Flow Fields	April 2004
	6. Performing Organization Code
7. Author(s) Garner RP[1], Wong KL[2], Ericson SC[2], Baker AJ[3], Orzechowski JA[3]	8. Performing Organization Report No.
9. Performing Organization Name and Address FAA Civil Aerospace Medical Institute P.O. Box 25082 Oklahoma City, OK 73125	10. Work Unit No. (TRAIS) 11. Contract or Grant No.
12. Sponsoring Agency name and Address [1]Office of Aerospace Medicine, Federal Aviation Administration 800 Independence Ave, S.W., Washington, DC 20591 [2]UT Joint Institute for Computational Sciences Oak Ridge National Laboratory, Oak Ridge, Tennessee 37831 [3]University of Tennessee Computational Fluid Dynamics Laboratory Mechanical, Aerospace, and Engineering Sciences Department Knoxville, Tennessee 37996	13. Type of Report and Period Covered 14. Sponsoring Agency Code

15. Supplemental Notes

16. Abstract

Civil transport aircraft have clearly been demonstrated as a preferred target of terrorist organizations through the years. The threat of the release of a noxious chemical or biological agent into the passenger cabin is real. Protection of occupants is critical for maintaining public confidence in air travel. Therefore, the Federal Aviation Administration (FAA) Civil Aerospace Medical Institute (CAMI) is directing a project for quantitative evaluation of the distribution of contaminants released during mid-flight in commercial airliner passenger cabins. The effort uses the CAMI 747 Aircraft Environmental Research Facility (AERF) to collect airflow data to be used to validate computational fluid dynamics (CFD) algorithms that can predict the potential distribution of a variety of particles in the aircraft. The cabin velocity vector field is measured at a number of points within the cabin using 3-dimensional sonic and conventional hot-wire anemometers. These data serve as a comparison basis to simulation via CFD that predicts the 3-dimensional ventilation flow field. This paper presents the first results of validation of CFD prediction of the cabin flow field in a segment of a 747 AERF. Agreement between the data and the simulation is good, hence represents the first steps towards development of a CFD system to quantitatively study contaminant distribution and assist in testing optimal responses to an attack on various airframes. The release of a chemical weapon has been the focus of work to date, but CFD is ultimately applicable to predicting the distribution of other entities through the aircraft environmental control system (ECS) such as biological, nuclear, or other toxic agents. Concurrently, the approach is applicable to generic cabin air quality issues where an understanding of flow fields is of critical importance in terms of the role that fresh air and contaminant levels play in passenger comfort, health, and safety.

17. Key Words	18. Distribution Statement
Cabin Airflow, Computational Fluid Dynamic, Model Validation, Aerosol Distribution, Coanda Effect	Document is available to the public through the National Technical Information Service; Springfield, VA 22161

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	9	

Form **DOT F 1700.7** (8-72) Reproduction of completed page authorized

CFD Validation for Contaminant Transport in Aircraft Cabin Ventilation Flow Fields

INTRODUCTION

The risk of release of a noxious material into a commercial aircraft cabin during flight is significant due to the worldwide terrorist threat targeting civilians. Reaction management of this potential disaster must be based on quantitative analyses of potential scenarios affecting the cabin environment. As opposed to in-flight testing, which is prohibitively expensive and logistically awkward, a mathematical modeling with validation component has been proposed.

Development of an accurate simulation of the flow field in a cabin space utilizing computational fluid dynamics (CFD) is a significant challenge. First, the numerical analysis requires tremendous computing power to solve the sets of equations in a reasonable time frame. Second, the complexity of the environment must be handled. Any obstructions have to be taken into consideration since they can influence the flow field and add to the computational task. For these reasons initial steps towards the development of a transport aircraft cabin model approach were made using the 747 Aircraft Environmental Research Facility (AERF, Figure 1) at the FAA Civil Aerospace Medical Institute.

METHODS

The CFD approach is potentially viable for predicting the 3-dimensional, very low (2-5) air-change per hour (ACH) flow fields associated with aircraft environmental control system (ECS) operation in commercial transports. A CFD model has been identified describing the unsteady, time-accurate, buoyant ventilation flow field in an aircraft cabin at cruise conditions, capable of execution using an augmented laminar Taylor-stabilized finite turbulent kinetic energy (TKE) model. CFD simulations were conducted using a finite element theory implementation of this model, via the University of Tennessee CFD Lab *PICMSS* (Parallel Interoperable Computational Mechanics Simulation System) code. Comparison simulations have used an available commercial code (*Fluent*).

The comparative validation data were collected utilizing principally 3D sonic anemometers (Campbell Scientific, CSAT3) in an isolated cabin segment of the 747-AERF, which was devoid of seats. Figure 2 provides an overview of the data collection approach. The selected sonic anemometers simultaneously measured the 3 Cartesian components of the velocity field and were sensitive to velocity magnitude as low as 0.1 cm/sec. The experimental protocol was established to guarantee freedom from drift over the lengthy data collection period. Additional hot wire data were taken in the immediate vicinity of the supply jet entry into the cabin. Flow uni-directionality and speed in the immediate vicinity of the inlet made this experimental methodology viable.

Data were collected in parallel sections that ran the length of the segment utilized. At each transverse plane station, the data acquisition period was 7 minutes with recording rate of 30 samples/second. Within the 3-5 minute window, the sampled data were resolved into mean and fluctuating components on 1-second intervals. The mean data were subsequently averaged over four 30-second periods to generate the essential flow field picture.

Figure 1. Boeing 747-100 modified for use as Aircraft Environmental Research Facility (ERF). The facility can be modified to meet a variety of research and educational requirements.

RESULTS

Figure 3 shows the totally averaged transverse plane velocity vector field superposed on an outline generated from a CFD mesh. Several z-station data sets confirm existence of the Coanda effect, with the supply jet remaining attached to the underside of the luggage bin, hence clearly penetrating the cabin to the center luggage carrier. The resulting circulating flow field encompasses the entire cabin half span, with a smaller counter-rotating flow field generated in the upper region between luggage carriers. The character of this flow pattern does not exhibit significant z-dependence, as expected, based

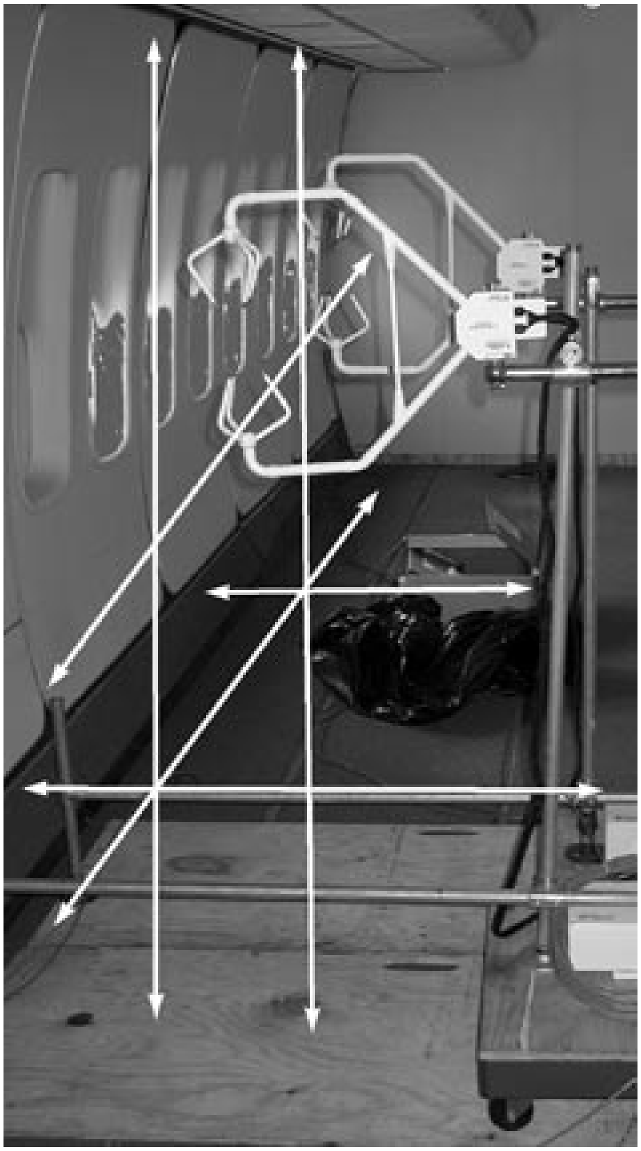


Figure 2. Overview of the movement of the 3D anemometers utilized for data collection. Coordinate reference points in the segment utilized was X=0 in the center of the cabin, Y=0 at the floor of the cabin, and Z=0 at the bulkhead that had been installed in the rear of the segment.

on the geometric simplicity of the empty cabin and its associated symmetries. The following comments summarize the acquired cabin airflow data:

1. The velocity field is definitely time varying and unsteady, with a periodic unsteadiness of the order 3-4 minutes.
2. The mean velocity extrema within the Coanda jet region are ~1 m/s, while an average speed in the cabin proper is less than ~0.2 m/s.
3. The rms data level holds steady at ~0.006–0.01 m/s, hence the average turbulence kinetic energy k is ~ 0.0006 m^2/s^2.
4. For average mean flow magnitude of ~0.15 m/s, the average turbulence level is of the order ~6 %.

CFD Predictions. CFD predictions for the cabin interior flow field were conducted using 2 different code-closure model systems. The computations were 3 dimensional; however, z-symmetry plane boundary conditions produced flow fields lacking z-dependence. Hence, CFD solutions on all z-planes are identical, the transverse plane discretization of which is shown in Figure 4 in a global view including magnifications to illustrate supply and exhaust ducts meshing details. These local geometries were directly measured for the CAMI 747 AERF. The highly non-uniform transverse mesh contained ~14,000 node points per plane. The mesh was identical in every z-plane.

As stated, design of the 747 AERF ventilation supply relies on the Coanda effect to maintain supply jet attachment to the overhead luggage carrier until the flow reaches the first break in the carrier geometry. The experimental data clearly show this, and the supply jet penetrates to the center of the cabin until encountering the face of the central luggage carrier.

The CFD prediction generated using the time-accurate CFD formulations stabilized with a modified Taylor finite element approximation (Williams and Baker, 1996), as implemented in *PICMSS* code (Wong et al., 2002), captures the flow phenomena for Re/L = 1000. The flow field does not come to a steady state after 2000s of simulation but oscillates about the essential velocity vector field as shown in Figure 5a. Figure 5b contains a magnification of the velocity field in the supply duct region exhausting into the cabin, and flow field attachment to the luggage carrier bottom is essentially steady.

Predictions for comparison were made using the Fluent commercial code - finite volume implementation of a non-time accurate CFD theory. The Fluent simulation used the standard TKE closure model. The solution at-

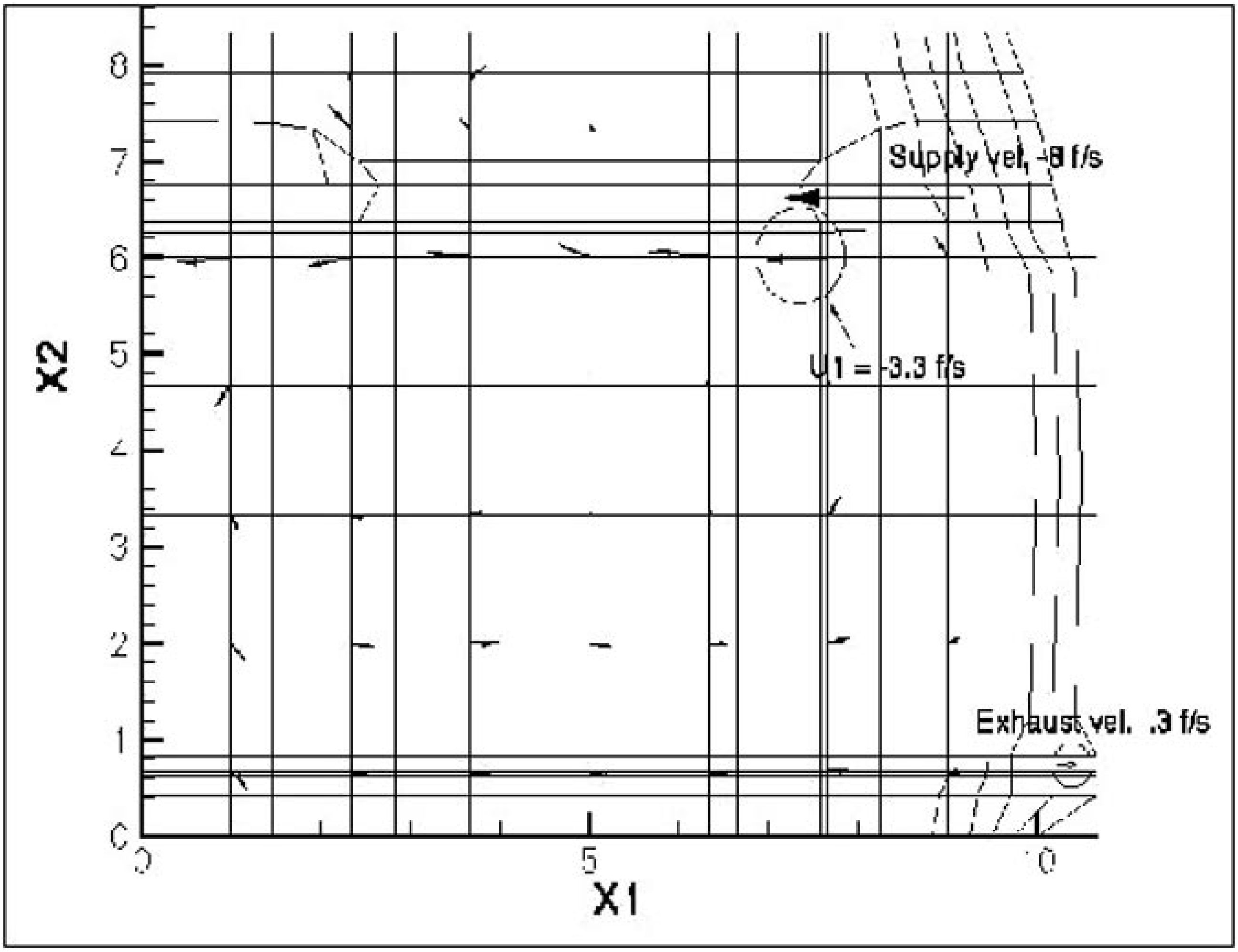


Figure 3. The averaged transverse plane velocity vector field.

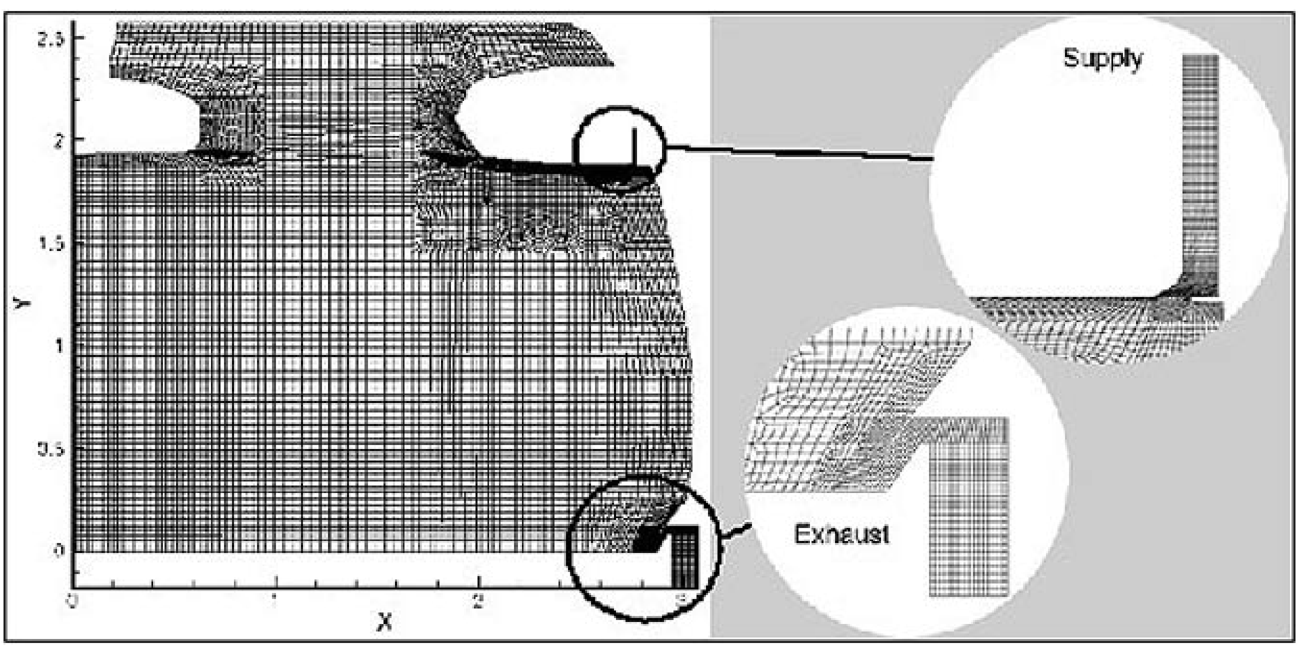
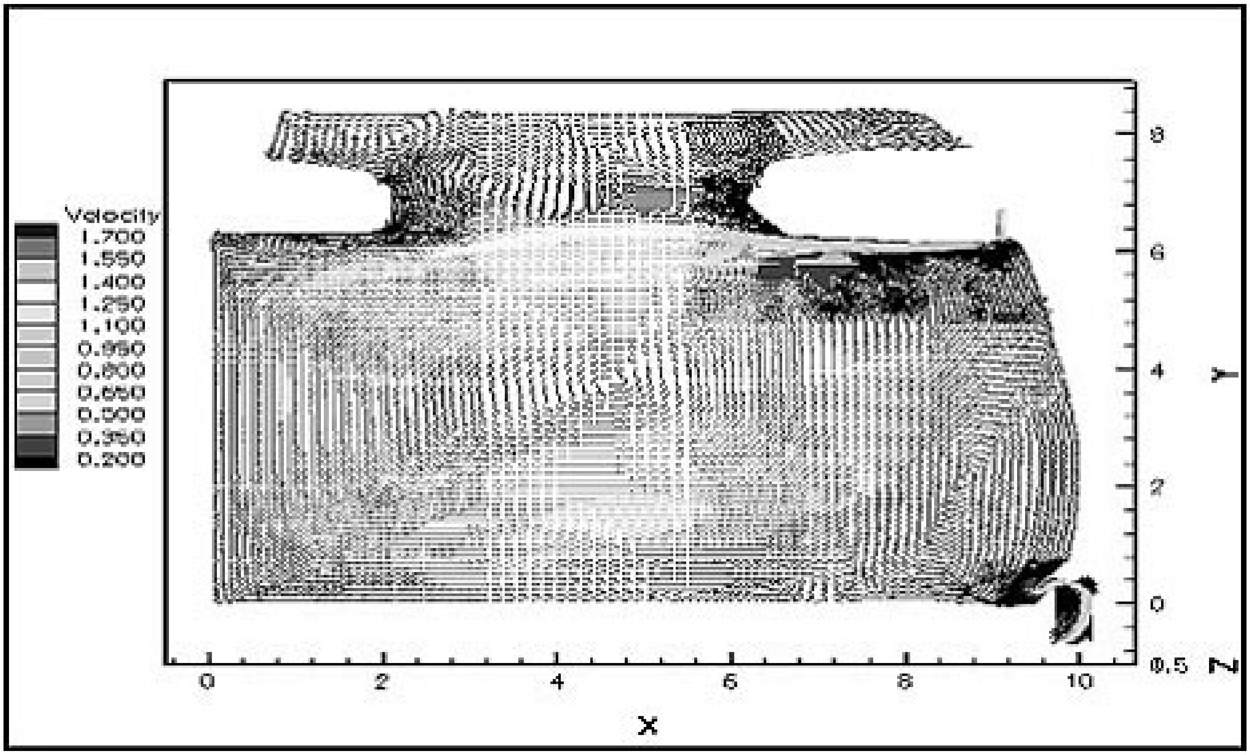


Figure 4. Cross section of the cabin discretization with close up views of the supply and exhaust ducts.



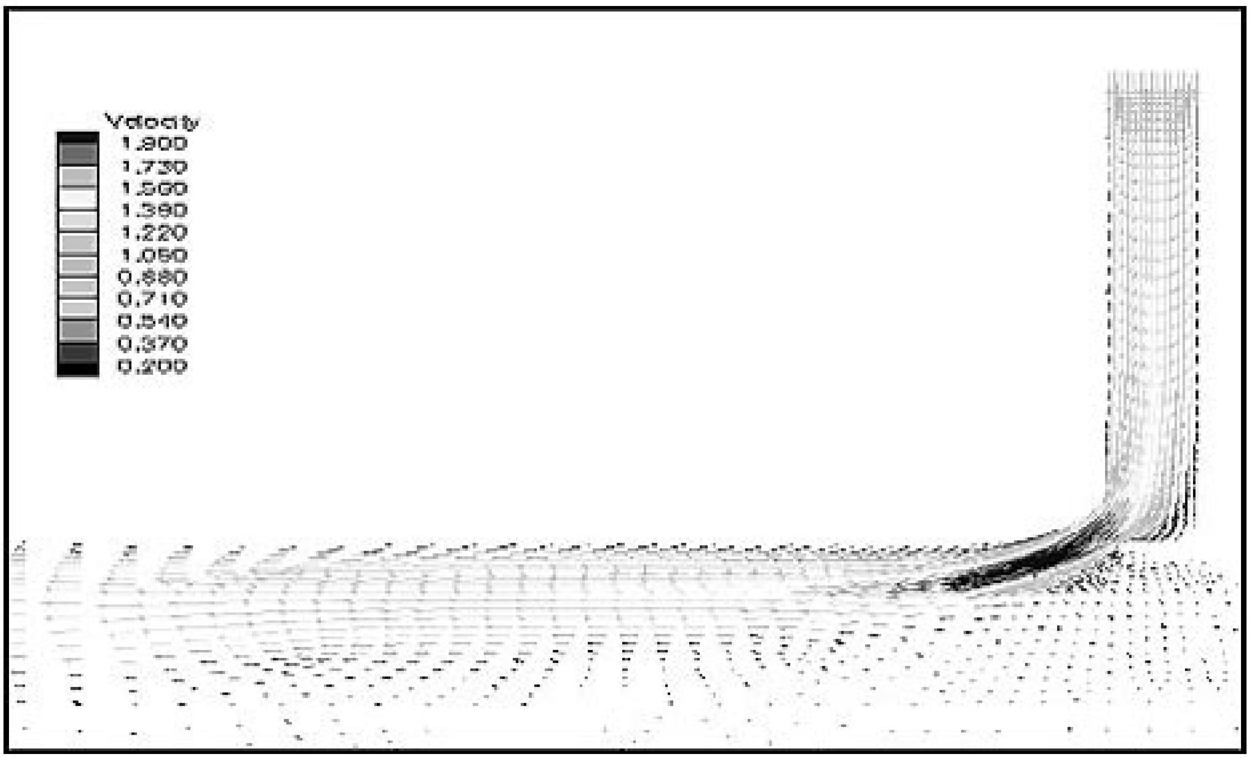


Figure 5. a) PICMSS FE simulation at a Re/L = 1000. b) Velocity vectors in the area of the air inlet supply region illustrating Coanda effect flow attachment to the underside of the luggage bin.

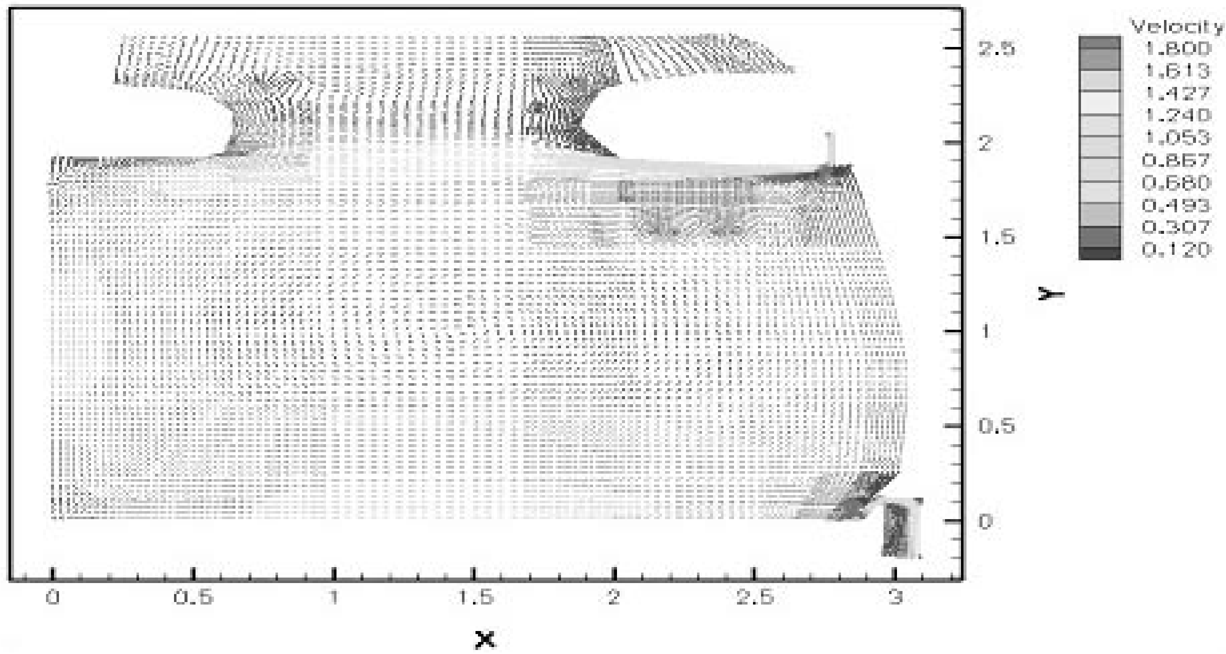


Figure 6. Steady flow field solution from the commercially available code *Fluent*.

tained steady state using the third-order stabilization scheme. The supply jet penetrates between the luggage carriers, Figure 6, and the recirculation region spanned the entire cabin region; hence this flow field resembles the averaged experimental data.

DISCUSSION

It would seem prudent that protective strategies against the threat of a chemical or biological weapons attack on a commercial transport aircraft should be developed. Due to the complexity of the cabin environment (airframe, flight altitude, etc.), judgments on any response to a given situation are subjective at best. Time accurate simulations will allow a variety of emergency circumstances to be evaluated in a quantitative manner. Insight into the best combination of actions to take in face of a particular challenge can be built. This approach also allows the response to any threat that may be anticipated to be integrated with current emergency protective equipment and procedures as closely as possible. This would assist in making protective systems aboard aircraft as efficient as possible in terms of both cost and function.

Accurate simulations are also applicable to day-to-day operations. Cabin air quality is a very important issue for both the passengers and crewmembers. Advanced models have the potential to be used for optimizing aircraft environmental control system (ECS) parameters for both current and future aircraft. They could also provide the ability to study the specifics of a contamination accident after it has occurred. Both quantitative details and remedial steps could be examined.

CONCLUSIONS

A CFD model has been developed to predict the 3-dimensional commercial aircraft cabin flow field characteristic of a Boeing 747. Validation data, as obtained in this configuration, confirm the time-accurate, optimally stabilized results predicted via the PICMSS implementation. The CFD algorithm designed for this application produced an unsteady prediction in agreement with data. Results obtained using a commercial code also produced an average flow field comparable to the experimental data and the PICMSS implementation.

REFERENCES

Williams, P.T., and Baker, A.J. (1996). Incompressible computational fluid dynamics and the continuity constraint method for the 3-D Navier-Stokes equations, *J. Numerical Heat Transfer, Part B,* 29,137-273.

Wong, K.L., and Baker, A.J. (2002). A modular collaborative parallel CFD workbench, *Journal of Supercomputing*, 22, 45-53.