Vol. 10, No 2/2021, 53-59

DOI: 10.17512/bozpe.2021.2.07

Construction of optimized energy potential
Budownictwo o zoptymalizowanym potencjale energetycznym

ISSN 2299-8535 e-ISSN 2544-963X

Application of the Ansys Fluent program for modelling environmental phenomena in a living room

Mariusz Kosiń[1] (*orcid id: 0000-0003-2683-7784*)
[1] Czestochowa University of Technology

Abstract: The article presents a three-dimensional numerical analysis of heat and humidity parameters carried out for a living room in a multi-family building. The aim of the analysis was to compare alternative heating methods with the existing one. Two cases were included in the analysis: case I – the existing state, case II – the existing state with an additional panel heater. In order to verify the numerical analysis for case I, the temperature and relative humidity were measured. During the heating period, the thermal conditions in the room did not favor the comfort of users, especially in one part of the room. The study may be a tool for future research related to the implementation of the climatic conditions of rooms with similar structural and functional features.

Keywords: numerical modeling, CFD, thermal and humidity distribution, thermal comfort, temperature distribution

Please, quote this article as follows:

Kosiń M., Application of the Ansys Fluent program for modelling environmental phenomena in a living room, Construction of Optimized Energy Potential (CoOEP), Vol. 10, No 2/2021, 53-59, DOI: 10.17512/ bozpe.2021.2.07

Introduction

The author was prompted to take up the topic of the article due to the thermal discomfort in one of the rooms in an apartment located on the top floor of a multi-family building. Due to the linear arrangement of rooms and ensuring, in accordance with regulations on technical conditions that are required in buildings and their location (Journal of Laws, 2021), the room has two opposite windows. The heating solution adopted in the room consisted of a radiator on one of the walls. The feeling of cold, reported in the analyzed room, appears in conjunction with the beginning of the heating season.

One of the considered solutions to improve humidity and the thermal comfort of the room, is the placement of an additional heater under the window. With the progressive changes in energy consumption requirements and expectations regarding the thermal comfort inside the facility, it is necessary to understand the effects of the above-mentioned factors. Poor air quality in the room causes discomfort to the users.

In the article, the authors used a numerical simulation of heat and moisture flow. Due to the application of computational fluid dynamics (CFD), it is possible to obtain the necessary information about the environment of the analyzed room. Two cases were included in the analysis:

I – the existing state,

II – the existing conditions in the room with an additional panel heater.

The aim of the presented research was to use CFD to determine the improvement of the hygrothermal conditions. In order to verify the results obtained from the numerical analysis, a comparison was made with the results of the study. The results were validated for case I.

The results presented in the article can provide information on the design of rooms in terms of environmental comfort, as well as contribute to the study of the climate when using various heating and ventilation systems.

1. Analysis

The living room of a multi-family building with the following dimensions was analyzed: 3.5 m wide, 4.9 m long and 2.8 m high (Fig. 1). Two types of heating were included in the calculations. Case I reflected the actual condition of the room, which is heated by a 0.9 m x 0.6 m steel two-panel radiator placed on a wall with a balcony window (Fig. 2). Case II assumed the placement of an additional single-panel radiator with dimensions of 0.9 m x 0.6 m. The radiator was placed on the opposite wall to case I (Fig. 2).

a) b)

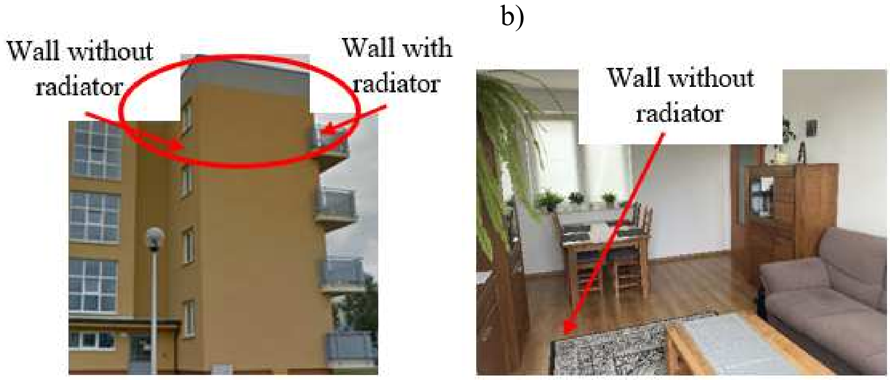


Fig. 1. Photo of the analyzed room: a) external view, b) internal view (*own photo*)

2. Case study

Modelling of the spatial distribution of temperature and relative humidity in the accepted cases was carried out by computational fluid dynamics (CFD) with the use of Ansys Fluent.

The steps of the computational analysis included the geometric modelling, finite element mesh creation, analysis and result evaluation. In order to carry out the analysis, the boundary conditions were determined: inlet – air inlet area, outlet – air outlet area, heating element and walls.

Due to the shortening of the calculation time, a mesh was generated that contained tetrahedral elements. Detailed information on mesh generation and solution methods was taken from the literature (*Fluent 19.2*, Major & Kosiń, 2016).

Figure 2 shows the computational model of the analyzed room. In the analysis, the heater panel was set to a temperature of 60°C. The air velocity on the surface of the inlet opening was assumed to be 3.6 m/s, which is equivalent to an air pressure of about 10 Pa. The inlet air temperature was set at a constant –10°C. The mass fraction of water vapour in the inlet air was assumed to be 4.17 g/kg based on the Mollier diagram (Szymański & Wolańczyk, 2014). A pressure boundary condition was applied at the outlet. The air outflow was possible through a gap under the door (0.9 m x 0.02 m) leading to the corridor of the apartment. For the window and the outer surface of the window wall, the following temperature boundary conditions were used: for the outer wall –10°C and the inner wall 20°C.

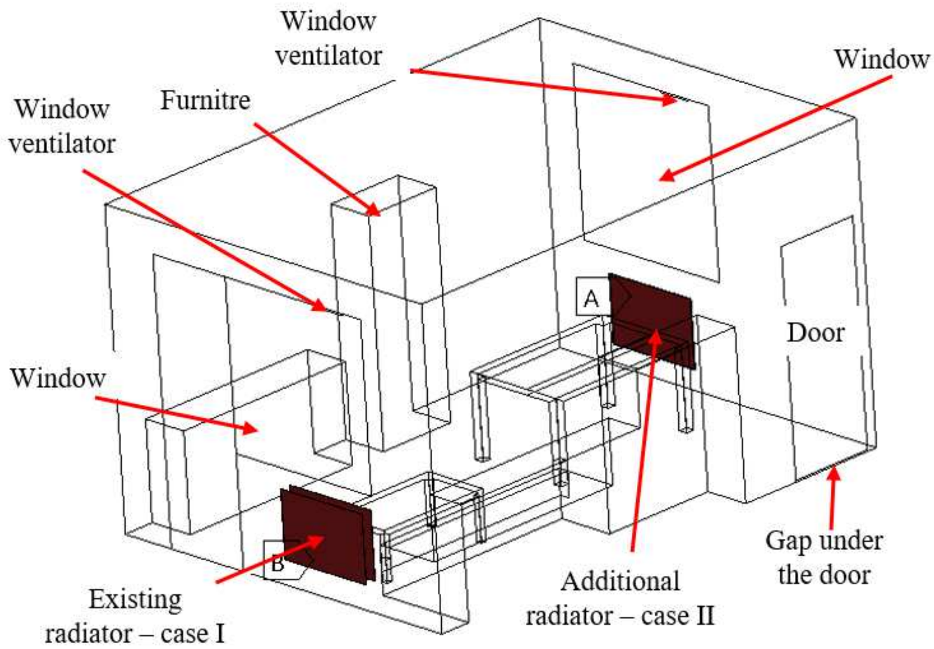


Fig. 2. Living room calculation model (*own research*)

3. Results

In order to verify the numerical calculations, the temperature and relative humidity were measured for case I. The measurement site was located in the central part of the room, and the readings were taken at five heights (Fig. 3a). A temperature and humidity recorder (AZ model 8700) was used for the measurements (Fig. 3b). The measurement accuracy of the relative humidity was ±5% and for the temperature, it was ±1°C.

The data measured and calculated by Ansys Fluent (Major et al., 2018) show similar trends. Based on the test results, the differences between the results obtained from the CFD analysis were estimated. The discrepancy between the temperatures was 12% and for humidity 14%. Taking into account the many parameters influencing the environmental climate of the room, the obtained results can be considered as satisfactory.

a)

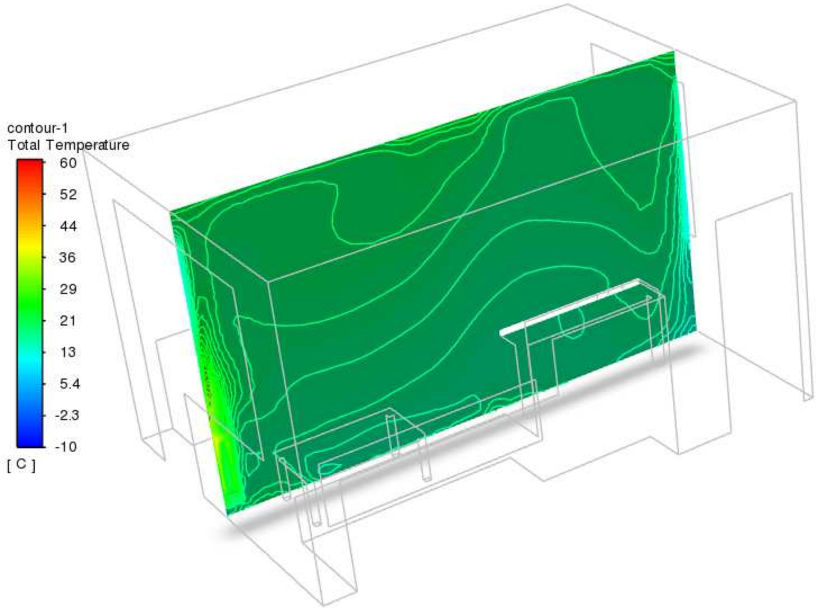
b)

Fig. 3. Temperature and humidity measurement: a) measuring place, b) AZ model 8700 hygrometer (*own photo*)

Figures 4 and 5 show the temperature and relative humidity fields in the vertical plane located in the middle of the room.

In case I, the area of cooler climate is visible in the area of the window without the radiator (Fig. 4a). On the other hand, the simulation with an additional heating panel (case II) shows a balanced temperature distribution (Fig. 4b).

a)



b)

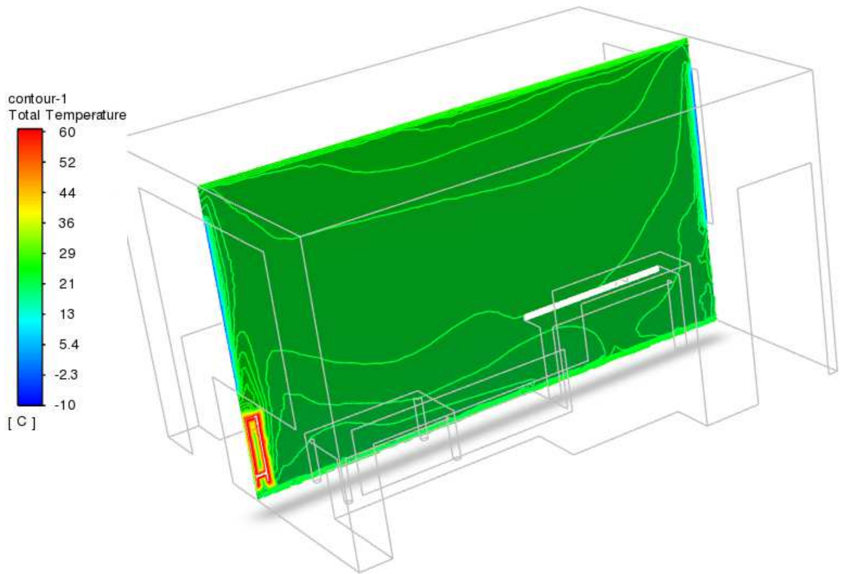
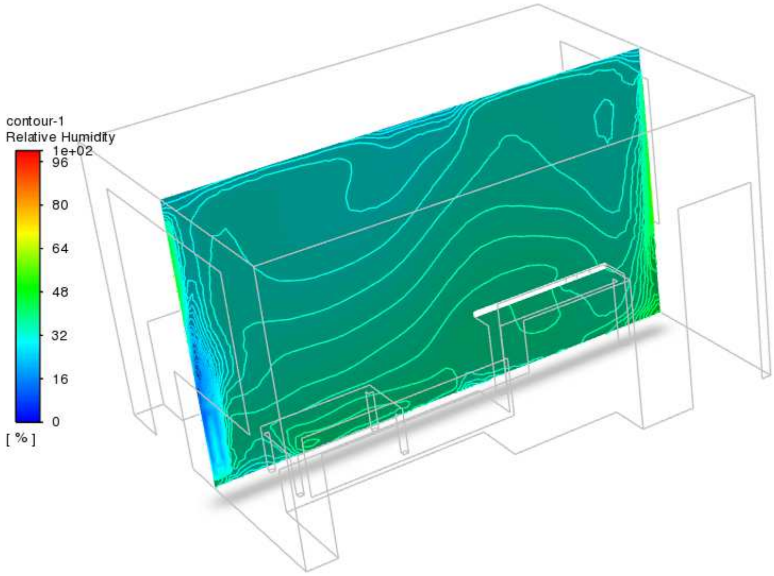


Fig. 4. Temperature distribution: a) case I, b) case II (*own research*)

Figure 5 shows the distribution of relative humidity. In the case of closed rooms, the air humidity should be in the range of 30-65%. The best feeling for people staying in the room is when the relative air humidity is 40-60% with the optimal temperature in the room of 20-22°C. In case I (Fig. 5a) the air is more humid than in case II, where the air is drier (Fig. 5b).

a)



b)

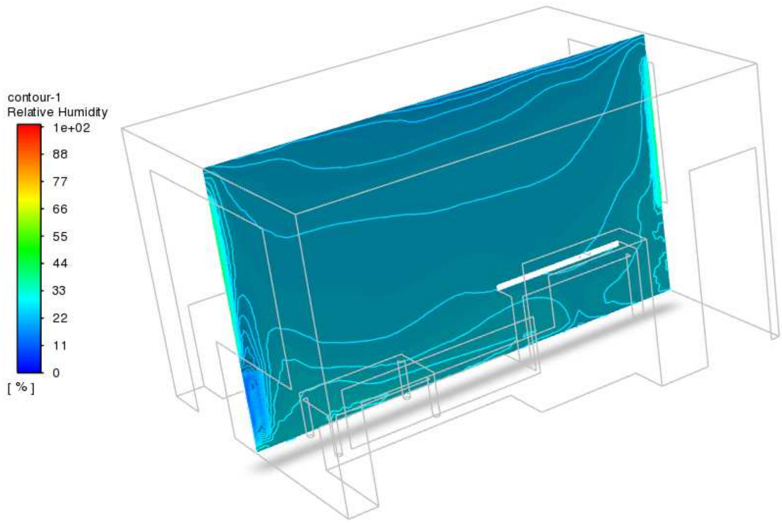


Fig. 5. Relative humidity distribution: a) case I, b) case II (*own research*)

Conclusions

The aim of this study was to model, using numerical methods, issues related to the assessment of temperature and humidity distribution with various heat carriers. The use of numerical programs allows for a relatively quick assessment of the adopted methods of heating rooms. Numerical calculations provide both information on the values of the parameters of interest to us as well as graphically present the distribution of these parameters in the analyzed room. Thanks to this, at the design stage, it is possible to analyze technological and construction solutions that are extremely important at the stage of the operation of the facility.

Bibliography

Fluent 19 User's Guide (2019), Fluent Inc., Lebanon, NH, USA.

Major, M. & Kosiń, M. (2016) *Modelowanie rozkładu temperatur w przegrodach zewnętrznych wykonanych z użyciem lekkich konstrukcji szkieletowych*. Budownictwo o Zoptymalizowanym Potencjale Energetycznym, 55-60.

Major, M. et al. (2018) *Numerical analysis of the temperature distribution in an office room*. Koszalin, Wydawnictwo Uczelniane Politechniki Koszalińskiej, 187-198.

Rozporządzenie Ministra Infrastruktury z dnia 12 kwietnia 2002 r. w sprawie warunków technicznych, jakim powinny odpowiadać budynki i ich usytuowanie.

Szymański, W. & Wolańczyk, F., (2014) *Termodynamika powietrza wilgotnego. Przykłady i zadania.* Warszawa, Wydawnictwo Politechniki Warszawskiej, 40-42.