

A CFD-Based Procedure for Evaluation of Ventilation of a Suddenly-Opened Closeout Space and Its Application to 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

The aim of the study is to understand risks associated with a crew member accessing behind the closeout panel or space. Since there is a possibility that a particular closeout space is filled with a harmful gas mixture that is different from the cabin atmosphere, the time needed to ventilate this space should be evaluated. The three-dimensional Computational Fluid Dynamics (CFD) model developed for prediction of time-dependent turbulent flow and concentration fields inside and near a suddenly-opened box is described in the paper. Several cases with different positions of the closeout space, initially filled with pure oxygen, are analyzed under the U.S. Laboratory (USL) ventilation conditions. A simplified flow model, where a suddenly-opened box is ventilated by uniform external throughflow, is considered as well. Parametric computations performed for this case has resulted in a special procedure for quick evaluation of the closeout-space purging process, using solely the steady-state ventilation velocity distributions, computed previously for the USL module.

INTRODUCTION

During maintenance operations of the International Space Station (ISS) modules, crew members need to access behind the closeout panel or space, such as rack or endcone areas. Normally these spaces are closed, and there is a possibility that a particular closeout space is filled with a gas mixture that is leaked from the system in it and, therefore, different from the cabin atmosphere. Sometimes, there is a possibility that crew members risk exposure to an unbalanced gas concentration while they are accessing into the closeout space. In order to avoid this, related instructions have to be developed.

The objective of this study is to develop a three-dimensional (3D) Computational Fluid Dynamics (CFD) model for accurate prediction of time-dependent flow

and concentration fields inside and near a suddenly-opened box.

Below a simulated closeout space is initially filled with pure oxygen. In application to the U.S. Laboratory (USL) case, detailed analyses were performed for several positions of the closeout space with considerably different local velocities of the cabin ventilation in addition to the purging processes. Additionally, a simplified time-dependent flow case, with ventilation of a suddenly-opened box by uniform throughflow, is considered. Varying the throughflow velocity, a number of time-dependent CFD solutions were obtained for this simplified flow model, and these solutions were used to develop a special procedure for quick evaluation of the purging process as applied to the closeout spaces of the USL module. All the modelling was performed using the FLUENT Version 6.3.26 software package [1].

COMPUTATIONAL MODEL FOR U.S. LABORATORY CLOSEOUT SPACE

Computational Fluid Dynamics (CFD) modeling of the USL airflow was performed for two closeout space placements including a rack position, and an endcone position. Two ventilation scenarios were considered for the nominal ventilation with all Common Diffusers (CD) operating as well as an off-nominal regime when one of the CD's, adjacent to the closeout space is closed. Both the ventilation scenarios were considered for the rack closeout space location, while only the off-nominal regime was examined for the endcone closeout space location. The list of the three cases computed with some problem formulation aspects is provided in Table 1.

Table 1. List of cases computed for the U.S. Laboratory.

Case	Closeout space location	Ventilation scheme
1	Rack	Nominal
2	Aft endcone	Aft Port CD is closed
3	Rack	Aft Port CD is closed

The CFD models take into account the USL cabin aisle volume and a closeout space zone. In both cases the closeout space is a box with dimensions of $W \times H \times D = 0.53 \times 0.61 \times 0.92$ m. The boxes considered differ a little bit: the rack box is a rectangular parallelepiped, while the endcone box is slightly shaped. However, the volumes of the rack and endcone boxes are almost the same. The geometry models adopted for the present CFD analysis are shown in Figure 1, a for the rack closeout space position and in Figure 1, b for the endcone closeout space position.

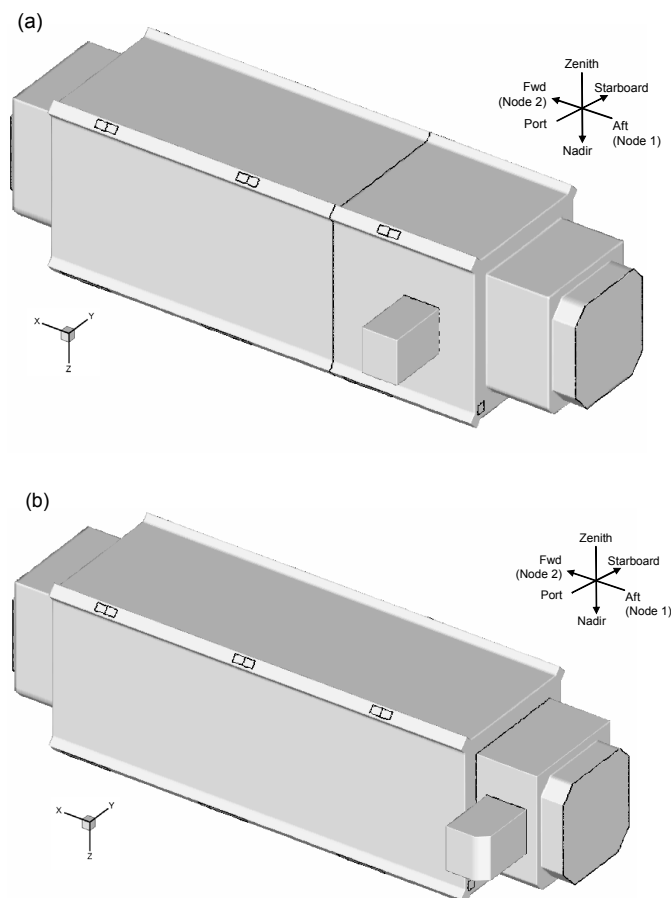


Figure 1. U.S. Laboratory geometry models with closeout space incorporated: (a) Case 1 and (b) Case 2.

The U.S. Laboratory inlet and outlet boundaries are as follows. Six USL CD's are placed in the ceiling corner pockets of the racks. Six Return Grilles (RG) are placed in the floor corner pockets of the racks. The Inter-Module Ventilation (IMV) diffuser is placed in the aft endcone. It provides air from the Node 1 supply air ducts through an IMV valve.

The steady-state air flow inside the U.S. Laboratory cabin was computed for each case setting the infinitely thin solid wall between the cabin and the closeout space. After that the box/cabin wall was changed into the interior fluid zone that simulates an instantaneous removal of a closeout space thin covering panel. The time-dependent computations for both the flow and oxygen concentration field were started from the steady-state USL flow and zero-velocity field in the box.

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. The boundary conditions are as follows. The no-slip boundary condition is imposed on all of the solid U.S. Laboratory surfaces and on the box surfaces as well. The starboard CD's of the U.S. Laboratory 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). In case of off-nominal ventilation, the port CD adjacent to the aft 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 to 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, while the aft hatch to Node 1 is open. The pressure outlet boundary condition is set at the Node 1 hatch surface. The inlet/outlet velocity boundary conditions adopted are summarized in Table 2.

Boundary and initial conditions for the oxygen transport problem are as follows. The U.S. Laboratory cabin initial oxygen volume fraction is equal to 0.21 (21 %). The oxygen volume fraction in air supplied from Node 1 through the IMV diffuser is equal to 0.21 as well. The closeout space is supposed to be filled initially by pure oxygen, so that the correspondent initial oxygen volume fraction is equal to unity (100%).

Two computational grids were generated for the U.S. Laboratory with different closeout space location. The grids are fully unstructured and consist of hexahedral

mesh elements. Both grids are clustered to the solid walls and to the diffusers. For both the grids the number of cells is about 800,000. The grids were created using the GAMBIT 2.3.16 generator [2].

Table 2. Flow rates and boundary conditions.

	Flow rate, m ³ /h	Velocity magnitude, m/s
Common Diffusers	Case 1: 93.5 (×6); Cases 2, 3: 93.5 (×5)	1.07
Aft IMV Diffuser	195	3.1
Return Grille	Case 1: -93.5 (×6) Cases 2, 3: -78.0 (×6)	-0.30 -0.25
Forward hatch boundary	Closed	Wall
Aft hatch boundary	-195	Pressure outlet BC

The unsteady Reynolds-Averaged Navier-Stokes (RANS) approach was employed. The standard k-ε model, with the standard wall functions, was used for computations [3]. The default formulation of the standard k-ε model, suggested that the model constant $C_\mu = 0.09$. In the U.S. Laboratory ventilation system it was impossible to get a proper convergence both at steady-state and unsteady computations with the default model constant. To overcome this difficulty, in the present computations the value of C_μ was increased up to 0.15, following the experience gained at our simulation of the Columbus model ventilation [4].

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).

To compute the steady-state flow inside the U.S. Laboratory cabin the governing equations for conservation of mass, momentum, and turbulence characteristics were solved using the steady pressure-

based segregated solver. The SIMPLEC pressure-velocity coupling scheme was chosen on the basis of preliminary computations. For all the cases considered, this scheme provided the best convergence. The second-order upwind spatial discretization scheme was used, both for the momentum and the k-ε model governing equations.

Time-dependent computations of both the flow and oxygen concentration field were performed with the second-order time-stepping scheme. For the first second of flow evolution process the time step was taken of about 0.01 s, and then it was gradually increased up to 5 seconds. A transient period of 600-1200 seconds (10-20 minutes) was computed for each case. Parallel computations were performed using four processes of a LINUX-based cluster.

U.S. LABORATORY CLOSEOUT SPACE PURGING PROCESS

AIR FLOW IN THE CABIN BEFORE OPENING A CLOSEOUT SPACE – The steady-state flow fields within the U.S. Laboratory cabin without any open closeout spaces were obtained for both ventilation scenarios: nominal and off-nominal ventilation. The U.S. Laboratory ventilation in regard to crew comfort was discussed previously [5]. Note that for the comfort discussion only velocity distribution in the fluid zones 0.15 m away from the cabin walls was under consideration, and the flow features in the vicinity of the walls were not analyzed. However, special attention is required to the near-wall velocity field in the case of the closeout space purging process analysis since this process is determined by the velocity level in the vicinity of the interface between the cabin and a closeout box.

For the nominal USL ventilation, Figure 2 gives a velocity distribution over a surface 0.1 m away from the cabin walls. It is seen that near-wall velocities vary in a wide range, both stagnant regions and high velocities zones are visible in the plot. Remarkably that the rack closeout space location chosen (marked in Figure 2 by a wireframe) is characterized by relatively high velocities near the box cover, while in the endcone case even for the nominal regime the near-wall air velocities are low (less than 0.1 m/s).