A Coupled 1D/3D Co-Simulation Approach in Simulating Aircraft Cabin Temperature Field

Xiyuan CHEN*, Jianzhong YANG, Tianpeng YU and Shibin YANG

Tianjin Key Laboratory of Civil Aircraft Airworthiness and Maintenance, Civil Aviation University of China, Tianjin, China

***Abstract* — This paper investigates a co-simulation approach through which one dimensional (1D) system simulation model of aircraft environmental control system (ECS) and three dimensional (3D) CFD model of cabin are coupled together, data exchange between typical 1D system simulation program AMESim and 3D Computational Fluid Dynamics(CFD) program Fluent is completed with a developed co-simulation interface. With 1D system model of ECS and 3D CFD model of cabin built in AMESim and Fluent respectively, this paper investigates the transient temperature distribution in aircraft cabin to demonstrate the effectiveness of 1D/3D co-simulation approach, a comparison of temperature distribution calculation result between 1D/3D co-simulation and standalone 1D system simulation is fulfilled, the result indicates an obvious temperature stratification in cabin and the co-simulation method can provide more detailed information than standalone 1D system simulation. Great benefit of 1D/3D co-simulation is proved to make 1D system simulation and 3D CFD become complementary.**

Keywords - 1D/3D co-simulation; system simulation; CFD; cabin temperature; transient simulation

I. INTRODUCTION

As the popularity of traveling by aircraft increasing rapidly in past decades, friendly cabin environment air quality has become a significant factor in aircraft design. Aircraft Environmental Control System(ECS) is designed to provide ideal gas condition to cabin passengers with satisfactory airflow distribution.

1D ECS system simulation modeling and 3D Computational Fluid Dynamics (CFD) can play important roles in simulating ECS component performance and the air distribution in aircraft cabin. The complex system modeling of ECS can be implemented in 1D system simulation program, the work of [1] and [2] build the 1D mathematical system model and model library of ECS components to analysis dynamic process in short computing time, the performance of key components of ECS can be calculated in few minutes with a minimal computing costs in work [3]. However, 1D system simulation of ECS will use several assumptions to improve the computation speed, work [4] indicates that a significant discrepancy between the computed and measured airflow is caused by difficulties in obtaining accurate flow boundary conditions from the ECS. Furthermore, the temperature variation in cabin is considered as an inertial element of the feedback circuit in system simulation model of ECS temperature control subsystem by work [5], an uniform temperature distribution assumption in cabin is used and will compromise the accuracy of calculation. Moreover, system simulation cannot predict the detailed information of air distribution in cabin.

Compared to the system simulation approach, 3D CFD methods can predict detailed air distribution such as temperature distribution in cabin accurately. Based on the finite volume method, flow field is divided to a number of grid cells, and several control equations can be solved in CFD programs to obtain a high spatial resolution value of cabin air distribution. With the development of meshing technology in work [6] and turbulent model of enclosed environment in work [7], a significant effort has been made in CFD to simulate the non-uniform aircraft cabin environment distribution, work [8] investigates the transient thermal field in aircraft with standalone CFD method. In work of [9-10] use 3D CFD model of cabin to investigate airborne contaminant transportation in cabin proving the non-linear dispersion of concentration field. In order to validate the accuracy of simulating cabin air distribution with CFD model, Comparison between results of CFD simulating and experimental data is completed by work [11], showing a great capability of CFD in predicting the transient non-uniform air distribution in aircraft cabin. However, CFD numerical simulation in literature treat the cabin in isolation and make assumptions about the inlet air from the ECS in work [12]. The assumed inlet boundary condition of cabin mock-up may not reflect what the ECS is able to deliver. Furthermore, CFD approach spend much more computing time than the system simulation, a simulation of cabin airflow under transient condition will take several days.

Therefore, coupling system simulation of ECS with CFD simulation of cabin air distribution is very attractive considering their complementary advantages and is to be implemented in present investigation of work [13]. Previous research in building industry in work [14-15] have proven that co-simulation of 1D system simulation and 3D CFD simulation can bridge the discontinuities between them and can provide more practical calculation.

This paper focus on 1D/3D transient co-simulation of temperature distribution in 1D ECS and 3D cabin. The 1D system simulation model of ECS and the 3D CFD model of cabin mock-up are built up in typical commercial simulation software AMESim and Fluent, respectively. A system simulation-CFD co-simulation interface is developed and the data exchange between two programs at every iteration is accomplished, the output parameters of the AMESim system simulation model of ECS is coupled as the boundary inlet condition of 3D cabin mock-up. In order to demonstrate the effectiveness of co-simulation in temperature simulating, the heating and cooling condition is calculated based on the implementation of co-simulation approach.

II. MODELING OF THE 1D ECS

A. *Working Principle of ECS*

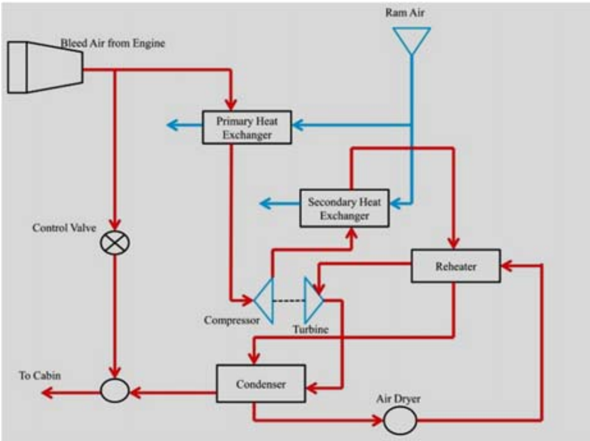


Figure 1 Working principle of ECS

As is illustrated in figure1, bleed air extracted from aircraft engine compressor with high temperature and pressure is delivered to the ECS. The ECS includes two heat exchangers cooled by ram air from outside, an Air Cycle Machine(ACM) consists of a compressor and a turbine, a condenser, a reheater and an air dryer. The two heat exchangers are used to cool the hot bleed air from engine, a fan will drive the ram air flow to heat exchanger as the refrigerant. The cooled air from heat exchangers enters ACM, in which the turbine expands the air and cools it even further. Additionally, the output work of turbine is transmitted to coaxial compressor which increase the air pressure before it enters the secondary heat exchangers. In order to reaching and getting out of saturation condition of water vapor, the reheater and condenser will warms up the cold air from secondary heat exchanger and cools the hot charged air, respectively. The Air Dryer removes part of the water vapor in order to decrease the air humidity. A control valve is used to control the output temperature of ECS by changing its open ratio through which air from ECS and bleed air will be mixed at a specific proportion. The bleed air will become satisfactory fresh supply air to aircraft cabin with comfortable temperature and humidity.

B. *System Simulation of 1D ECS*

By providing abundant ready-made component library, AMESim has been widely used in aerospace area to simulate system performance in terms of ECS, hydraulic, fuel control, de-icing, flight control and so on.

One dimensional model of ECS provides an advantage over three dimensional CFD by both reducing the computing time the model complexity. In this investigation, the 1D ECS system simulation model is built up in AMESim program using standard components and customizable components given by thermal library and signal control library in AMESim. This allows the designer to quickly know about working condition of system and evaluate performance of surrounding flow circuit in view of the performance and design of an individual component.

III. MODELLING OF 3D CABIN MODEL

Three dimensional CFD model is commonly used to calculate airflow distribution in enclosure space such as aircraft cabin by solving Navier-Stokes (N-S) equations. With the Reynolds Averaged Navier-Stokes (RANS) turbulence model used, N-S governing equations of mass, momentum, energy, and species conservation can be solved in finite volume. The conservative governing equations can be written in an general form:

$$\frac{\partial(\rho\Phi)}{\partial t}+div\left(\rho u\Phi\right)=div(\Gamma\,\mathrm{grad}\,\Phi)+\mathrm{S} \qquad (1)$$

In equation(1), ρ stands for flow density, t stands for time, u is the flow velocity, Γ is the diffusion coefficient. Φ is a general variable, the equation(1) becomes mass conservation equation when Φ represents pressure, momentum conservation equation when Φ represents velocity, energy conservation equation when Φ represents temperature. S stands for the source term.

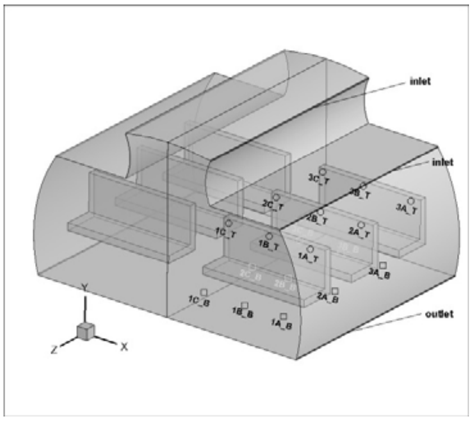


Figure 2. Three dimensional cabin geometry mock-up

In this investigation, a three dimensional non-occupied cabin mock-up was built up with a 4m×2m×3m geometry size and three rows of seats are placed in cabin. As is shown in figure2, four air inlets are symmetrically located at ceiling and side wall area, two outlets of cabin mock-up is located at the bottom area of cabin sidewall symmetrically. In order to monitor the airflow distribution and temperature stratification inside the cabin, eighteen monitoring points are placed in one side of cabin with six monitoring points located in each row, two different monitoring points are set within individual seat area which stands for the head zone of seated passenger and leg zone of seated passenger, respectively. In this paper, the temperature of four inlets' boundary condition is delivered by 1D AMESim model of ECS system with dynamic data exchanged at every time step of CFD.

IV. THE CO-SIMULATION STRATEGY FOR AMESIM-FLUENT COUPLING

The realization of the data exchange between two different programs is the most critical technology for AMESim-Fluent co-simulation. Input and Output(IO) Parameters of AMESim and Fluent solver is predefined and exchanged dynamically and synchronously during calculation. At each time step of co-simulation running, an extended User Defined Function(UDF) of Fluent will call the dynamic link library(DLL) of the compiled AMESim model which covers information of IO parameters and solver, with the DLL uploaded dynamically in memory, AMESim solver will be called by Fluent program and the coupling of AMESim-Fluent is realized, as is shown in figure3.

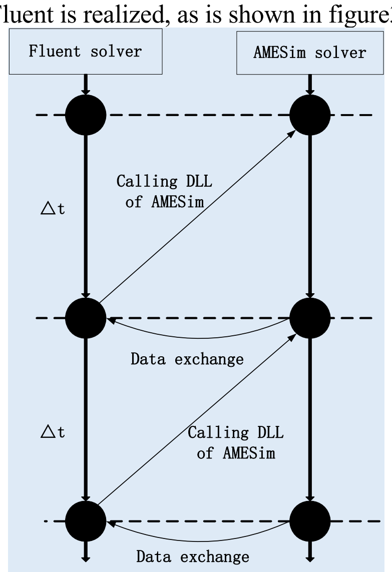


Figure 3. Dynamic data exchange between AMESim solver and Fluent solver

In AMESim program, a series of export functions are developed which can be classified by model initialization, IO parameter assignment and calculation solving, and AMESim 1D system model is encapsulated in a DLL together with the export functions. In this investigation, Application Programming Interface(API) in Windows system is used to linking and loading DLL generated by AMESim, initialization, IO parameter assignment and calculation solving can be implemented separately from AMESim software environment with C language script containing Windows API.

In order to realize the DLL invoking of AMESim system model, the windows API function Loadlibrary() is employed to load DLL and map its memory address, the Loadlibrary() function returns zero when it meets with a mapping failure and returns the handle of library modules after successful mapping. After the DLL is successfully loaded, GetProcAddress() function of API is used to searching memory address of export function encapsulated in DLL, which is essential to operating the AMESim system model out of the AMESim software environment, as is shown in figure4.

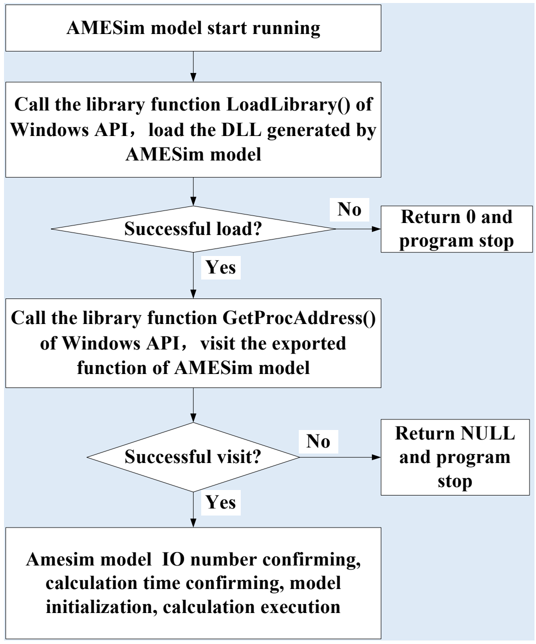


Figure4 Using Windows API C language script to operate AMESim outside

In Fluent program, the User Defined Function(UDF) of Fluent has been widely used to custom extending the conventional functions of Fluent, such as customization of boundary condition, material properties, source term of transport equations and user defined scalar. A variety of specific macro supported by UDF is used to fulfill the communication between AMESim and Fluent, with the C language script invoking DLL of AMESim model included in UDF, the operating of AMESim system model and data exchange between programs can be accomplished in Fluent. In co-simulation implementation, Fluent is set as the master program and AMESim is set as the slave program.

As is shown in Figure5, there are three macros used in co-simulation named by DEFINE_ON_DEMAND, DEFINE_ON_PROFILE and DEFINE_EXECUTE_AT_END. The entire process of co-simulation is completed in Fluent program, after the UDF script is compiled, source code included in DEFINE_ON_DEMAND macro will be executed firstly, initialization and IO parameters confirmation of AMESim system model will be done in this section. Secondly, source code included in DEFINE_ON_PROFILE will be performed in which the boundary condition of CFD model in Fluent will be updated. The DEFINE_EXECUTE_AT_END is employed to save output value of AMESim model at the end of each time step, the saved value will take place of the boundary condition of Fluent model before a new step begin to calculate.

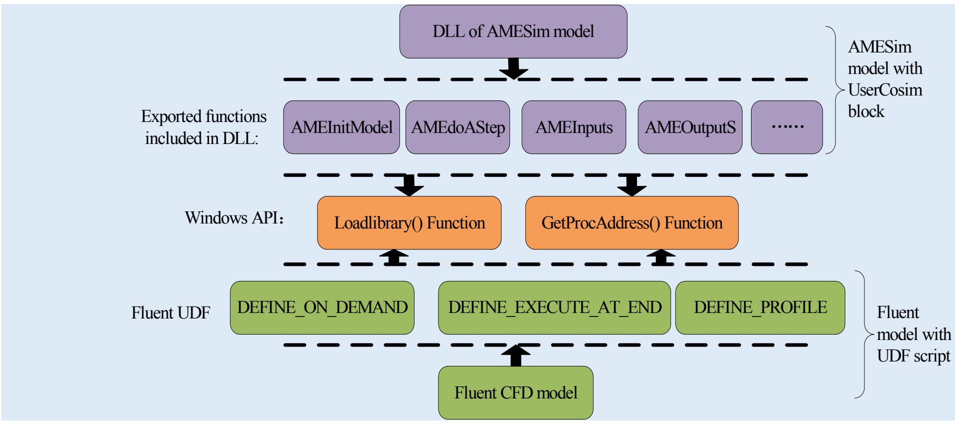


Figure5 Model communication between Fluent and AMESim using UDF script

V. CASE STUDY

In order to demonstrate AMESim-Fluent co-simulation applied in aircraft cabin temperature field simulation, this study used a completed one dimensional ECS model built in AMESim and three dimensional CFD model of cabin mock-up built in Fluent. The output parameter of AMESim model is coupled as the boundary condition of CFD model in Fluent, data exchange between two program solvers will be executed at every time step, the single time step calculation result of AMESim model will be delivered to CFD model, the boundary condition of CFD model will be updated by AMESim single step output value, during this meantime, the AMESim solver will stop working until the next single step of CFD model is completed, the data exchange process above will be repeated until the whole calculation is accomplished.

Aiming to satisfy the requirement of thermal comfort of cabin passengers, the aircraft cabin needs to be cooled and heated in hot weather and cold weather, therefore, this paper uses heating and cooling conditions to demonstrate the co-simulation approach. As in shown in figure6, in cooling case simulation, initial temperature of ECS output is set as 303K and 298K for target temperature of ECS output. For heating case simulation, 290K is set as the initial temperature of cabin and 295K for target temperature, respectively. The output temperature of ECS is shown in figure 6, the target temperature of ECS is reached after a period of time. According to Federal Aviation Administration(FAA) airworthiness regulation 25.831, inlet velocity of cabin CFD model is set as 2m/s. Time step for both models is set as 0.1 second to guarantee synchronicity of co-simulation running, whole calculation time for two cases is set as 300s.

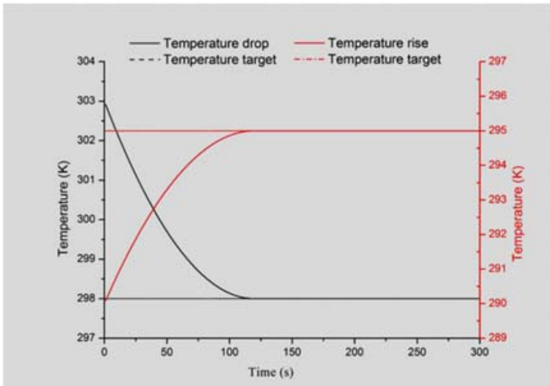


Figure 6. Target temperature of ECS model for cooling case and heating case

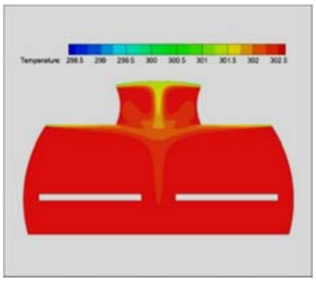
VI. RESULT

A. *Cooling Case Simulation Result*

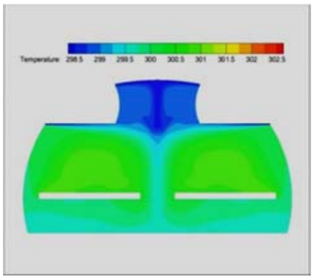
The temperature distribution variation of cabin over time is illustrated in figure7, owing to the cooled fresh air supplied to cabin, the entirety temperature of cabin mock-up becomes lower along with as the output temperature of ECS cooled down, showing the refrigeration effectiveness of ECS. However, the temperature distribution in cabin presents a distinct stratification because of the airflow turbulence in cabin. Due to the influence of airflow turbulence, two eddies form in the seats area of cabin because of the airflow collision from two inlet airs, the cooled air entered cabin sinks quickly due to the thermal buoyant effect and reach the aisle area and zone under seats, with the temperature of passengers and windows-side area staying higher relatively.

DOI 10.5013/ IJSSST.a.17.40.11

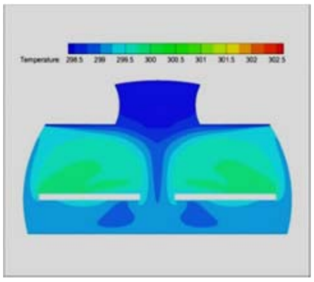
ISSN: 1473-804x online, 1473-8031 print



t=30s



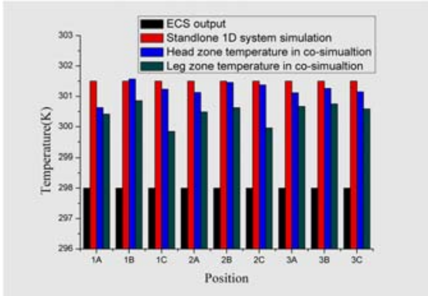
t=150s



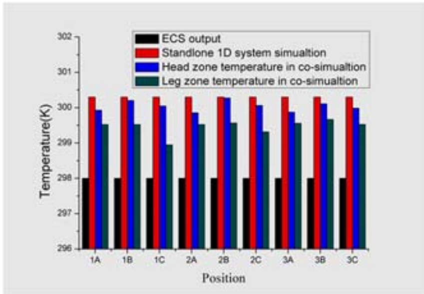
t=210s

Figure7 Temperature distribution of cabin at different time in cooling simulation case

Figure8 presents the temperature parameter comparison between AMESim ECS output, standalone 1D system simulation, passengers head area and leg area in cabin with co-simulation approach when t=120s and t=180s. An obvious hysteresis between cabin temperature and AMESim output temperature is found which is consistent with research treating cabin as an inertial element of ECS temperature control circuit. In addition, distinct non-uniform temperature distribution is revealed in each row of cabin that refrigeration effect of middle column is the worst in three rows, column near window secondly and column near aisle is the best. Furthermore, calculation result of standalone 1D system simulation can almost reflect the temperature of passengers' head area in cabin seats, the maximum difference value dose not exceed 1K. However, an obvious temperature calculation result discrepancy occurs in passengers' leg area of seats near aisle between the standalone 1D system simulation and co-simulation, the maximum difference value is 1.8K in 1C_B monitor position, which can prove that standalone 1D system simulation cannot predict detailed temperature distribution in cabin. Above all, the cooling case simulation demonstrates the application and effectiveness of co-simulation method on aircraft cabin temperature field calculation.



t=120s



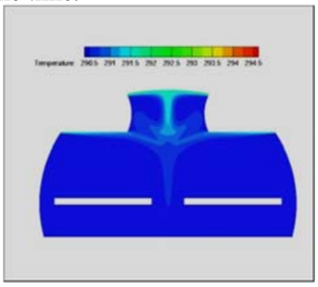
t=180s

Figure 8 Comparison between ECS output temperature, cabin temperature with standalone 1D system simulation and cabin passengers surrounding temperature at different time in cooling simulation case when t=120s and t=180s

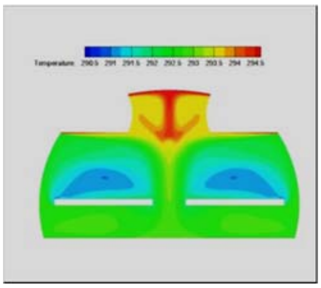
B. *Heating Case Simulation Result*

A significant stratification of temperature distribution in cabin mock-up is presented in heating case simulation similarly as is proven in cooling case. Figure10 shows the temperature distribution in cabin at different time. Due to the thermal buoyant effect, heated inlet air fill the ceiling area of cabin firstly and then it comes down to the seat area with two symmetrical eddies formed. The temperature of passengers head area rises slower than the leg area, in terms of the leg area, heating efficiency of column near aisle is the best, and middle column is the second, column near window is the worst.

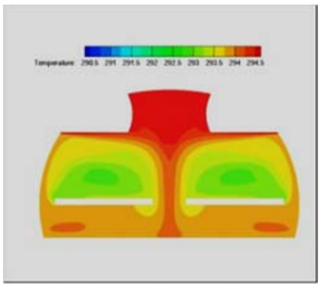
A comparison between temperature at different columns in same row is presented in figure11, An obvious hysteresis between cabin temperature and AMESim output temperature is also found in heating case simulation. Heating efficiency of leg area is relatively better than head area in three rows, in terms of head area, temperature of seat near window rises most quickly in same row and temperature of seat near aisle rises most quickly referring to leg area. As same to the cooling case, the calculation result of 1D system simulation can almost represent the temperature in passengers' head area with maximum difference value limited in 0.5K, however, standalone 1D system simulation is not capable enough to predict the temperature in passengers' leg zone of cabin, maximum difference value reaches 1.5K at 2C_B point when t=120s. Both of cooling case and heating case shows the advantage of co-simulation over standalone 1D system simulation by providing the non-assumption ECS boundary and detailed transient temperature distribution in cabin at the same time.



t=30s

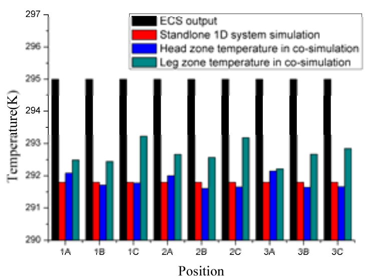


t=120s

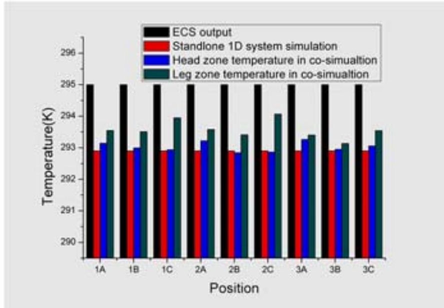


t=210s

Figure9 Temperature distribution of cabin at different time in heating simulation case



t=120s



t=180s

Figure 10 Comparison between ECS output temperature, cabin temperature with standalone 1D system simulation and cabin passengers surrounding temperature at different time in heating simulation case when t=120s and t=180s

VII. CONCLUSION

This investigation applied a AMESim/Fluent co-simulation approach to calculate the temperature distribution in ECS and cabin dynamically. It was proven that co-simulation can bridge the discontinuities between one dimensional system simulation and three dimensional CFD. With their complementary advantages combined, CFD model of cabin can calculate the temperature stratification in cabin with non-assumption boundary condition from ECS model in AMESim within an acceptable computing time. Dynamic data exchange between AMESim/Fluent is completed in this paper to do transient calculate, a cooling case and a heating case are calculated to demonstrate the established potential of co-simulation method in aircraft ECS design.

ACKNOWLEDGMENTS

This paper is supported by Fundamental Research Funds for the Central Universities-Civil Aviation University of China Subject: Research on 1D-3D co-simulation technology of aircraft cabin air environment, the project number is 3122015D021.

REFERENCES

[1] He Jun, Zhao Jingquan. Dynamic simulation of the aircraft environmental control system, Chinese Journal of Aeronautics, vol.14,no.3, pp.129-133, 2001

[2] Song Junxiao, Yuan Xiugan. MATLAB simulation model of environmental control system, Journal of Beijing University of Aeronautics and Astronautics, vol.28, no.1, pp43-46, 2002

[3] Pérez-Grande Isabel, Leo Teresa J. Optimization of a commercial aircraft environmental control system, Applied Thermal Engineering, vol.22, no.17, pp1885-1904, 2002

[4] Zhang T, Chen Q and Lin C.H, Optimal sensor placement for airborne contaminant detection in an aircraft cabin, Building and Environment, vol. 42, no.4, pp1675-1684, 2007

[5] Romani Rubens, de Góes, Luiz Carlos. Cabin temperature control model for commercial aircraft, United states: AIAA Modeling and Simulation Technologies Conference 2012,2012

[6] Duan R, Liu W, Xu L, Huang Y, Shen X, Lin C H, Liu J, Chen Q and Sasanapuri. Mesh type and number for CFD simulations of air distribution in an aircraft cabin, Numerical Heat Transfer, Part B: Fundamentals, vol.67,no.6): pp489-506,2015

[7] Wei Liu, Jizhou Wen, Chao-Hsin Lin, Junjie Liu, Zhengwei Long, Qingyan Chen. Evaluation of various categories of turbulence models for predicting air distribution in an airliner cabin, Building and Environment, vol.65,no.1, pp118-131, 2013

[8] Vincenzo Bianco, Oronzio Manca, Sergio Nardini, Mario Roma. Numerical investigation of transient thermal and fluidynamic fields in an executive aircraft cabin, Applied Thermal Engineering, vol.29, no.16, pp3418-3425, 2009

[9] Isukapalli Sastry S. Computational fluid dynamics modeling of transport and deposition of pesticides in an aircraft cabin, Atmospheric Environment, vol.68, no.1: pp198-207, 2013

[10] Li Fei, Junjie Liu, Jianlin Ren, Xiaodong Cao, Yifang Zhu. Numerical investigation of airborne contaminant transport under different vortex structures in the aircraft cabin, International Journal of Heat and Mass Transfer, vol.96,no.1, pp287-295, 2016

[11] ChangWei Yang, XiWen Zhang, ZhaoHui Yao, Xiaodong Cao, Junjie Liu, Feng He. Numerical study of the instantaneous flow fields by large eddy simulation and stability analysis in a single aisle cabin model, Building and Environment, vol.96,no.1, pp 1-11, 2016

[12] Kelsall David, Ludhi, Abdul, Bayrasy Pascal, Wolf Klaus. Achieving an improved understanding of the factors affecting an aircraft environmental control system by coupling a 1D cabin air distribution system model with a 3D passenger cabin model using Co-simulation middleware, United states:SAE Technical Papers, 2009

[13] Zhai Z, Chen Q Y. On approaches to couple energy simulation and computational fluid dynamics programs, Building and Environment, vol.37,no.1,pp857-864, 2002

[14] M Goldsworthy. Dynamic coupling of the transient system simulation and fire dynamics simulation programs, Journal of Building Performance Simulation, vol.5,no.2, pp105-114, 2011

[15] Wang Liangzhu, Chen Qingyan. Applications of a Coupled Multizone-CFD Model to Calculate Airflow and Contaminant Dispersion in Built Environments for Emergency Management, HVAC&R RESEARCH, vol.14,no.6, pp925-939, 2008

[16] Liu, Y., Yang, J., Meng, Q., Lv, Z., Song, Z., & Gao, Z. (2016). Stereoscopic image quality assessment method based on binocular combination saliency model. Signal Processing, 125, 237-248.

[17] Yang, J., Xu, R., Lv, Z., & Song, H. (2016). Analysis of Camera Arrays Applicable to the Internet of Things. Sensors, 16(3), 421.

[18] Lv, Z., Penades, V., Blasco, S., Chirivella, J., & Gagliardo, P. (2016). Evaluation of Kinect2 based balance measurement. Neurocomputing.

.