

NUMERICAL INVESTIGATION OF NATURAL CONVECTION IN A RECTANGULAR CAVITY BY CFD CODES

Evrım ÖZRAHAT^{1*}, Mehmet TOPRAK², Sebahattin ÜNALAN³

¹Kayseri University Bünyan Vocational School, <https://orcid.org/0000-0002-6912-9633>

²Erciyes University, Institute of Sciences, <https://orcid.org/0009-0008-1396-2416>

³Erciyes University, Engineering Faculty, <https://orcid.org/0000-0002-5605-2614>

ABSTRACT. In this study natural convection in air-filled rectangular cavity is studied numerically by two commercial CFD codes. Numerical simulations are carried out for air and water for different aspect ratios and wall temperatures. Calculations are compared with experimental correlations in the literature and with each other. Calculation time and mesh number are the main parameters considered in this study. According to the numerical results SIMCENTER FLOEFD is an effective CFD code as ANSYS FLUENT and gives accurate results for this type of problems.

Keywords: Natural Convection, Conjugate Heat transfer, CFD Codes, Rectangular cavity.

Article History: Received 20.09.2023; Accepted:29.09.2023; Available online:30.09.2023

Doi: <https://doi.org/1052924/EFIJ8822>

1. INTRODUCTION

Natural convection in a cavity occurs in many living areas Heating cooling in a room or heat transfer in a double pane window or heat transfer in an air-filled brick can be examples of natural convection in daily life. So, researchers are interested in understanding this phenomenon. Frederick [1], carried out a numerical study of two-dimensional natural convection of air in rectangular, differentially heated cavities. For three Rayleigh numbers the aspect ratio S was varied. It was found that heat transfer between the active walls has a maximum at each Rayleigh number, which is located at values of S between 1 and 2. The aspect ratio for which the maximum heat transfer occurs is determined by the process of transition from a shallow cavity regime to a slender cavity regime.

Manz [2] investigated heat transfer by the natural convection of air layers within vertical, rectangular cavities with aspect ratios (A) of 20, 40 and 80 in relation to applications in building facade elements, such as insulating glazing units, double-skin facades, doors, etc. using a computational fluid dynamics (CFD) code. Boundary conditions were assumed to be isothermal hot and cold wall, and zero heat flux at bottom and top cavity surfaces. Rayleigh numbers were between 1000 and 10^6 , i.e., flow was either laminar or turbulent, and a conduction, transition or boundary layer regime was applied. His study improved the starting position for future applications of the code to more complex cases of facade elements, where less or even

no experimental data are available in literature. Ganguli at all [3], performed independent CFD simulations in order to predict the variation in heat transfer coefficient setup by natural convection for various tall slender vertical geometries with varying gap widths and temperature differences covering the ranges reported in the literature. A good agreement of Nusselt number ($\pm 10\%$) has been found between the CFD predictions and the literature data. Bahlaoui at all [4], reported numerical results of mixed convection and surface radiation within a horizontal ventilated cavity heated from below and provided with an adiabatic thin partition on the heated surface. Air, a radiatively transparent medium, is considered to be the cooling fluid. Mahdavi at all [5], studied laminar natural convection in a rectangular cavity with three different heat transfer fluids: water, ethylene glycol (EG)–water and air experimentally and numerically. The enclosure has a uniform aspect ratio (AR). The EG–water mixture is made up of 60% EG and 40% water. The main experiments aimed to reach proper thermal boundary conditions for the two differentially heated vertical walls of the cavity. Early experiments revealed that it is hard for the heated and cooled walls to reach a uniform temperature when the cavity is filled with water or EG–water, while a uniform distribution of temperature was achieved when it is simply filled with air. Commercial computational fluid dynamics (CFD) software, ANSYS-FLUENT 15, simulated the entire setup to include two special heat exchangers and the cavity between them to investigate all the transport phenomena. The simulation results were in good agreement with

*Corresponding author: evrimozrahathat@kayseri.edu.tr

measured data. Cianfrini et al. [6], studied natural convection in air-filled rectangular cavities inclined with respect to gravity, so that the heated wall is facing upwards, numerically under the assumption of two-dimensional laminar flow. A computational code based on the SIMPLE-C algorithm is used for the solution of the system of the mass, momentum and energy transfer governing equations. Simulations are performed for height-to-width aspect ratios of the enclosure from 0.25 to 8, Rayleigh numbers based on the length of the heated and cooled walls from 10^2 to 10^7 , and tilting angles of the enclosure from 0° to 75° . The existence of an optimal tilting angle is confirmed for any investigated configuration, at a location that increases as the Rayleigh number is decreased, and the height-to-width aspect ratio of the cavity are increased, unless the value of the Rayleigh number is that corresponding to the onset of convection or just higher. Dimensionless correlating equations are developed to predict the optimal tilting angle and the heat transfer performance of the enclosure. Prasopchingchana [7] investigated the effects of cavity aspect ratios and cavity inclination angles to natural convection in a rectangular cavity numerically. Investigation is performed at the Rayleigh number (Ra) equal to 10^4 , the cavity aspect ratios from 1 to 50 and the cavity inclination angles from 0 to 180° . Consequently, Heat transfer enhancement or decreasing due to the effects is exposed. In addition, streamline contours in the rectangular cavity are illustrated. Multi-cellular flow figuring on the appropriate conditions is exhibited. A new correlation of the average Nusselt number, the cavity aspect ratio and the cavity inclination angle are formulated at Ra equal to 10^4 . Hernández-Castillo et al. [8], studied natural convection coupled with radiation numerically in a tall rectangular parallel piped cavity filled with air and constant differentially heated opposite vertical walls. This type of tall cavity is found in walls built with hollow blocks.

Three-dimensional steady state numerical simulations are performed. Numerical models for pure natural convection and natural convection coupled with radiation are validated with experimental results. For pure natural convection, a Nu_c correlation as a function of Ra_w is obtained. A

simplified calculation of the total heat transfer, Nu_t , is proposed. Sundur and Mescher [9] explored natural convection of air in a tall rectangular cavity by solutions of the fully compressible transient Navier–Stokes equations in two-dimensional form, without the Boussinesq approximation. A temperature difference is imposed on the two vertical side walls, each isothermal, with the two horizontal boundaries adiabatic. Thermo-physical properties of air, including density, viscosity, thermal conductivity and specific heat, are all variable with temperature. Contrary to the conclusions in most previous numerical studies invoking the Boussinesq approximation, this study finds the instability of the conduction regime to be a traveling wave drifting downward in the cavity. The wave-drift speed is a strong function of the dimensionless wall temperature difference, e , defined as the temperature difference between the vertical walls divided by twice the mean wall temperature. Wave-drift speeds and wave numbers are calculated over a range of e less than 0.5 and Rayleigh numbers less than 9000. The dimensionless temperature difference e is also found to have significant

effects on local heat transfer and transient flow structure inside the cavity.

Parallel to the literature findings this study performs numerical analyses with two commercial CFD codes. Also includes the comparison with experimental data from the literature.

2. NUMERICAL MODEL

The 3D rectangular cavity filled with air has 100x100x400 mm dimensions as seen in Figure 1. Boundary conditions were assumed to be isothermal hot and cold wall, and zero heat flux at bottom and top cavity surfaces. Two opposite walls are determined as constant temperature boundary condition at 273 and 283 K. The filling fluid air is assumed as incompressible ideal gas and its thermophysical properties are given in Table 1. The simulation is done for Rayleigh number 10^6 corresponding to the 273 K and 283 K wall temperatures. Gravity is taken into account with Boussinesq approximation. Simulations are done both with ANSYS FLUENT [10] and SIMCENTER FLOEFD [11]. For all computations same computer (Intel core i7 4 GPU) is used. Convergence criterion is 10^{-6} for all parameters. It should be noted that this cavity has an aspect ratio (H/L) equal to 4.

Fig. 1. Rectangular cavity filled with air

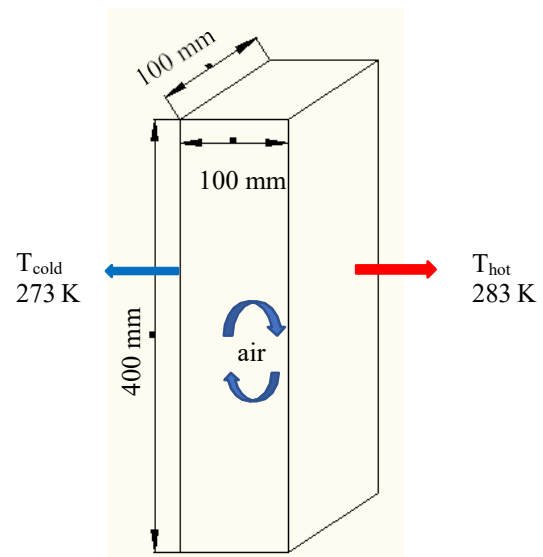


Table 1 Thermophysical properties of air

	A	B	C	D	E
k	62500	422	2:26	0,096	
cp	256000	425	10:08	0,107	
nü	500000	437	18:57	0,109	

For the calculation of Nusselt number inside the cavity following correlation is used [12]:

$$Nu_L = 0,22 \left(\frac{Pr}{0,2+Pr} \right)^{1/4} \left(\frac{Ra_L}{L} \right)^{1/4} \quad (1)$$

$$2 < \frac{H}{L} < 10$$

$$Pr < 10^5$$

$$10^3 < Ra_L < 10^{10}$$

3. NUMERICAL RESULTS

Both for the mesh independency and comparative study 3 models determined with increasing mesh size (Model-1, Model-2, Model-3 in Table 2). Calculations are done with the same computer with same GPU. Temperature contours inside the cavity at x-y plane at steady state are given in Figure 2 and Figure 3 for both FLUENT and FLOEFD respectively.

Fig. 2. Temperature contours at xy plane inside the cavity (FLUENT)

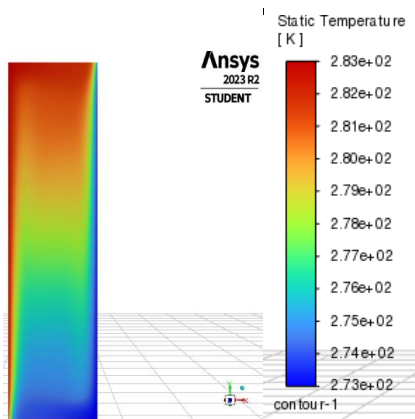
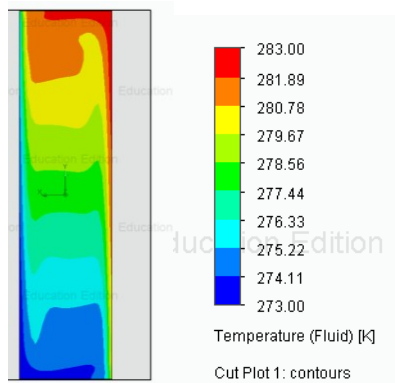


Fig. 3. Temperature contours at xy plane inside the cavity (FLOEFD)



Temperature of the fluid rises when flowing up near the hot wall and declines flowing down the cold wall as expected. Comparison of the models with different parameters are summarized in Table 2. Increasing the mesh size increased

the calculation time and iteration number for both codes. Velocity value differs in a negligible level between the Model-2- to Model-3 as 0,2%. For Model 1 calculation time of the two codes is very close. For model 2 calculation time is nearly same too. However, for more mesh numbers

(Model-3) calculation time differs significantly.

Table 2 Comparison of numerical simulation results

Ra= 10 ⁶	Ref [12]	ANSYS FLUENT	SIMCENTER FLOEFD
Nusselt Number	7,019	7,541	7,311
Deviation (%)		7,429	4,160

In addition, an important parameter Nusselt number for the air-filled cavity is calculated by equation 1 for Model-3 for both numerical codes and experimental study from the literature [12]. The results are summarized in Table 3. Error values are also calculated and given in Table 3. According to the table numerical results from FLUENT deviates by 7 percent from the experimental value. Numerical results from FLOEFD deviates by 4 percent from the experimental value. These results are proving the acceptability of CFD codes in numerical simulations.

4. CONCLUSIONS

CFD codes are effective ways for the analyses of natural convection in cavities. Both codes used in this study are acceptable for accurate results. According to the simulation results, results from FLUENT deviates by 7 percent from the experimental value. Numerical results from FLOEFD deviates by 4 percent from the experimental value. These results are proving the acceptability of CFD codes in numerical simulations.

REFERENCES

- [1] R. L. FREDERICK, On the aspect ratio for which the heat transfer in differentially heated cavities is maximum. International communications in heat and mass transfer, 26.4: 549-558, 1999.
- [2] H. MANZ, Numerical simulation of heat transfer by natural convection in cavities of facade elements. Energy and buildings, 35.3: 305-311, 2003.
- [3] A.A. GANGULI, A. B. PANDIT, J.B. JOSHI, CFD simulation of heat transfer in a two-dimensional vertical enclosure. Chemical Engineering Research and Design, 87.5: 711-727,2009.
- [4] A. BAHLAOUI, A. RAJI, M. HASNAOUI, M. NAIMI, T. MAKAYSSI., M. LAMSAADI (2009). Mixed convection cooling combined with surface radiation in a partitioned rectangular cavity. Energy Conversion and Management, 50(3), 626-635, 2009.
- [5] M. MAHDAVI et al. Experimental and numerical study of the thermal and hydrodynamic characteristics of laminar natural convective flow inside a rectangular cavity

with water, ethylene glycol–water and air. *Experimental Thermal and Fluid Science*, 78: 50-64, 2016.

[6] C. CIANFRINI, et al. Effects of the aspect ratio on the optimal tilting angle for maximum convection heat transfer across air-filled rectangular enclosures differentially heated at sides. *Journal of Thermal Science*, 26: 245-254, 2017.

[7] U. PRASOPCHINGCHANA, U. Numerical study of natural convection in a rectangular cavity with variation of cavity aspect ratios and cavity inclination angles. In: *IOP Conference Series: Materials Science and Engineering*. IOP Publishing. p. 012044, 2019.

[8] P. HERNÁNDEZ-CASTILLO, P., J.A. CASTILLO, G. HUELSZ, Heat transfer by natural convection and radiation in three dimensional differentially heated tall cavities. *Case Studies in Thermal Engineering*, 40: 102529, 2022.

[9] S. SONDUR, A.M. MESCHER, Non-Boussinesq effects on natural convection airflows simulated in a tall

rectangular cavity. *Numerical Heat Transfer, Part A: Applications*, 84.4: 297-314, 2023.

[10] ANSYS FLUENT 2023 STUDENT VERSION
<https://www.ansys.com/academic/students>

[11] SIMCENTER FLOEFD

<https://resources.sw.siemens.com/en-US/download-computational-fluid-dynamics-simcenter-solid-edge-student>

[12] F. Incropera, W. Dewitt, *Fundamentals of Heat and Mass Transfer*, Wiley, 2004.