

PAPER • OPEN ACCESS

Effect of Solar Chimney and PCM cooling ceiling to the air flow inside a naturally-ventilated building

To cite this article: Suhendri 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **245** 012051

View the [article online](#) for updates and enhancements.

Effect of Solar Chimney and PCM cooling ceiling to the air flow inside a naturally-ventilated building

Suhendri

Labtek IXB Architecture, Institut Teknologi Bandung, Ganesha 10 Bandung, Indonesia

Email: suhendri.aritb@gmail.com

Abstract. Computational fluid dynamics (CFD) gives significant contributions to building design fields. Building ventilation is the most common problem to simulate in CFD software. 2D and 3D simulation can be used to simulate cross ventilation within the building. ANSYS Fluent is commonly used CFD software, and it has been used in this study to assess the environment of an atrium building under buoyancy-driven natural ventilation and wind-driven natural ventilation. Buoyancy is one of the main forces driving flows. Buoyancy effect happened when the moving fluid is lighter or heavier than surrounding fluid. This paper will elaborate the use of ANSYS Fluent in assessment the environmental performance of a building. Some cases on a building were set in this study. The aim is to simulate accurately buoyancy-driven air flow inside a building for natural ventilation, visualize and analyse simulation results, provide architectural/engineering solution to natural ventilation of buildings. Cooler air appears from ceilings and creates cooling to the room down. Meanwhile, solar chimney generates wind flow across the building due to temperature differences within the chimney.

1. Introduction

Building design, and other engineering fields that related to it, have a high demand on building performance review. Since sustainable building has been promoted as one of the action that can slow down the Global Warming, it is important to review the environmental performance of buildings. Fluid dynamics inside and outside the building is considered in building environmental performance review. Computational fluid dynamics (CFD) then gives significant contributions.

Building ventilation is the most common problem to simulate in CFD software. 2D and 3D simulation can be used to simulate cross ventilation within the building. Ventilation is also related to building thermal environment, hence the CFD software can also be used to review thermal environment of the building.

ANSYS Fluent is commonly used CFD software. It has been used in many engineering fields, also in building design. This paper will elaborate the use of ANSYS Fluent in assessment the environmental performance of a building. Some cases on a building was set in this paper. The aim is to simulate accurately buoyancy-driven air flow inside a building for natural ventilation, visualize and analyze simulation results, provide architectural/engineering solution to natural ventilation of buildings.



2. Methodology

2.1. General Specification of Building

An office building was used as a case study. It consists of three storey open planned office. The floors are linked to an atrium that gets 44 W/m^2 solar heat gain on the inside facade surface. It is used a phase change material (PCM) that spread uniformly in the lower part of ceilings. Phase-change temperature of the PCM is 22°C , and with the natural ventilation, it is used for passive cooling in summer and night time cooling/thermal storage.

The dimensions of the building are shown in the Figure 1. The opening size of windows in the external wall increases from 0.3 m , 0.4 m to 0.5 m from the ground floor to the second floor. The size of openings in the internal wall is 0.5 m .

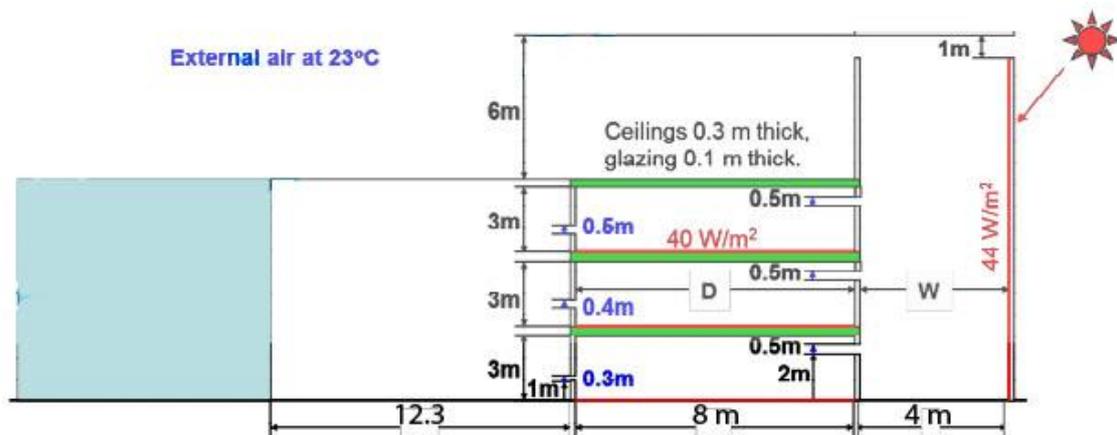


Figure 1. Building case

Outdoor temperature is at 23°C during the daytime. Equivalent internal heat gains for convection heat transfer on the floors of offices are 40 W/m^2 . Conduction and radiation heat transfer can be neglected in the simulation. The building is sufficiently long to allow simulation of two dimensional flows along the depth and height directions.

2.2. Buoyancy-Driven Natural Ventilation Simulation

Buoyancy is one of the main forces driving flows. Buoyancy effect happened when the moving fluid is lighter or heavier than surrounding fluid. Therefore, it is gravity that makes the buoyancy effect. Because air has different density in different temperature, so buoyancy effect can occur to it.

ANSYS Fluent has 3 steps. Firstly, the case boundary step. The geometry of the building must be built, then build the grid mesh of built-geometry. The mesh is an artificial artefact for the numerical method to assist calculation. Secondly, the solution setup step where the boundary condition, energy equation, and solution method that is needed in the calculation must be set. Then the calculation step, where the solution then can be calculated, and when it is finished, the results will appear in the fluent window.

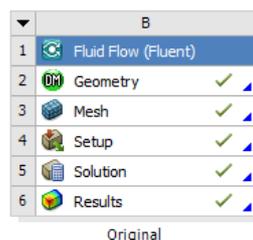


Figure 2. ANSYS Fluent Window

2.2.1. Geometry. For buoyancy-driven natural ventilation simulation, the geometry was built with the exact dimensions as explained. Because the wind effect will be neglected, it is not needed to draw the surrounding, in this case is the left building.

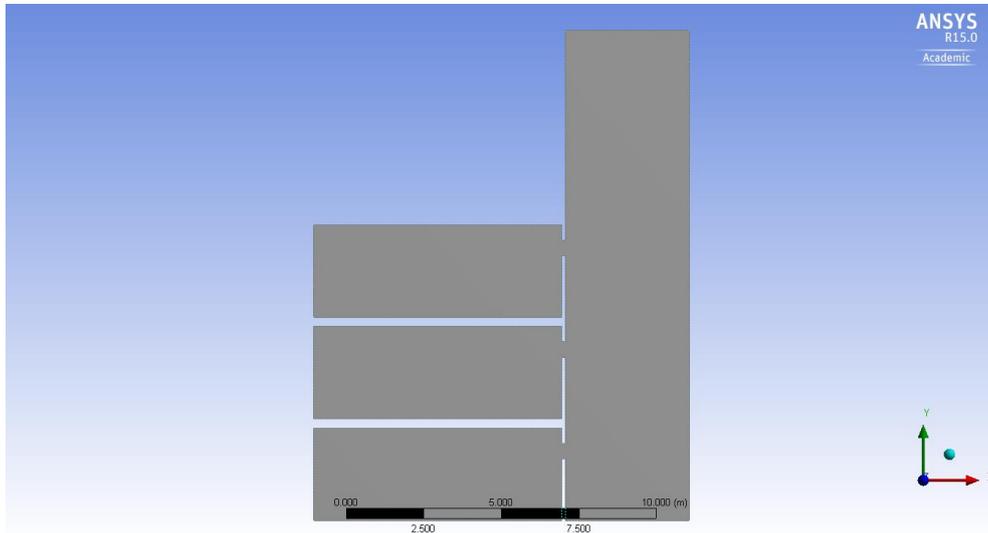


Figure 3. Geometry drawn for Buoyancy-Driven Ventilation

2.2.2. Meshing. The original mesh was set with face sizing to the selected geometry with 10cm mesh size. It had 14.093 nodes and 13.757 elements. Another thing to set in Mesh step is give name for the boundaries. It is important for the Setup step. As the grid used is an artificial artefact for the numerical method, the solution should be independent of the grid used. If the grid is coarse, averaging in the cells will remove fine details of the flow which may have the effect of altering the computational solution. A grid sensitivity analysis should be performed to demonstrate a particular solution is not dependent on the grid that has been used. (Richard Chitty & Chunli Cao, 2014)

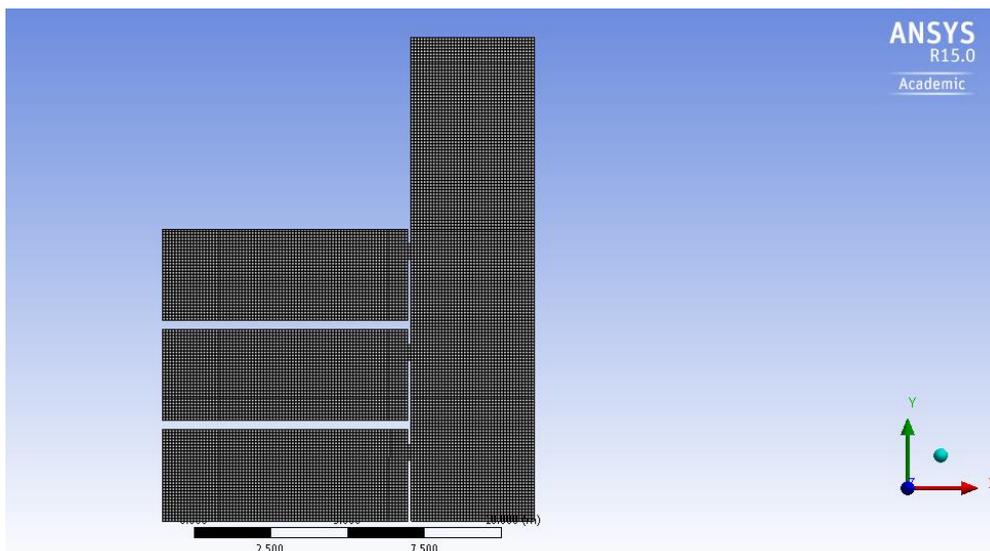


Figure 4. Mesh Set for Buoyancy-Driven Ventilation

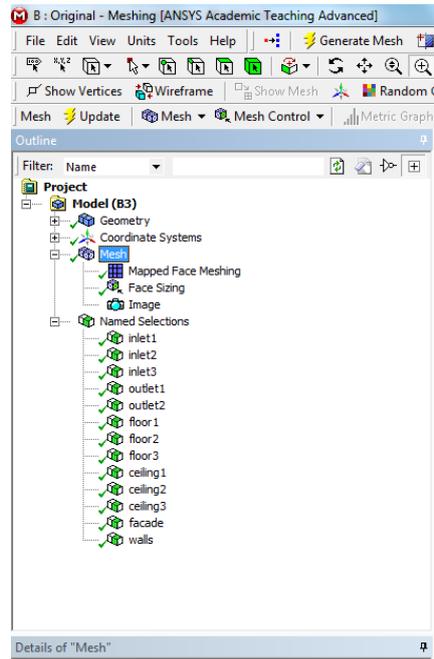


Figure 5. Boundary Name Set for Buoyancy-driven Ventilation

2.2.3. *Solution Setup.* These are some important matters that need to set in solution setup.

- a. *Models.* In Models setup, Energy Equation should be “on”. Then Viscous condition was used RNG k-epsilon model with enhanced wall treatment rather than used laminar model. For the remaining settings, let it as default.

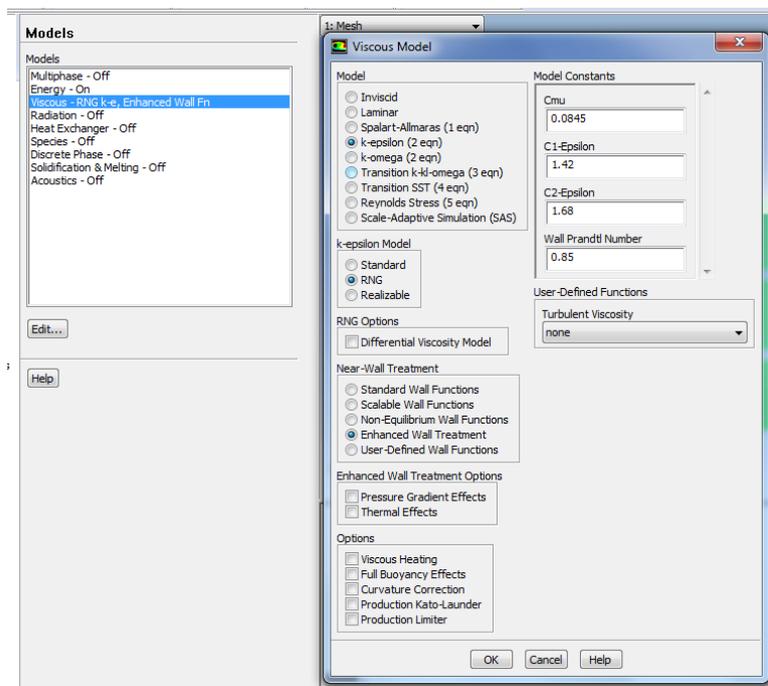


Figure 6. Model Setup for Buoyancy-driven Ventilation

- b. *Materials*. The fluid that was going to simulate is air, and in this part of the solution setup, air thermal properties must be set as the case study. In this case the air temperature is 23°C. For the buoyancy effect simulation, the type of air density should be changed to “boussinesq” for more reliable results. The thermal properties of air can be got in thermal properties calculator website (<http://www.mhtl.uwaterloo.ca/old/onlinetools/airprop/airprop.html>).

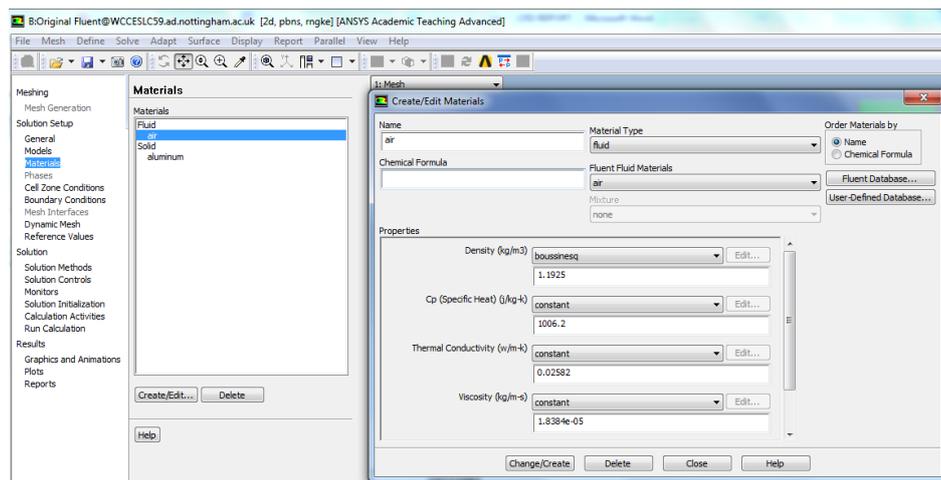


Figure 7. Materials Setup for Buoyancy-driven Ventilation

- c. *Cell Zone Conditions*. Operating Conditions of the surface body should be work on gravitational force. Enter the value of the gravitational acceleration in Y direction and also set the operating temperature and density as in Models setting.

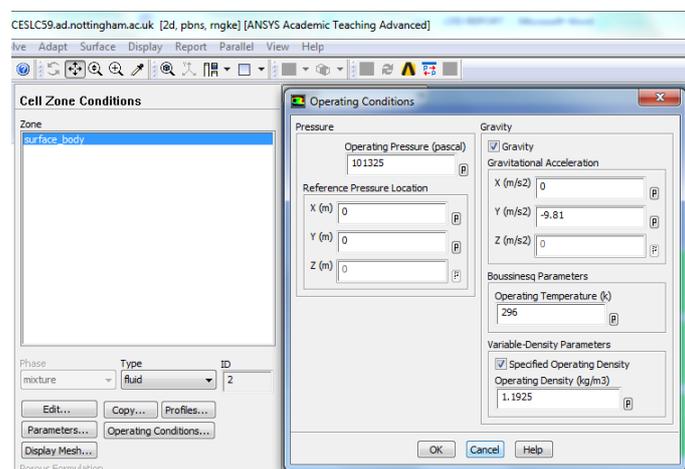


Figure 8. Cell Zone Conditions Setup for Buoyancy-driven Ventilation Boundary Conditions

It is the vital part of solution setup. Each boundary must be set as close as possible to the real condition. In this case, there are three types of boundary, pressure-inlet (inlet1, inlet2, inlet3), pressure-outlet (outlet1 and outlet2), and the remaining is wall. In pressure-inlet and pressure-outlet type of boundary, thermal and momentum properties should be set as in the picture below.

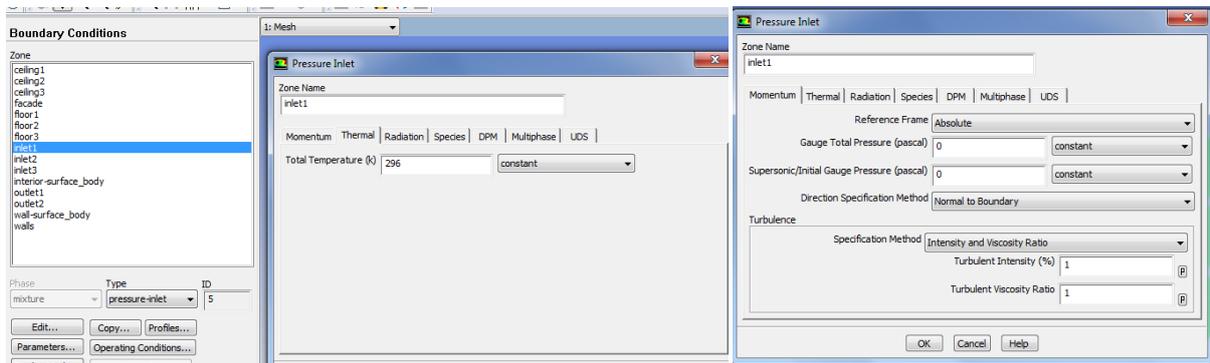


Figure 9. Boundary Conditions Setup for Buoyancy-driven Ventilation

For the wall type boundary remain as default except for floors, ceilings, and the façade. A change should be done in Thermal tab of their conditions. Floors and façade must have heat flux at 40 W/m² for floors and 44 W/m² for façade, and ceilings must have temperature of 22°C.

d. *Monitors*

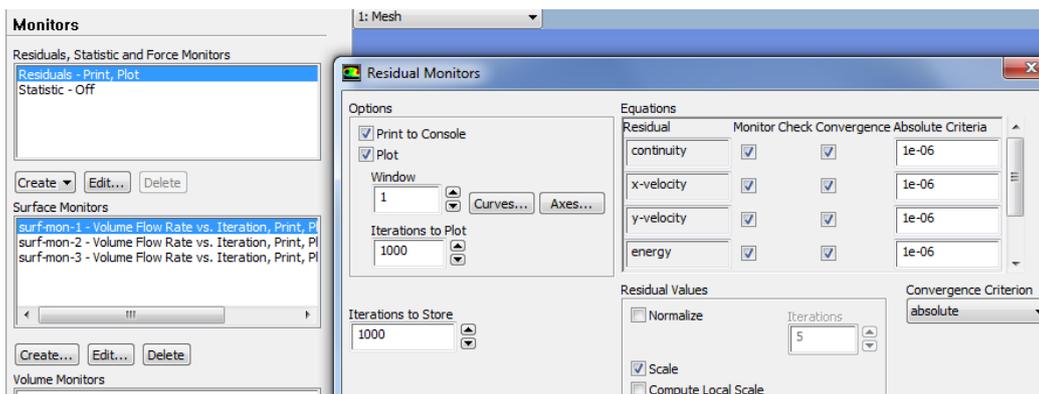


Figure 10. Monitors Setup for Buoyancy-driven Ventilation

- e. *Running calculation & mesh independence study.* The calculation is started from 3000 iterations. In order to get reliable results, a mesh independence study should be done so it is obvious that the solution independent of the grid used, as it has been said in Meshing explanation. In mesh independence study the calculation is repeated for the refined mesh. The results of each calculation are then compared to see if it changed with the change of the mesh or not. If the results from refined mesh do not change too much, it can be said that the previous mesh is good enough for calculation.

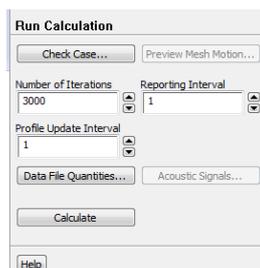


Figure 11. Iteration Number for Calculation

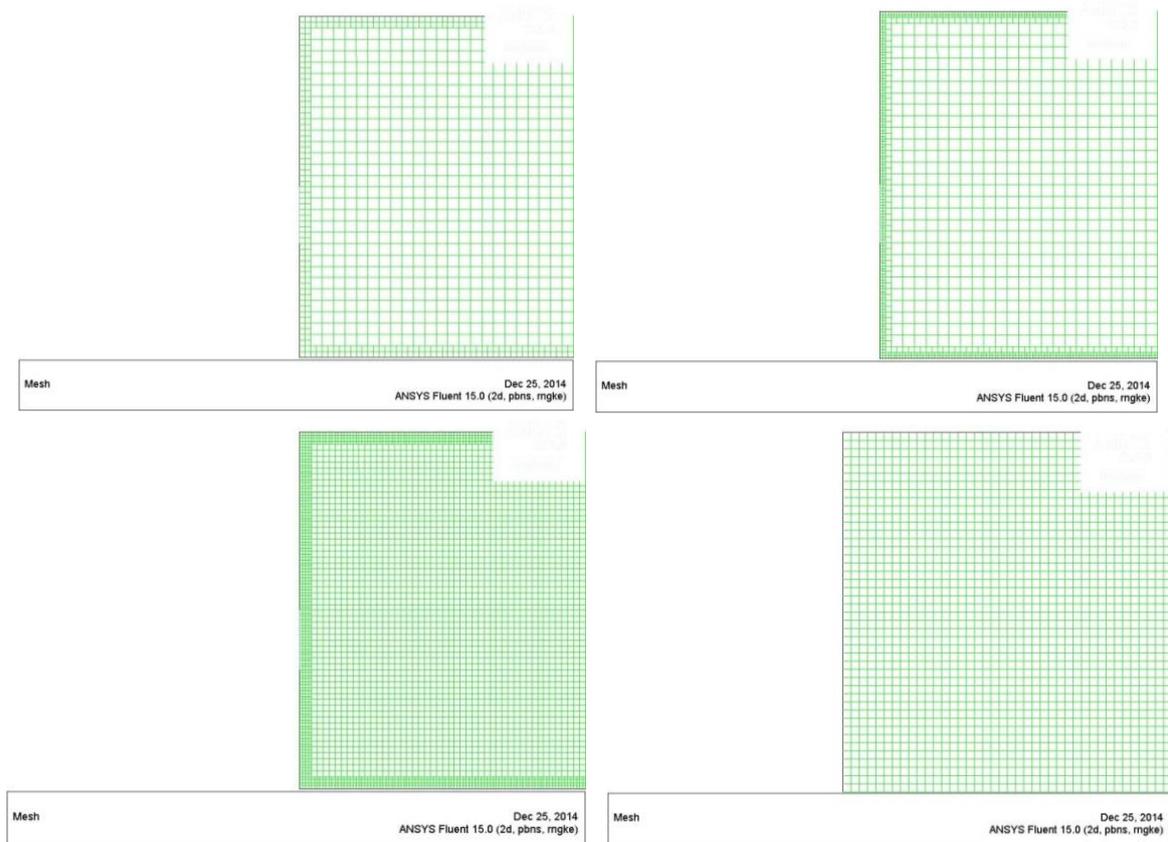


Figure 12. Four Mesh after three times refinements to compare

Three refinements have been done and the results have been compared. From the comparison, it can be concluded that mesh from the first refinement (the second mesh) is good enough for the simulation. The difference between second, third, and fourth mesh is negligible, hence the calculation results from the second mesh are reliable and the mesh can be used for further simulation in Case 2.

Table 1. Mesh Independence Study

Original Mesh		DIFFEREN PERCENTAGE	1st refinement		DIFFEREN PERCENTAGE	2nd refinement		DIFFEREN PERCENTAGE	3rd refinement	
Volumetric Flow Rate (m3/s)			Volumetric Flow Rate (m3/s)			Volumetric Flow Rate (m3/s)			Volumetric Flow Rate (m3/s)	
inlet1	0.1778064	24%	inlet1	0.22567962	-1%	inlet1	0.2234363	0%	inlet1	0.22286837
inlet2	0.2251427	-1%	inlet2	0.22180019	-2%	inlet2	0.21738662	-1%	inlet2	0.21478359
inlet3	0.1602695	23%	inlet3	0.20269775	-4%	inlet3	0.19434042	-2%	inlet3	0.18978833
outlet1	-0.498686	-36%	outlet 1	0.34516385	-3%	outlet 1	0.33644882	2%	outlet 1	0.34230977
outlet2	-0.064634	130%	outlet 2	0.30500937	-2%	outlet 2	0.29871446	-5%	outlet 2	0.28512427
Net	-0.000102		Net	4.31E-06		Net	1.91E-08		Net	6.22E-06

2.2.4. *Change in Setup for Case 2.* For Case 2, boundary conditions had to change. Temperature and floor heat flux have been changed to the condition as shown in table below.

Table 2. Case for Case 2

Time	Outdoor air temperature (°C)	Floor Heat Flux (W/m ²)
Early evening	20	20
Late evening	19	15
Midnight	17	10
After Midnight	14	5

3. Results & Discussion

3.1. Case 1

From figure 21 can be known that warmer air appears near the heat flux sources, such as floor and façade. Cooler air appears from ceilings, shows that the PCM is working to cooling the room down. First floor appears as the warmest floor of the building, while the third floor is the coldest floor. Average tower's temperature is even more than the temperature of the room because of heat flux from the façade which is more than the floor heat flux. It also has a temperature differences at the different height. Warm air concentrates at the top of the tower and cooler air concentrates at the bottom part of the tower because warm air is lighter than cold air.

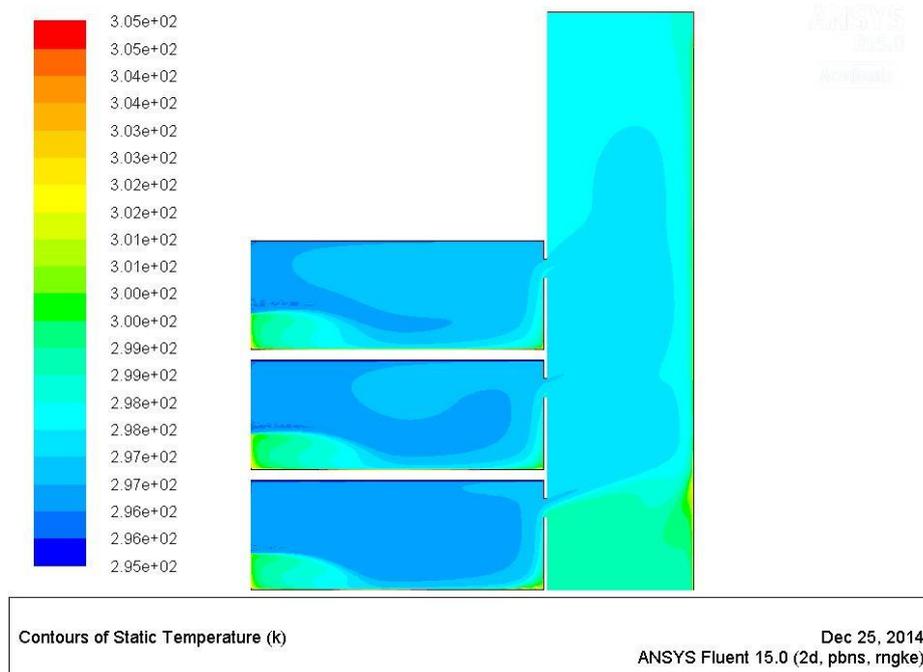


Figure 13. Temperature Simulation Result Case 1

Temperature differences within the tower itself and between the tower and the room lead to buoyancy effect. From Figure 21 it can be seen that the wind velocity is higher for the lower floor. It can be stated that the larger differences in temperature between two areas means larger buoyancy effect. With Case 1 boundary conditions, the building can provide 0.65 m³/s or 650L/s fresh air. With CIBSE indoor air quality standard of “moderate” level, the building could accommodate up to 100 persons.

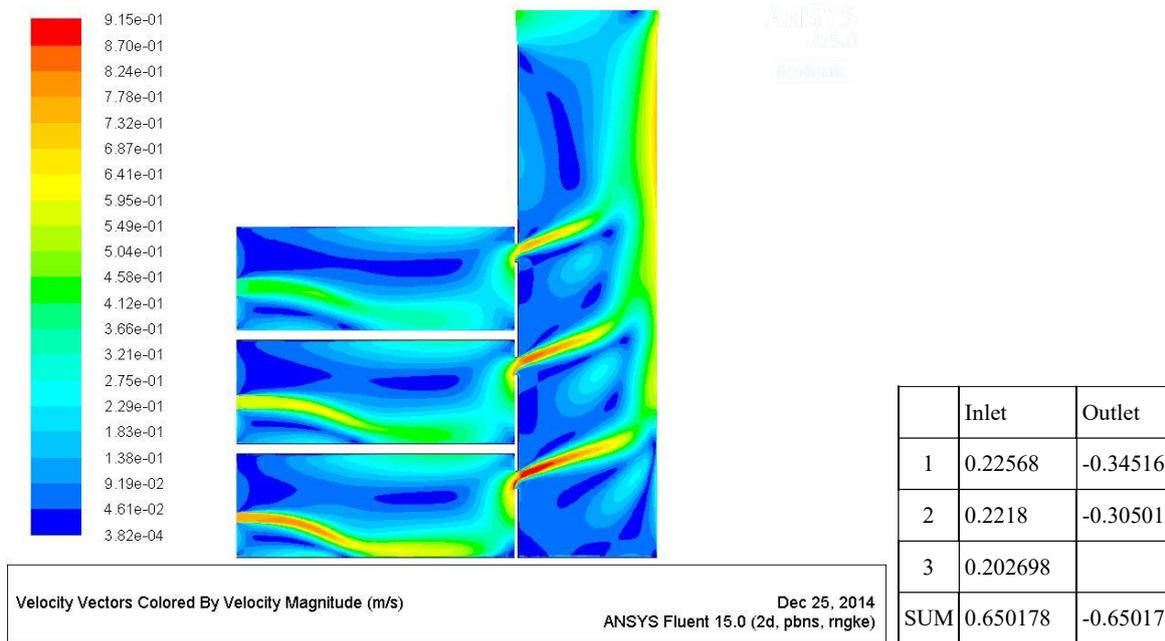
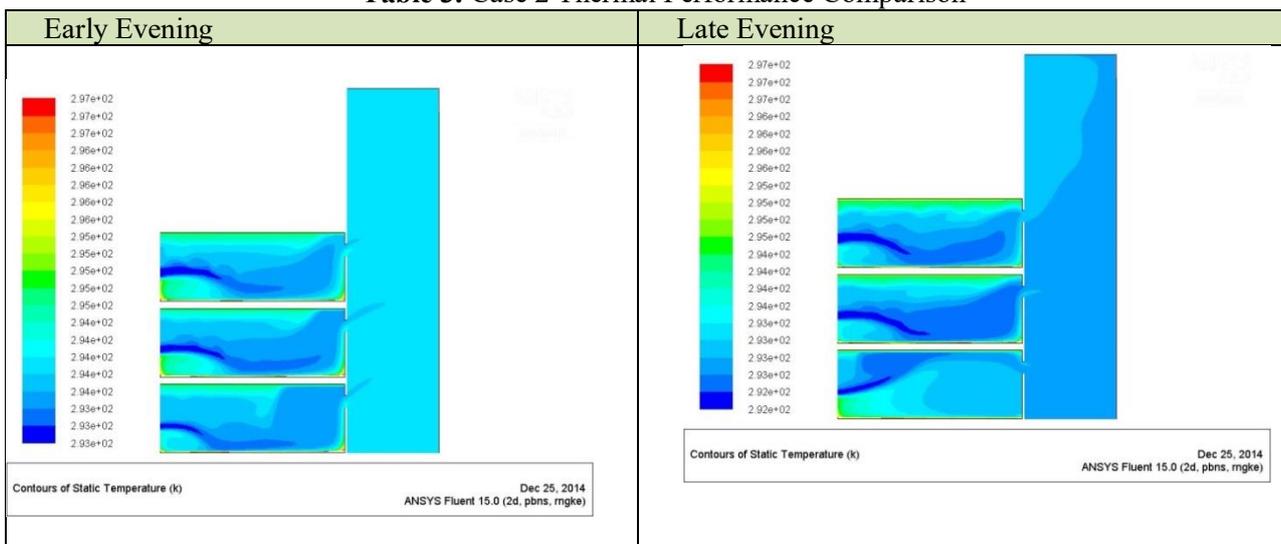


Figure 14. Velocity simulation result Case 1 and table of volumetric flow rate of the building

3.2. Case 2

Simulation of Case 2 in ANSYS Fluent shows the ventilation rate of the building and convection heat transfer rate of the ceiling. Table 6 and Figure 15150 shows the ventilation rate of the building in different time at night. Midnight appears as the time when the building has the lowest ventilation rate. However, the differences of each time not to large. But, if they are compared to ventilation rate in the afternoon, ventilation rate during night is lower. It may happen because the tower does not get heat flux from the sun during the night. As it is explained before, temperature differences caused by the heat flux from the façade accelerate the buoyancy effect.

Table 3. Case 2 Thermal Performance Comparison



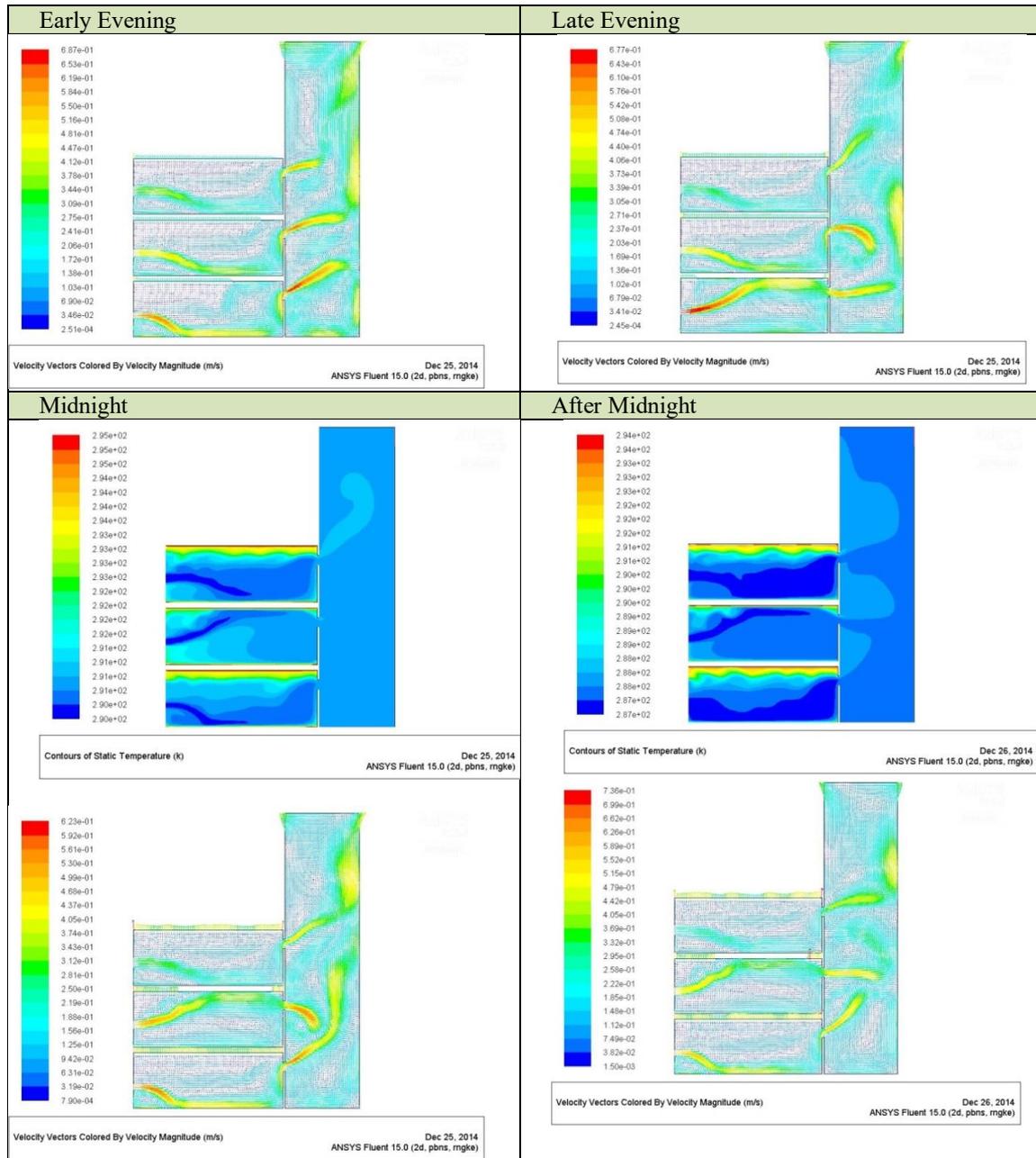


Table 4. Volume Flow Rate (m3/s) Case 2

	early evening		late evening		midnight		after midnight	
	Inlet	Outlet	Inlet	Outlet	Inlet	Outlet	Inlet	Outlet
1	0.1737	-0.2768	0.1948	-0.2801	0.1526	-0.2689	0.1576	-0.2702
2	0.1729	-0.2264	0.1785	-0.2302	0.1950	-0.2191	0.1992	-0.2395
3	0.1564		0.1369		0.1404		0.1528	
SUM	0.5030	-0.5032	0.5102	-0.5103	0.4880	-0.4879	0.5096	-0.5097

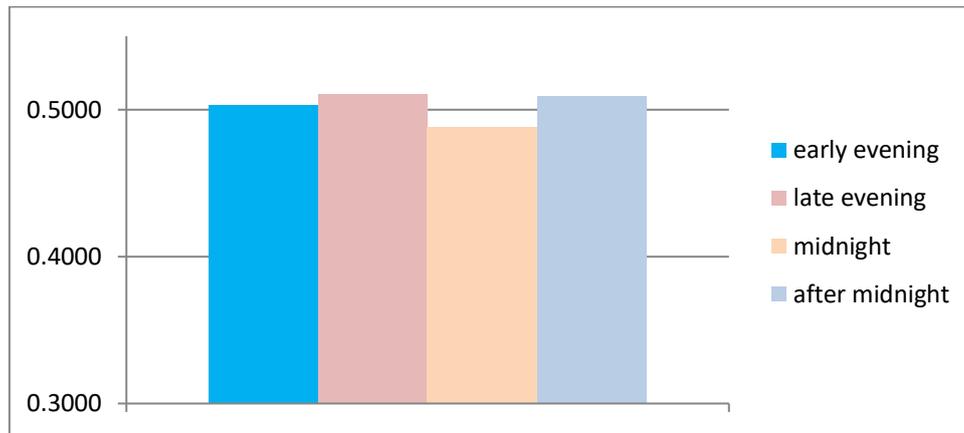


Figure 15. Volume Rate Ratio Comparison

Table 6 and Figure 23 show the convection heat transfer rate of the ceilings. Heat transferred from ceilings increase as the night goes over. It is due to temperature at the late night is lower, and the phase change material in ceilings has to adapt its surroundings. Larger temperature differences mean the larger heat flux from the ceilings.

Table 5. Total Surface Heat Flux (W/m²)

	early evening	late evening	midnight	after midnight
ceiling1	2.01595	6.914418	5.718842	11.30296
ceiling2	1.634324	2.507234	9.399104	14.97938
ceiling3	1.29899	2.313789	5.554428	11.66279

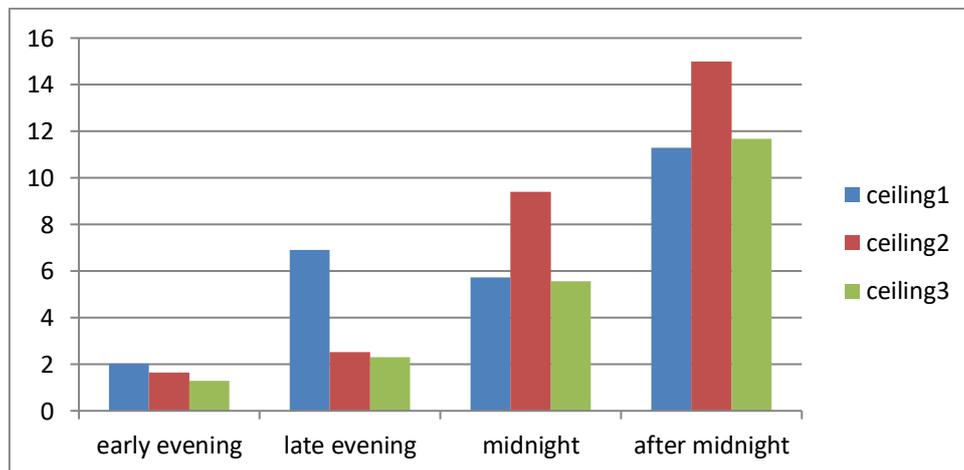


Figure 16. Total Heat Flux from Ceilings

4. Conclusion

Solar chimney is known as one of strategy for passive cooling has been simulated, and the result shows how buoyancy effect is used to drive ventilation with solar chimney strategy. Cooler air appears from ceilings and creates cooling to the room down. Meanwhile, warm air concentrates at the top of the chimney and cooler air concentrates at the bottom part of the chimney generates wind flow across the building. Temperature differences within the chimney itself and between the chimney and the room lead to buoyancy effect.

It is clear that the larger the temperature differences between the chimney and the room, the the larger buoyancy effect. With Case 1 boundary conditions, the building can provide 0.65 m³/s or 650L/s fresh air. With CIBSE indoor air quality standard of “moderate” level, the building could accommodate up to 100 persons.

Furthermore, midnight appears as the time when the building has the lowest ventilation rate. However, the differences of each time not to large. It may happen because the chimney does not get heat flux from the sun during the night. As it is explained before, temperature differences caused by the heat flux from the façade accelerate the buoyancy effect.

References

- [1] Chassignet, Eric, P, et. al. (2010) « Buoyancy –Driven Flows, Cambridge University Press, Cambridge
- [2] CIBSE (2008) *CIBSE Concise Handbook*, Page Bros, Norwich
- [3] Olgyay, Victor (1973), *Design With Climate: Bioclimatic Approach to Architectural Regionalism*, Princeton University Press, New Jersey
- [4] Chitty, Richard, Cao, Chunli (2014) *Computational Fluid Dynamics In Building Design: An Introductory Guide*, IHS BRE Press, Berkshire