

## ANALYSIS OF FLOW STRUCTURE IN A HELICOPTER CABIN TO IMPROVE THE THERMAL COMFORT USING CFD MODELING

Doruk ŞAHİN<sup>1</sup> and Mehmet Metin YAVUZ<sup>2</sup>  
Middle East Technical University  
Ankara, Turkey

### ABSTRACT

*Proper distribution of conditioned air inside the cabins of passenger transportation type aircrafts is crucial in terms of passengers' comfort. In the present study, computational fluid dynamics (CFD) models are developed using ANSYS FLUENT to investigate the thermal comfort levels of the passengers in a newly designed passenger transportation helicopter. The results reveal that the performance of the existing personalized air distribution system of the helicopter is inadequate to provide a comfortable environment to the passengers. For the purpose of enhancing the thermal comfort levels of the passengers, an iterative procedure is followed to redesign the personalized air distribution system. The new design both provides the desired air velocities and effective distribution around the faces of passengers along with more uniform temperature distribution inside the cabin.*

### INTRODUCTION

Recently, more and more people are travelling via air transportation. As aviation industry develops rapidly, interest of the flying public to the environmental conditions in aircraft cabins is increasing [Du, 2017]. To make the passengers feeling satisfied, it is inevitable to achieve a thermally comfortable cabin environment. On the other hand, for the design purposes, investigating flow field inside an aircraft cabin is rather challenging because of the high turbulence intensity, unstable flow, complex geometry, high occupant density and high thermal loads.

Several experimental and numerical studies have been performed to investigate the air distribution within the aircraft cabins. Yan et al. [Yan, 2009] investigated the flow field within a Boeing 767-300 cabin mock-up both numerically with standard k- $\epsilon$  model and experimentally. They found that the complex domains of the aircraft cabins make flow field highly 3-D. Kühn et al. [Kühn, 2009] performed PIV and temperature field measurements in an A380 cabin mock-up for isothermal and cooling conditions by varying air inlet configurations and flow rates. The results showed that locations of air inlets and flow rate ratio between air inlets (at constant total air exchange rate) highly affect flow field in an aircraft cabin. Li et al. [Li, 2015] conducted experimental measurements in the cabin of an MD-82 airplane in “two of five gaspers

---

<sup>1</sup> M.Sc. in Mechanical Engineering Department, Email: e167787@metu.edu.tr

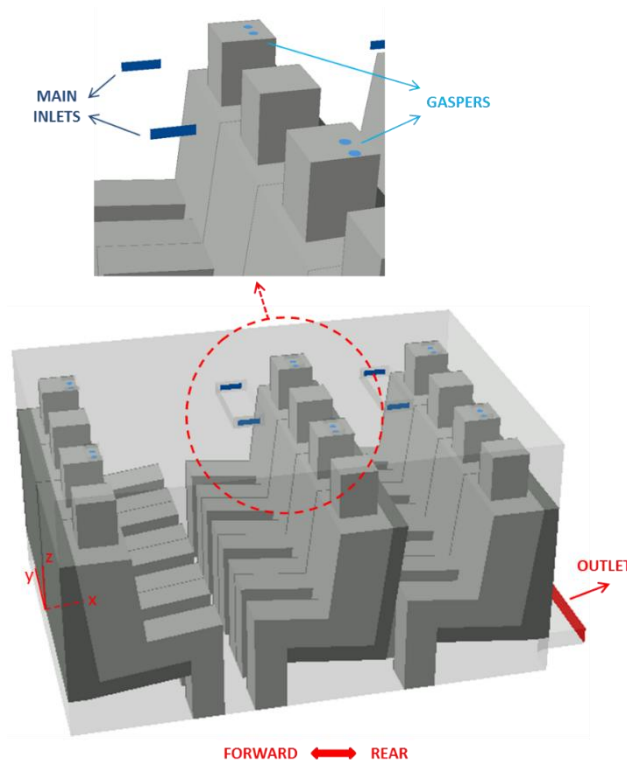
<sup>2</sup> Assoc. Prof. in Mechanical Engineering Department, Email: ymetin@metu.edu.tr

(personalized air inlets) are on” condition and compared the obtained data with the Liu et al.’s [Liu, 2013] study which is performed for “all gaspers are off” condition. The results revealed that personalized air inlets have much larger effect on air motion than main air inlets because of having higher momentum flux. Moreover, the results showed that personalized air inlets increase the air movement within the cabin and make air distribution more uniform. Zhang et al. [Zhang, 2009] investigated airflow and temperature distribution in a half-occupied cabin by both numerical simulations with RNG  $k-\varepsilon$  model and experimental measurements. By comparing with Baker’s [Baker, 2005] study which is conducted for isothermal cabin conditions, they found that thermal plumes from passengers make flow field in occupied cabins stable; therefore, time averaging method is appropriate to use in occupied cabin flow simulations.

In the current study, flow and temperature fields in a newly designed twelve-passenger capacity passenger transportation helicopter cabin are numerically investigated and an air distribution system design enhancing the thermal comfort levels of the passengers is proposed. For this purpose, computation fluid dynamics (CFD) models of the helicopter cabin are developed.

### DESCRIPTION OF THE INVESTIGATED HELICOPTER

The investigated helicopter has a capacity of twelve passengers. Cooled air is supplied to the cabin by four main inlets and twelve personalized inlets called gaspers. Main inlets are non-adjustable type and their objective is to satisfy thermal comfort requirements related to the general cabin environment. On the other hand, gaspers are adjustable type of air inlets which can be opened, closed and adjusted between two extremes by the passengers. The simplified model of the helicopter cabin is shown in Figure 1.

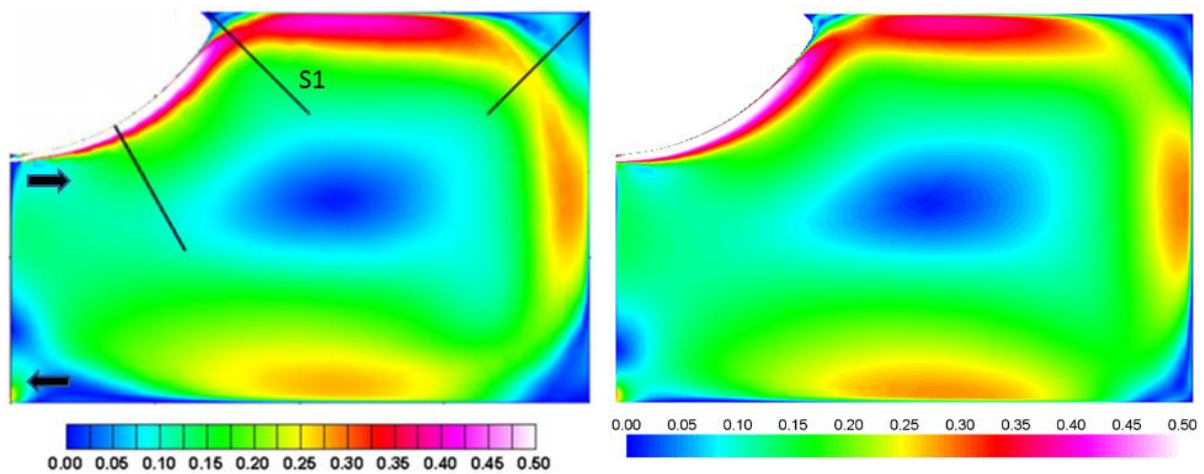


**Figure 1. Simplified model of the investigated helicopter cabin**

## COMPUTATIONAL ANALYSIS

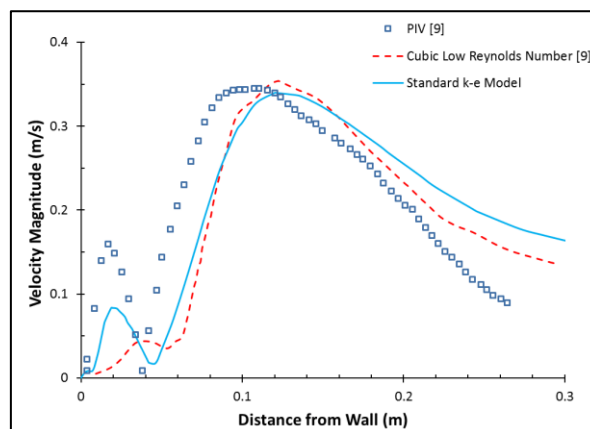
For the helicopter cabin flow and temperature field investigation, Reynolds-Averaged Navier-Stokes (RANS) simulations are conducted. To include the effect of the buoyancy, Boussinesq approximation is used. Reynolds stress term of the RANS equation are modeled by using Boussinesq hypothesis. Standard  $k-\varepsilon$  turbulence model proposed by Launder and Spalding [Launder and Spalding, 1972] is used since it is robust, computationally economic and reasonably accurate for a wide range of industrial flows. Moreover, well convergence behavior of standard  $k-\varepsilon$  model is considered since the study comprises an iterative design procedure wherein boundary conditions; therefore, flow structures are variable from case to case and flow fields are rather complex because of the existence of many air inlets and high occupant density.

The performance of standard  $k-\varepsilon$  model for cabin-like internal flows is evaluated based on Günther et al.'s [Günther, 2006] study which includes both experimental data and numerical results of various turbulence models. Velocity contour comparison in symmetry plane of the test case and velocity profile comparison on a line located at the first separation region (S1) of the test case are shown in Figure 2.



(a) Cubic low Reynolds number  $k-\varepsilon$  model of Günther et al.'s [Günther, 2006] study

(b) Standard  $k-\varepsilon$  model of the present study



(c) Velocity magnitude comparison on first separation location

**Figure 2. Validation study results**

As the near wall treatment, “enhanced wall treatment” is selected to resolve up to all viscous-affected regions since various types of fluid flow phenomena exist like separation, recirculation, vortices, attachment to or detachment from physical boundary, etc. because of the complex and highly dense domain of the cabin and should be correctly predicted by CFD model.

Simulations are performed as steady-state. The pressure-based coupled algorithm is employed to couple the pressure and velocity. First-order scheme is used for pressure discretization and second-order upwind scheme is used for discretizing all other variables. Such discretization strategy has been proven to be effective by many previous studies [You, 2016], [Zhang, 2007], [Wang and Chen, 2009], [Liu, 2013].

### DEVELOPMENT OF CFD MODEL

To investigate, the thermal comfort conditions of the passengers, CFD models are developed to simulate cabin flow and temperature distribution for fully occupied conditions. The cabin model generated to be used in the simulations is half of the real cabin model since the helicopter cabin is symmetrical to the longitudinal plane passing through the middle of the cabin. Simplified passenger dummies are modeled to investigate the thermal comfort conditions of the passengers which also dissipate heat into the cabin.

Boundary conditions applied in the simulations are tabulated in Table 1. It should be noted that, outside air temperature (OAT) is defined as 44°C, which is the hottest extreme environment temperature within the operational envelope of the investigated helicopter.

**Table 1. Boundary conditions applied in helicopter cabin simulations**

<i>Outside</i>	Temperature (°K)	317.2 (44°C)
	Convective Heat Transfer Coefficient (W/m <sup>2</sup> -K)	50.6
<i>Main Air Inlet</i>	Velocity (m/s)	Variable according to different cases
	Turbulence Intensity (%)	5
	Hydraulic Diameter (mm)	48.05
	Temperature (°K)	285 (11.8°C)
<i>Personalized Air Inlet</i>	Velocity (m/s)	Variable according to different cases
	Turbulence Intensity (%)	5
	Hydraulic Diameter (mm)	Variable according to different cases
	Temperature (°K)	285 (11.8°C)
<i>Outlet</i>	Pressure (Pa)	P <sub>static_gage</sub> =0
<i>Wall</i>	Velocity (m/s)	u <sub>wall</sub> =0
	Material	Aluminum
	Thickness (mm)	1
	Symmetry	1 longitudinal plane
	Adiabatic	Between cabin and cockpit
<i>Passenger</i>	Heat Generation Rate (W)	75

### Mesh Independence Study

A mesh independence study is conducted on helicopter cabin model for five mesh sizes. Velocity magnitude distribution around the faces of passengers and temperature variation on vertical lines are compared for the mesh independence study. The mesh counts of the five mesh size alternatives are tabulated in Table 2.

Table 2. Mesh counts of the five mesh size alternatives

	Mesh 1	Mesh 2	Mesh 3	Mesh 4	Mesh 5
Number of Elements	985000	2066000	3040000	4488000	7359000

To illustrate, mesh independence study results for one passenger (P1) are shown in Figure 3. According to mesh independence study results, 4.5 M volume mesh size is selected for the further computational analysis.

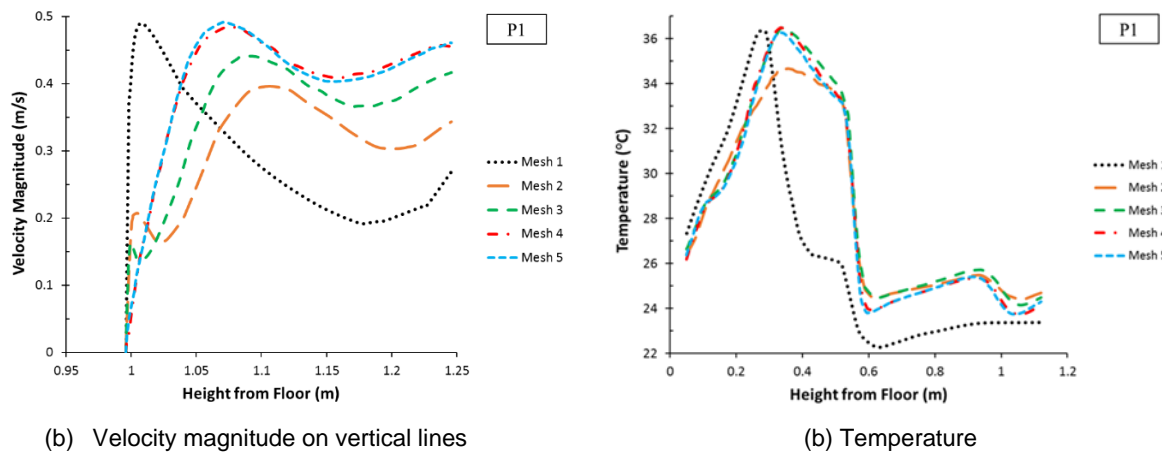


Figure 3. Mesh independence study results

## THERMAL COMFORT INVESTIGATION

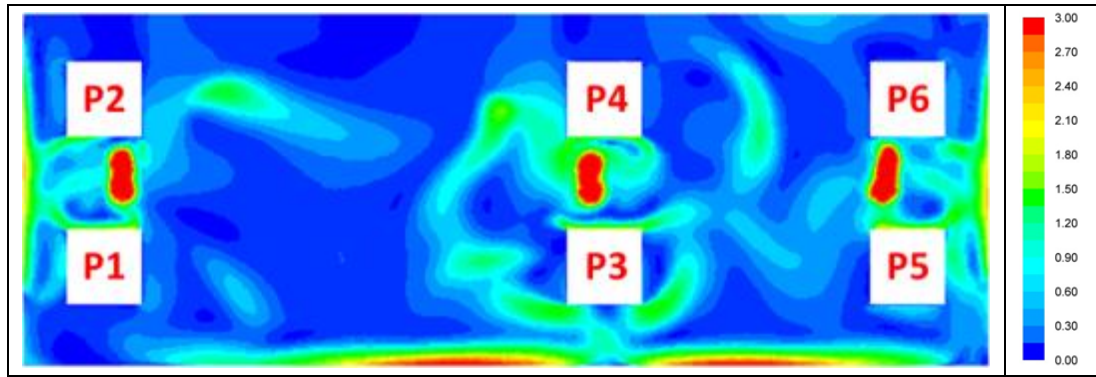
### Thermal Comfort Requirements

According to the information obtained from the literature [SAE ARP292, 2003], [ASHRAE 161-2013, 2013], [ASHRAE 161-2007, 2007], it is considered that the air velocity around the faces of passengers should be between 1 m/s and 3 m/s to make passengers feeling comfortable without exposing to drought or stagnation. In terms of temperature distribution, [SAE ARP292, 2003] states, “The variation in temperature should not exceed 5°F (2.8°C) measured in a vertical plane from 2 in (5 cm) above floor level to seated head height.”

In thermal comfort investigation, since the air movement has as a direct effect on the feeling of passengers, air flow in the vicinity of the faces of passengers is determined as the primary concern. As the secondary concern, temperature distribution throughout the cabin domain is investigated.

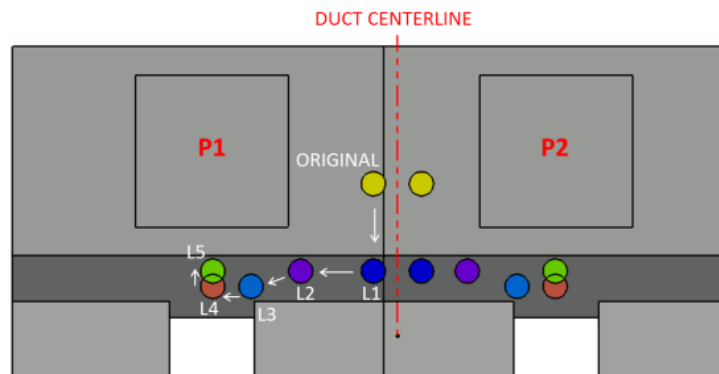
### Thermal Comfort Improvement

Thermal comfort levels of the passengers are investigated for the existing air supply system design and the results revealed the inadequacy of the existing system in creating comfortable environment to the passengers. Air movement around the faces of the passenger are shown in Figure 4.



**Figure 4. Velocity contours on the horizontal plane passing through the middle of the faces of passengers**

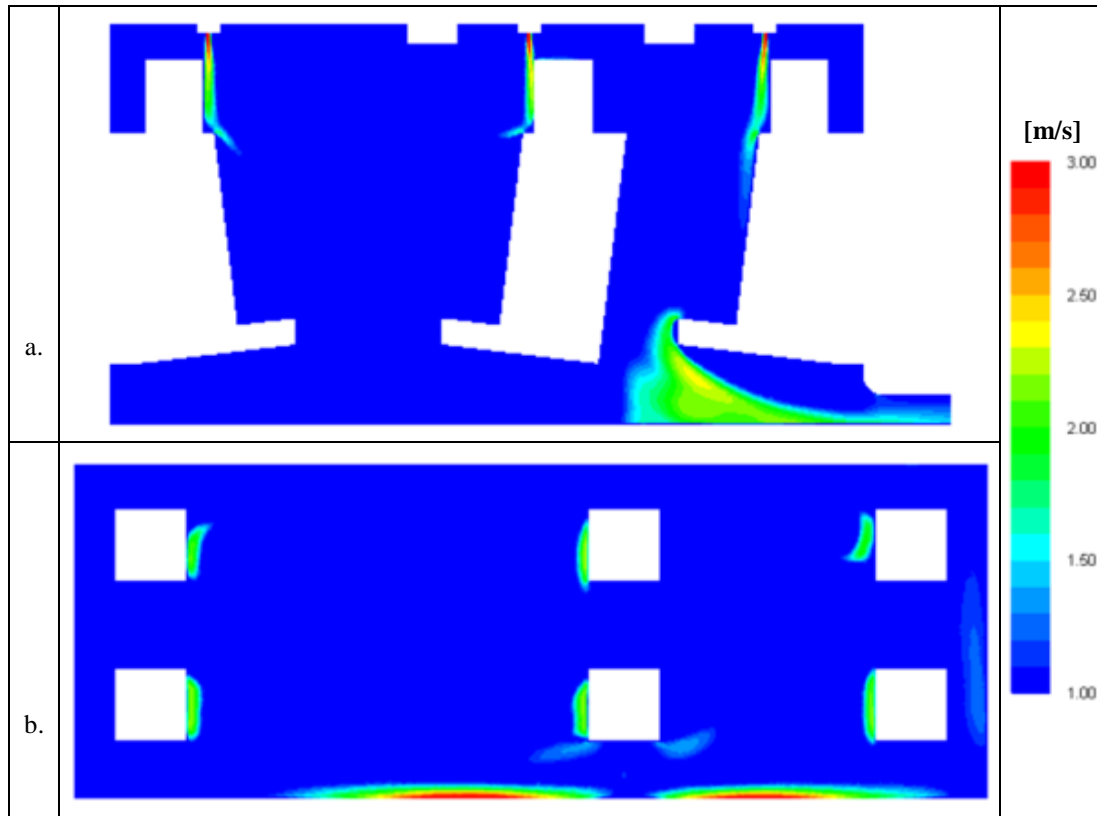
By evaluating air flow and temperature fields of the cabin with the existing air supply system, it is concluded that a design improvement is required to improve the thermal comfort conditions of the passengers. For this purpose, an iterative design procedure is followed. As the iterative design procedure, for each gasper design case, a numerical simulation is performed to investigate the passengers' comfort conditions. The results of the simulations are evaluated by focusing on the thermal comfort related parameters and according to that, the design is improved by modifying locations, shapes and flow areas of the gaspers. During the iterations, besides focusing gasper design, alternative flow rate ratio (total main inlet flow rate / total gasper flow rate) options different from the existing design are also investigated when considered as required. When determining the alternative cases, the limitations existing on account of environmental control system (ECS) performance requirements and spatial limitations of the ducting system design are considered. By processing successive designs and simulations, achieving an improvement in the thermal comfort conditions of the passengers is aimed. To illustrate, the first followed design path, comprising gasper location modification, are figured out in Figure 5 for two passengers (P1 and P2).



**Figure 5. First followed design path (modifying gasper locations)**

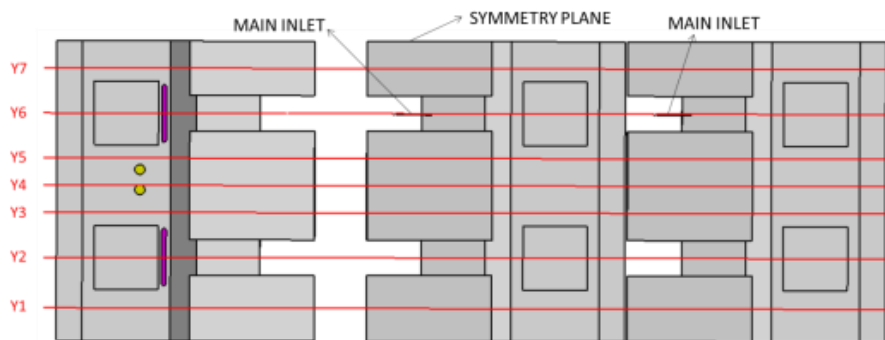
By the iterative study, a new design providing the desired air velocities around the faces of the passengers and showing up improvement in terms of temperature distribution is developed.

Considering air flow requirements, the velocity distribution around the faces of the passenger created by the new design is shown in Figure 6 on the vertical plane passing through the middle of the faces of the passengers sitting at the interior seats and on the horizontal plane passing through the middle of the passengers' head level. It can be observed that velocity magnitudes around the faces of the passengers are at the desired levels according to the comfort requirements. Moreover, simulation results show that by the new design, gasper air flow covers sufficiently broad area around the faces of the passengers owing to its elongated hole shape.



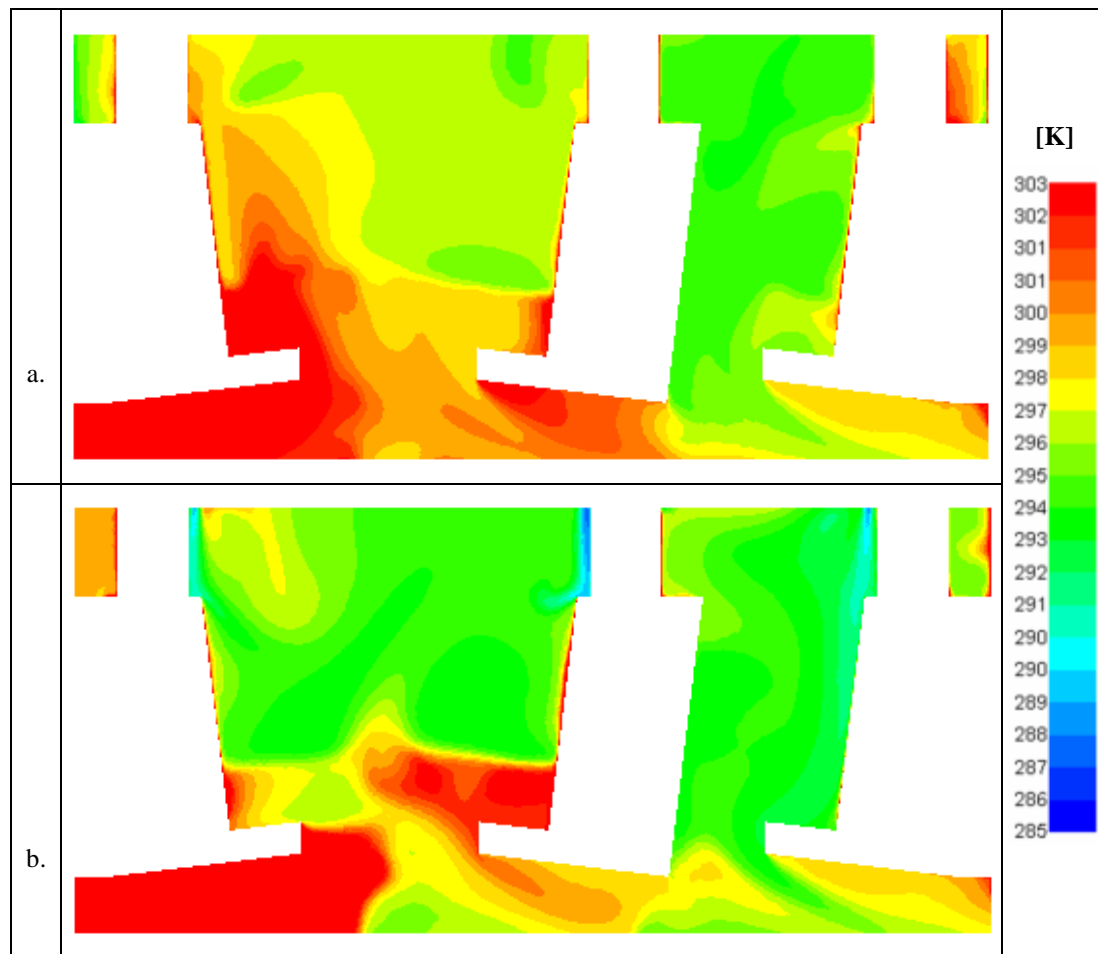
**Figure 6. Velocity contours on; (a) vertical plane passing through the interior passengers; (b) horizontal plane passing through the middle of the passengers' head level**

To investigate temperature distribution uniformity, seven vertical planes are determined to observe the temperature fields. The projection of the seven vertical planes and corresponding indexes are shown in Figure 7.



**Figure 7. Locations and indexes of the temperature field investigation planes**

Simulation results showed considerable temperature difference between the upper and lower parts of the cabin as illustrated in Figure 8 for Y6 Plane. The reason of this temperature difference is the insufficiency of the air movement in the lower part of the cabin since for the air passage, only narrow spaces exist between the legs of the passengers. It should be noted that, temperature contour of the existing design is also included into Figure 8 for comparison.



**Figure 8. Temperature contours on Y6 Plane for; (a) existing design; (b) proposed design**

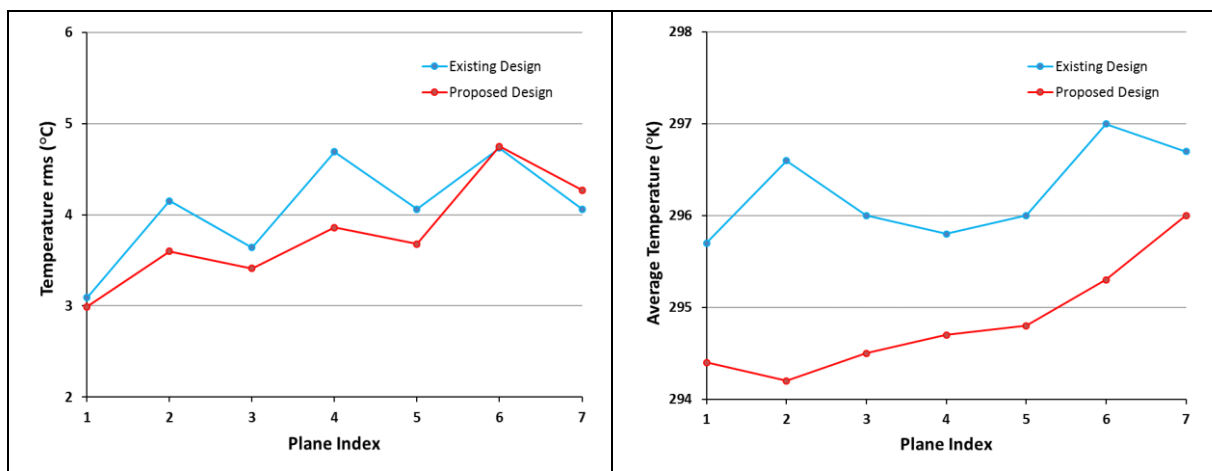
For the thermal comfort of the passengers, temperature distribution at the upper part of the cabin can be considered as more crucial since the faces and most of the body parts of the passengers exist in that region. Therefore, for the convenience of the thermal comfort investigation, vertical planes are divided into two as upper and lower regions. This division plane is shown in Figure 9.



**Figure 9. Division plane used for temperature distribution uniformity investigation**



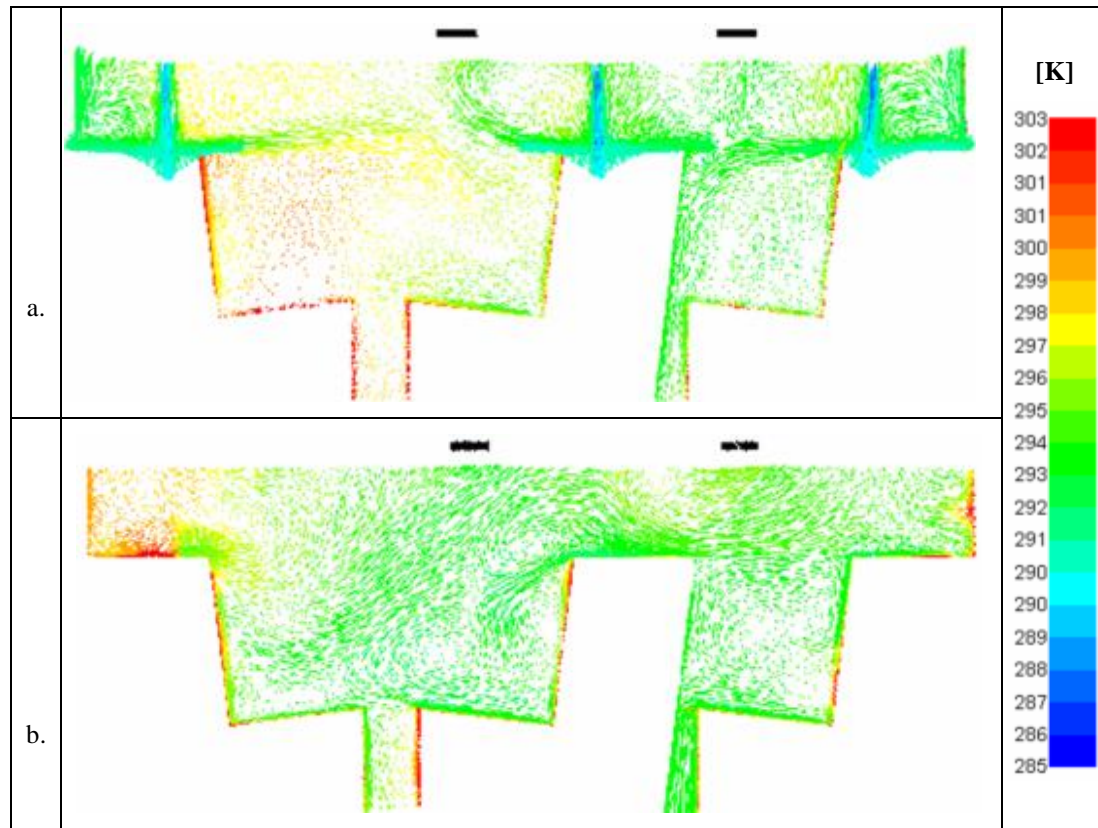
Considering the temperature requirement, uniformity of the temperature distribution is examined by using temperature root mean square (rms) values. When evaluating the temperature rms results, it is important to multiply values by two for reasonable evaluation since temperature rms value represents the standard deviation from the average whereas the temperature requirement defines the total range of the allowable variation. In Figure 10, temperature rms values for the upper cabin region is presented on seven investigated planes. The results show that the temperature rms values of the new design is lower than the existing design's which means that the new design provides more uniform temperature distribution in the upper region of the cabin. To evaluate uniformity of temperature distribution, average temperature values of the upper cabin region are also examined as shown in Figure 10. It can be observed that, average temperature values of the new design are nearly at least 1°C lower than the existing design's on each of the seven data planes. This indicates the improvement shown up by the new design on air distribution. Moreover, average temperature values of the new design vary rather smoothly between adjacent planes which cannot be observed for the existing design completely.



**Figure 10. Temperature rms values and average temperature values on investigated planes – Upper cabin region**

To observe the improvement on air movement; hence, on temperature distribution uniformity, velocity vector fields colored by temperature are compared for the existing design and the newly proposed design. In Figure 11, comparison is made on Y4 Plane which crosses the gaspers of the existing design. It should also be noted that main inlet velocities are same for both cases.

Considering the vector field magnitudes of both compared cases, it is observed that, although Y4 Plane crosses the gaspers of the existing design and gasper inlet velocities are much higher than the proposed design's, as a consequence of its location and geometry, the existing design cannot provide sufficient air movement. As observed from the vector fields of the existing design, jet flow arising from the gasper impinges on the body of the passenger and spreads radially before going towards to the upper parts of the cabin. Since gasper jets do not go through the bodies of passengers directly, temperature values are much higher around the bodies of passengers. On the other hand, since air velocities are higher for the proposed design almost on whole plane, it can be observed that conditioned air is distributed throughout the plane more uniformly.

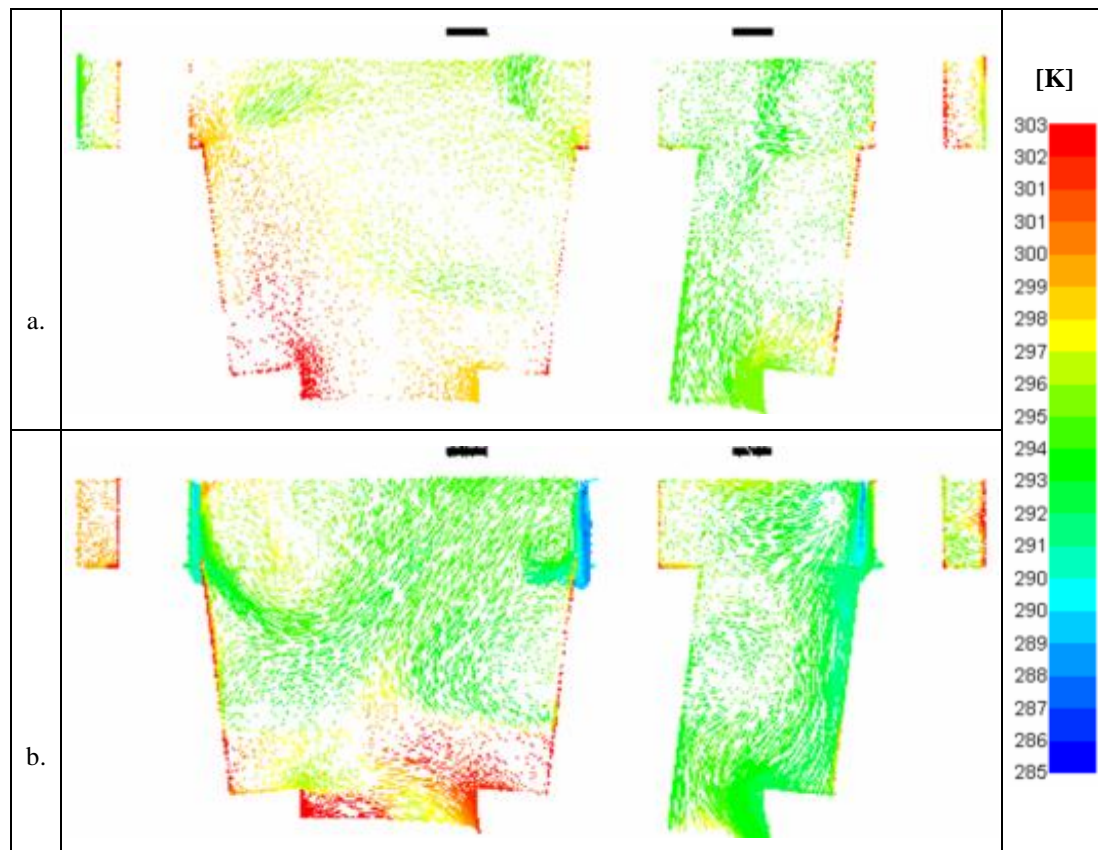


**Figure 11. Velocity vector fields colored by temperature on Y4 Plane for; (a) existing design; (b) proposed design**

To expand the temperature field investigation, another comparison is made on Y6 Plane which passes through the legs of the passengers sitting at the interior seats and is located on the air passage on the way of outlet, as shown in Figure 12.

It can be observed that, for the existing design, because of the low air velocities, hot air between the legs of the passengers sitting in the first and second rows penetrates to the upper region of the cabin by a thermal plume effect because of the density difference. Therefore, the upper part of the cabin becomes hotter and more non-uniform. On the other hand, although air is not distributed properly, a hot region is not observed around the passenger sitting at the third row since flow emerging from the gasper of that region passes around the third-row sitting passenger's body without spreading too much because of the physical feature of the cabin. Therefore, the temperature distribution is not as non-uniform as the forward part of the cabin. For the proposed design case of the same figure, it can be observed that air velocities are much higher and temperature values are much lower. The hot air between the legs of passengers are restrained by the jet flow; therefore, could not heat up the upper part of the bodies of the passengers.

Considering temperature variation requirement, it can be interpreted that because of the complex geometry and dense occupancy features of the cabin, showing exact compliance with the requirement is quite difficult by improving air inlet design. However, the new gasper design shows up improvement in terms of temperature distribution uniformity in the upper region of the cabin.



**Figure 12. Velocity vector fields colored by temperature on Y6 Plane for; (a) existing design; (b) proposed design**

## CONCLUSION

In this study, thermal comfort conditions of the passengers are numerically investigated in a newly designed passenger transportation helicopter by using CFD modeling. The result showed that the existing air supply system is inadequate in creating comfortable environment for the passengers. By an iterative procedure, a new design providing the desired air velocities around the faces of the passengers and showing up improvement in terms of temperature distribution is proposed.

## REFERENCES

- American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc. (2007) *ANSI/ASHRAE Standard 161-2007 Air Quality within Commercial Aircraft*, Atlanta, GA, 2007.
- American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc. (2013) *ANSI/ASHRAE Standard 161-2013 Air Quality within Commercial Aircraft*, Atlanta, GA, 2013.
- Baker A. J. (2005) "Validation for CFD prediction of mass transport in an aircraft passenger cabin," University of Tennessee, Knoxville, 2005.
- Du X., Li B., Liu H., Wu Y. and Cheng T. (2017) "The appropriate airflow rate for a nozzle in commercial aircraft cabins based on thermal comfort experiments," *Building and Environment*, vol 112, pp. 132-143, Feb. 2017.
- Günther G., Bosbach J., Pennecot J., Wagner C., Lerche T. and Gores I. (2006) "Experimental and numerical simulations of idealized aircraft cabin flows," *Aerospace Science and Technology*, vol 10, no. 7, p. 563–573, Oct. 2006.
- Kühn M., Bosbach J. and Wagner C. (2009) "Experimental parametric study of forced and mixed convection in a passenger aircraft cabin mock-up," *Building and Environment*, vol. 44, no. 5, p. 961–970, May 2009.
- Launder B. E. and Spalding D. B. (1972) in *Lectures in Mathematical Models of Turbulence*, London, Academic Press, 1972, p. 169.
- Li B., Li J., Huang Y., Yin H., Lin C.-H., Wei D., Shen X., Liu J. and Chen Q. (2015) "Experimental studies of thermal environment and contaminant transport in a commercial aircraft cabin with gaspers on," *Indoor Air*, vol 26, no. 5, p. 806–819, Nov. 2015.
- Liu W., Wen J., Lin C.-H., Liu J., Long Z. and Chen Q. (2013) "Evaluation of Various Categories of Turbulence Models for Predicting Air Distribution in an Airliner Cabin," *Building and Environment*, vol 65, pp. 118-131, July 2013.
- SAE International (2003) *SAE ARP292 - Environmental Control Systems for Helicopters*, C ed., 2003.
- Wang M. and Chen Q. (2009) "Assessment of Various Turbulence Models for Transitional Flows in an Enclosed Environment (RP-1271)," *HVAC&R Research*, vol 15, no. 6, pp. 1099-1119, Nov. 2009.
- Yan W., Zhang Y., Sun Y. and Li D. (2009) "Experimental and CFD study of unsteady airborne pollutant transport within an aircraft cabin mock-up," *Building and Environment*, vol 44, no. 1, p. 34–43, Jan. 2009.
- You R., Chen J., Shi Z., Liu W., Lin C.-H., Wei D. and Chen Q. (2016) "Experimental and Numerical Study of Airflow Distribution in an Aircraft Cabin Mockup with a Gasper On," *Building Performance Simulation*, vol 9, no. 5, pp. 555-566, Jan. 2016.
- Zhang Z., Chen X., Mazumdar S., Zhang T. and Chen Q. (2009) "Experimental and Numerical Investigation of Airflow and Contaminant Transport in an Airliner Cabin Mockup," *Building and Environment*, vol 44, no. 1, pp. 85-94, Jan. 2009.
- Zhang Z., Zhang W., Zhai Z. J. and Chen Q. Y. (2007) "Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 2—Comparison with Experimental Data from Literature," *HVAC&R Research*, vol 13, no. 6, pp. 871-886, Aug. 2007.