

Computational Fluid Dynamics Analysis for the Waste and Hygiene Compartment in the International Space Station

Chang H. Son
The Boeing Company

Evgueni M. Smirnov, Nikolay G. Ivanov and Denis S. Telnov
New Technologies and Services, Russia

Copyright © 2008 SAE International

ABSTRACT

Computational Fluid Dynamics airflow models for the Waste and Hygiene Compartment (WHC) in the U.S. Laboratory module and Node 3 were developed and examined. The International Space Station (ISS) currently provides human waste collection and hygiene facilities in the Russian Segment Service Module (SM) which supports a three person crew. An additional set of Russian hardware, known as the system, is planned for the United States Operational Segment (USOS) to support expansion of the crew to six persons. Integration of the Russian system into the USOS incorporates direct Environmental Control and Life Support System (ECLSS) interfaces to allow more autonomous operation. A preliminary design concept was used to create a geometry model to evaluate the air interaction with the module cabin at varied locations and performance of the avionics fan placed in WHC. The Russian and the privacy protection bump-outs (Kabin) were included into the present modeling. A crew-member model was used in the analysis. Ventilation performance for the cabin aisle way was addressed with regard to the WHC cabin air velocity requirements. Computational results for airflow from a crew safety perspective are presented and discussed.

INTRODUCTION

The International Space Station (ISS) ventilation system performance is an important factor for on-board crew health support. Each Environmental Control and Life Support System (ECLSS) zone should be tested from the ventilation effectiveness point of view and consequently, the crew safety perspective. Special attention should be paid to the WHC ventilation because the compartment is supplied with the privacy protection bump-outs, Kabin, that separate WHC from the cabin and prevent airflow in the WHC. To force air circulation inside WHC and to intensify air exchange between the compartment and the cabin, an avionics fan discharge

and intake are placed inside the WHC. The goal of this study is the assessment of the overall ventilation characteristics of the WHC.

Computational Fluid Dynamics (CFD) modeling of the WHC airflow was performed for two WHC locations: in the U.S. Laboratory and in Node 3. Four computational cases were considered. For each case a geometrical model of the WHC was merged into the corresponding module model. A crew-member model, ECLOID (ECLSS AndROID) was incorporated into the WHC model as well. Two possible locations of the avionics fan intake, aft and starboard, were considered based on a preliminary design WHC concept. Two values of WHC air flow rate provided by the fan were considered including an initial rate of 102 m³/h and a modified flow rate (reduced by approximately 30%). The list of the cases computed with particular problem formulation aspects is provided in Table 1.

Table 1. List of cases computed.

Case	ISS Module	Active WHC outlet	WHC flow rate, m ³ /h
1	USL	aft	102
2	USL	forward	102
3	USL	aft	68
4	Node 3	aft	68

The CFD models take into account the cabin aisle volume only. All the computations have been performed with the FLUENT 6.1 CFD software [1].

COMPUTATIONAL MODELS

The geometry models of U.S. Laboratory and Node 3 adopted for the present CFD analysis of the interior cabin volume ventilation are illustrated in Figure 1. The WHC details and the ECLOID are marked in the plots. The WHC details, the ACY and the Kabin privacy bump-outs, were included into the present modeling with simplifications.

The U.S. Laboratory inlet and outlet boundaries are notated in Figure 1, a. Six Common Diffusers (CD) are placed in the ceiling standoff faces. Six Return Grilles (RG) are placed in the floor standoff faces. The Inter-Module Ventilation (IMV) diffuser is in the aft endcone and it provides air from the Node 1 supply air ducts through an IMV valve.

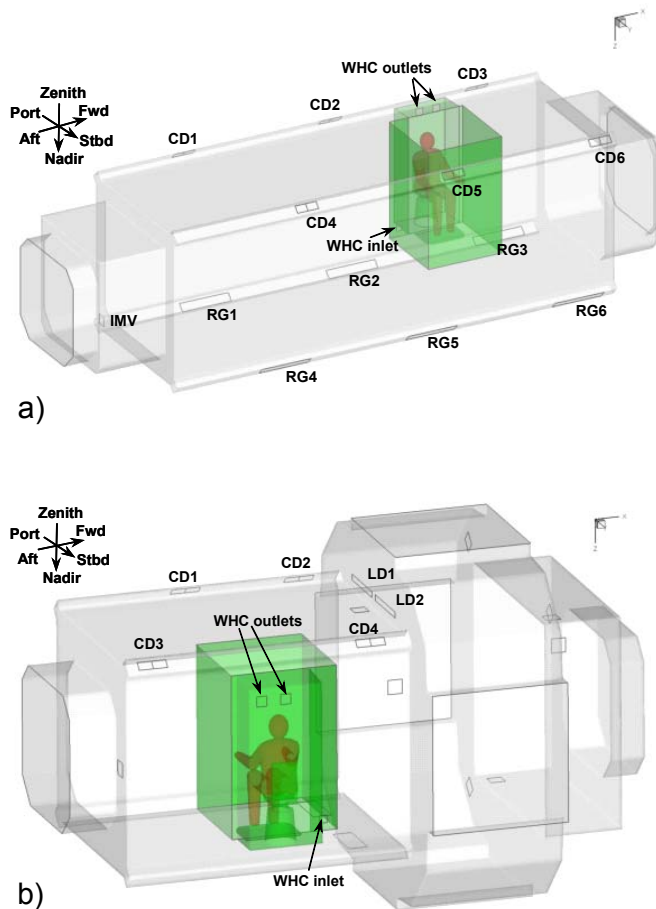


Figure 1. The complete geometrical models for CFD ventilation analysis: (a) U.S. Laboratory and (b) Node 3 with WHC and ECLOID.

The Node 3 inlet and outlet boundaries are shown in Figure 1, b. CD 1 and CD 2 are placed in the port rack bay standoff faces stretched in the longitudinal direction.

CD 3 and CD 4 are placed in the starboard rack bay standoff face. Two return grilles are also placed in the midbay, in the nadir wall of the module. LD 1 and LD 2 are arranged in the radial bay.

In both modules the regular ventilation system is supplemented with the WHC ventilation system that consists of one diffuser and two RG's, aft and forward. Depending on fan location, one of the WHC RG's is open and the other is closed.

The air ($\rho = 1.225 \text{ kg/m}^3$, $\mu = 1.8 \times 10^{-5} \text{ kg/m-s}$) is assumed to be an incompressible fluid. All the cases were computed with steady-state formulation.

The no-slip boundary condition is imposed on all of the solid U.S. Laboratory and Node 3 surfaces and on the ECLOID surfaces as well.

The inlet/outlet boundary conditions are as follows. For the U.S. Laboratory the starboard CD's direct the flow across the ceiling of the rack bay (the deviation from the normal to the diffuser surface is 25°), while the port CD's direct the flow across the port side wall (with the same deviation from the normal to the diffuser surface). The starboard CD adjacent to the forward endcone is closed. The flow from each operating CD is divided into two parts with flow angles of 30° and -30° in the aft-forward direction. The given velocity inlet boundary condition is set for each half of each diffuser with the velocity specification method of velocity magnitude and flow direction components. The same boundary condition is set at the IMV diffuser surface. This diffuser directs the flow with the deviation of 15° down in the nadir direction. The velocity inlet boundary condition is set also at the RG surfaces (being, in fact, outlets for the flow domain analyzed). The forward bulkhead to Node 2 is assumed to be closed and is treated as a solid wall. The aft hatch to Node 1 is open with the pressure outlet boundary condition set at the hatch surface.

For Node 3 the CD's direct the flow in the same way as the U.S. Laboratory CD's. The velocity inlet boundary condition is also set for both LD 1 and LD 2 and for the RG's, treated as outlets for the flow domain analyzed, with the velocity specification method of velocity magnitude normal to the boundary. The pressure outlet boundary condition is prescribed at the Node 3 hatch surface.

For the WHC ventilation, the velocity inlet boundary condition with velocity vectors normal to the boundary is set at the inlet and outlet surfaces. The closed return grille is treated as a solid wall. The inlet/outlet velocity boundary conditions adopted for the U.S. Laboratory are summarized in Table 2, and for Node 3 – in Table 3.

The computational grids generated for U.S. Laboratory and Node 3 have similar topology. Each grid is fully

Copyright © 2008 The Boeing Company. All rights reserved
unstructured and consists of both tetrahedral and hexahedral mesh elements. The grids were created using the GAMBIT 2.3.16 generator [2]. Figure 2 shows a general view of the grid generated for the case with the WHC placed in Node 3 (Case 5). The number of cells for each of the cases analyzed is given in Table 4.

Table 2. Flow rates and boundary conditions; U.S. Laboratory.

	Flow rate, m³/h	Velocity magnitude, m/s
Starboard and Port Common Diffusers	93.5 (×5)	1.07
Aft IMV Diffuser	212	3.5
Return Grille	-78 (×6)	-0.24
Forward hatch boundary (Node 2)	Closed	Wall
Aft hatch boundary (Node 1)	-212	Pressure outlet BC
WHC Diffuser	102 / 68	4.9 / 3.3
WHC Outlet	-102 / -68	-2.7 / -1.8

Table 3. Flow rates and boundary conditions; Node 3.

	Flow rate, m³/h	Velocity magnitude, m/s
Port Common Diffusers	119 (×2)	1.7
Common Diffuser	153 (×2)	2.2
Linear Diffuser	221 (×2)	1.7
Return Grilles	-374 (×2)	-1.1
Forward hatch boundary	230	Pressure outlet BC
WHC Diffuser	68	3.3
WHC Outlet	-68	-1.8

Table 4. Computational Grids.

Module	Number of Grid Cells
U.S. Laboratory with WHC (Cases 1, 2 and 3)	772,301
Node 3 with WHC (Case 4)	799,299

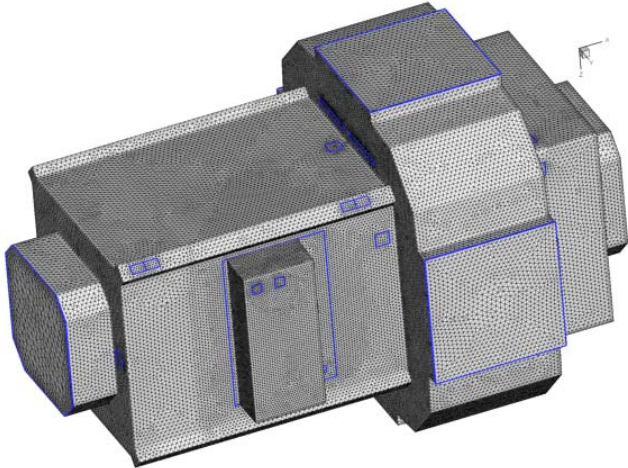


Figure 2. An outside view of the computational grid used for Case 3.

The Reynolds-Averaged Navier-Stokes (RANS) approach was employed. The standard k-ε model, with the standard wall functions, was used for computations [3]. It was impossible for the U.S. Laboratory and Node 3 ventilation to obtain a fully-converged steady-state solution with the default formulation of the standard k-ε model, suggesting the model constant $C_{\mu} = 0.09$. However, in the present computations the value of C_{μ} was increased up to 0.15 based on experience accumulated previously at RANS modeling of the Columbus module [4]. Test computations in ventilation configurations for which steady-state solutions could be obtained with the standard C_{μ} -value also showed that such an increase in the model constant does not significantly influence the averaged flow field.

For all the cases computed, the wall distance of a cell center adjacent to a solid wall measured in wall units, y_p^+ , ranged from 20 to 50 over the majority of the solid walls. The inlet turbulence intensity was taken as 10% for all the diffusers while the inlet ratio of the turbulent to molecular viscosity, ν_{tur}/ν , was varied from diffuser to diffuser to ensure that the inlet-jet effective Reynolds numbers, $Re_{eff} = V_{in} L_s / (\nu + \nu_{tur})$, are within the range from 200 to 300. Here, V_{in} is the inlet velocity value, and L_s is the inlet length scale (the diffuser width).