CFD analysis of the classroom thermal conditions built from the sandwich panel

R N laubi* and N K Abbood1
1* Basra University for Oil and Gas, College of Oil and Gas Engineering, Basra, Iraq
Email: enrashan@gmail.com

Abstract. Based on the daily source energy radiation on hot summer zone like Iraq, lecture room built from sandwich wall panel (SWP) is analysed by using Computational Fluid Dynamics (CFD) technique. An air conditioning unit is operated inside a lecture room. The two targets of this study were to realize the influence of air condition location - Four cases for lecture room utilized with the same dimensions at a different position for the air conditioner unit. A thermal condition of the lecture room was optimized with ANSYS FLUENT. Numerical analyses have shown that there is a relationship between the position of air-conditioner and the distribution of convection currents in the room. However, the mass-weighted average temperature in the room almost looks like they are identical in all the cases, but differences in static temperature are observed at the different zone for the same case w.r.t the position of the air-conditioner.

1. General introduction

The crowd of University students settle most of their time in lecture rooms through lecture or tutorial. Air stream delivery from the air-conditioning system becomes significant at this study, [1] Heat transfer and fluid flow in rooms has been widely studied both experimentally and numerically due to the great scientific interest in many responses to energy-related applications. Air conditioners are preparation, intended, to settle down the air temperature and humidity inside a close area. It is utilized for cooling as well as heating. CFD is a tool wherein the governing equations for fluid flows and heat transfer are solved in discrete shape on computers by way of simulating the fluid glide trouble. The numerical simulation is appropriate for parametric studies and it also offers accurate consequences through solving governing equations in each cellular of the fluid area [2]. Some of the sandwich panel’s characteristic in structure is its low value of thermal conductivity coefficient. Our study target to clarify the thermal behaviour of fluid in a classroom built from the sandwich panel which reducing heat transmission, in 2010 Mohamed Mahdy(et-al) They use a lightweight concrete in sandwich panel core which gives low thermal conductivity compared with normal-weight concrete one[3]. In 2011 . S. Verma (et-al) utilize CFD to predict and optimizing the execution of air conditioning equipment. They indicate that there are several recirculation zones in the evaporator compartment [4]In 2015 Vaseemul Rahaman(et-al) introduced a CFD Analysis of Air Conditioning in CAD Lab they investigated that the chilly roof has interest such a low vertical air inclination and that the Thermal comfort is gained, even for higher metabolism average or covering, insulation besides the solar radiation function a considerable service in the extremely-glazed room, even if only the diffuse portion of solar irradiance was a treat in the analysis. [5]. In 2016 S.Sarma and Jakhar choose the optimized location of the air-conditioner where they concentrate on the numerical discussion of the temperature domain and airflow inside chamber a during summer with the various location of the air-conditioner unit by CFD the result show that energy consumption minimum while the AC is placed in the west wall during summer and highest while air-
conditioner is placed in the north wall [6]. Thermal insulation of Iraqi buildings by adding an insulating multilayer inside the wall is introduced by Dhafer M. Hachim and Qahtan A. Abed in 2017. Where they design a new sandwich wall panel that promotes thermal isolation of Iraqi buildings by supplement, an insulating multilayer inside the wall is presented by, they found that the proposed sandwich wall panel can effectively conserve, a large amount of energy exhaustion in terms of electricity adjust on heating and cooling[7] in (2017) Christian Suárez (et-al) introduced a study CFD used in air delivery in a Railway carriage they investigated that air delivery in the occupant cell is a very influential factor to control temperature and airspeed to check, thermal comfort. A CFD pattern including the car’s geometry in specifies, the passengers, the Lanterns and other the influential lineaments, associated to the HVAC system [8] In (2019) numerical study presents by Saleh Nabi (et-al) where a coupled simulation to discuss the minimize performance of a room air conditioning system in Madelia, Dynamic characteristics correspond to several vane angles and airflow modes. The result shows that both parameter demonstration powerful influence, on the pull-down time [9].

2. Problem definition and governing equations

2.1. Modelling of classroom design and mesh choosing

The first step in simulation is a drawing three-dimensional geometry using DESIGN MODELER. So in the present study, the model consists of the 3D classroom (5m×3m×10m) which made from the sandwich panel with an air condition units located at the different position, are showing in figure1 DESIGN MODELER software includes large abilities to draw the simple and complex geometries The meshes generated are very influential part in simulation process because the mesh influences the accuracy, convergence and speed of the solution. In the finite volume method (FVM which refers to the small volume surrounding each node point on a mesh) volume integrals in a partial differential equation that include a divergence term are converted to surface integrals, using the divergence theorem the mesh type used. In the present study, the mesh used was tetrahedral. A mesh consisting entirely of triangular elements is referred to as a triangular mesh; a mesh with only tetrahedral elements is a tetrahedral mesh.

![Figure 1. Classroom design model with north, south, east and west position for air-condition.](image)
2.2. The grid independent
The grid-independent is illustrated in Table 1.

| Element size(cm) | Elements | Nodes | Area average weighted (K) |
|------------------|----------|-------|--------------------------|
| 14               | 504882   | 95478 | 296.8872                 |
| 15               | 359164   | 63045 | 296.8871                 |
| 16               | 298598   | 52806 | 296.8871                 |

2.3 Set up and Boundary condition
In numerical engineering procedure, the geometry involves specifics unnecessary for simulation. Only the physics interested is to be included, simulating such a completely itemized model will expansion solver work times. Heating of face due to radiation can be done in Computation Fluid Dynamics model utilize Discrete Transfer Radiation Model, or P-1 Radiation Model, or Rosseland Radiation Model, or Surface-to-Surface (S2S) Radiation Model, or Discrete Ordinates (DO) Radiation Model [10]. At the present work, the outdoor measurement parameters included environmental temperature $T_\infty$ assumed to be constant for four cases on the same day $T_\infty = 300K$ while the indoor measurement parameters inclusive temperature and velocity of output air from the air conditioner unit which also remains constant for all cases. To resolve and construe Cases that include fluid flows and heat transfer numerically the commercial software package ANSYS15 is applied. The model involves the control volume generation to explore the governing equations with the determined boundary conditions. Next to subdivide the domain into discrete CV formalization a computational grid or mesh; the governing equations are integrated across the individual CVs forming surface integrals by Gauss’ divergence theorem. The COUPLED algorithm has been used to realize the coupling between pressure and velocity with the PRESTO scheme for the pressure equation and the cell-face estimate of pressure could be gained from simple arithmetic averaging of centroid values. The features of the PRESTO scheme for pressure interpolation are obtainable for all meshes in FLUENT. With pressure factors of 0.3 momenta of 0.8 and 1.0 for energy applied for convergence of all the considered variables. Tetrahedral elements applied for the complete domain. The diffusion expression is discretized by the second-order scheme and convective expression is discretized with the second-order upwind scheme. The solution convergence criteria were set to $10^{-6}$ for continuity, Naive stocks equations and $10^{-8}$ for energy

3. Boundary condition

3.1. Thermal conditions
These can be classified by the following:
a) Roof, east and north surface subjected to mixed radiation and convection condition with external Emissivity (0.9) and h (22.666 W/(m²•K)). All the walls are said to have no-slip boundary conditions. Where Basra is located at latitude 30.5085201 and longitude 47.7803993, which input to FLUENT Setup.

b) The other two walls (floor and west wall ) were subjected to convection heat transfer with h=8.3 W/(m²•K),

c) The door with dimension (1m, 2m, 0.2m) is modelled by assuming it to be an isolated surface.

3.2. Flow conditions:
The air conditioner flow was maintained at constant velocity (5m/s) and constant temperature (292 k).

3.3. The outlet:
The outlet is maintained to be at ambient pressure.

Figure 3. The solar radiation boundary condition

4. Governing equation

The Continuity, X Momentum, Y Momentum, Z Momentum and Energy are showing below

\[ \frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) + \frac{\partial}{\partial z}(\rho w) = 0 \]

\[ \frac{\partial}{\partial x}(\rho u^2) + \frac{\partial}{\partial y}(\rho v u) + \frac{\partial}{\partial z}(\rho w u) = -\frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \]

\[ \frac{\partial}{\partial x}(\rho v^2) + \frac{\partial}{\partial y}(\rho v v) + \frac{\partial}{\partial z}(\rho w v) = -\frac{\partial P}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \]

\[ \frac{\partial}{\partial x}(\rho w^2) + \frac{\partial}{\partial y}(\rho v w) + \frac{\partial}{\partial z}(\rho w w) = -\frac{\partial P}{\partial z} + \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \]

The K-Epsilon turbulence model one of the quietest turbulence models was used as it supplies reasonable accuracy for an enormous range of turbulent flows. The turbulent kinetic energy k, and its rate of dissipation \( \varepsilon \), for this model, are gained by the following equations.

\[ \frac{\partial}{\partial x_i} \rho k u_i = \frac{\partial}{\partial x_i} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \]

\[ \frac{\partial}{\partial x_i} \rho \varepsilon u_i = \frac{\partial}{\partial x_i} \left[ (\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3k} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \]

where \( G_k \) symbolize the generation of turbulent kinetic energy the grow as a result of mean velocity gradients, \( G_b \) is the generation of turbulent kinetic energy that appears as a result of buoyancy, and \( Y_M \) symbolizes the fluctuating dilation incompressible turbulence that contributes to the overall dissipation rate. \( S_k \) and \( S_\varepsilon \) are source terms \( C_{1\varepsilon}, C_{2\varepsilon} \) and \( C_\mu \) Heat transfer by thermal radiation is extremely important to consider in many modeling cases such as the case being investigated here. The radiation effects from the sun’s rays that enter the computational domain can be calculated by the solar load model.
\[
\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s)I(\vec{r}, \vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^4 \Phi(\vec{s}, \vec{s}') ds'
\]

Where \( \vec{r} \) is position vector, \( \vec{s} \) is direction vector, \( \vec{s}' \) is scattering direction vector, \( a \) = absorption coefficient, \( n \) is the refractive index, \( \sigma_s \) is scattering coefficient, \( I \) is radiation intensity, which depends on position (and direction), \( \Phi \) is phase function, \( T \) is local temperature. The radiative heat transfer equation is illustrated below [11].

![Figure 4. The radiated heat transfer](image)

Fig. 4 illustrates the effect of air conditioning on thermal boundary distribution in the fluid domain, the results indicate that mean temperature gradient is affected by air-conditioning location so it is unsymmetrical along with all domino. The solar radiation effects that influence on the roof, east and north surfaces will cause an increase in air temperature and therefore, the buoyancy force tend to flow hotter layer upwards due to density gradients arising from temperature gradients in the air layers. This is due to a trace of the cold draft emanating from the air conditioner. All temperature profiles are changed steeply near the hot and cold walls, the temperature can be described by orderly change, and where temperatures increase linearly far away from the air conditioner. This region is called the inner layer of the boundary layer. It is interestingly noticed that the air currents near the air condition will be cooler than the other zoon. Fig. 6 shows the predicted temperature distribution contours at the mid -z plane of the classroom, the first case.
in this figure where air condition located in the north side where temperature increasing tends to be orderly far away from air condition unit, that results as same as when air condition located at the south side. On the other hand, more regularity is observed in the temperature gradient field in the other two cases when air condition located at the east and west side.

Figure 6. Temperature distribution counters for air-conditioning have a north, south-east and west air condition position respectively

Figure 7. Velocity distribution counters for air-conditioning have a north, south-east and west air condition position

For four cases the effects of various air conditioner position on velocity contours are represented in Fig. 7 the first case with air-condition located at the north side, Two primary zones formed inside the room, high-velocity reign being formed around the position of air conditioner, While modulated zone occupies the whole space of lecture room. At south situation velocity profile is as same as the north situation but with a small difference. Also observed that there is a stagnation zone at the top of the air conditioner due to no-slip boundary condition near the surface wall. In order to get a clear comparative Table 2 show the temperature of fluid layers, Average temperature remains lowest when the air-conditioner is mounted in the west wall, whereas highest temperature distribution is observed while air-conditioner is installed in the north wall. Figure 8a shows the variation of local temperature distribution along with
high of lecture room that affected by air condition unit The results show that the minimum value of temperature occurs near the middle y plane (y=1.5 m) which corresponding to cold air inlet for all cases, and its minimum value take place at position four (air condition located at the west wall). However, the behaviour of four cases gives a similar pattern along the y-direction, which distinguishes by linear decreasing in temperature whit closer to upper air conditioner slot. Figure 8b shows the variation velocity alone the y-direction, no significant change in velocity distribution occurs between the four cases. two primary apex in velocity curve was formed along the y-direction, due to high air velocity in y= 0.5 and y=1.5.

Figure 8. a) Local Temperature distribution along the high of lecture room with the high of lecture room. b) local velocity distribution along with the high of lecture room.

| Position of Air conditioner | Mass-weighted Average Temperature (k) |
|----------------------------|--------------------------------------|
| North wall                 | 296.237                              |
| South wall                 | 295.127                              |
| East wall                  | 296.843                              |
| West wall                  | 294.811                              |

In order to get a clear comparative Table 2 show the temperature of fluid layers, Mass-weighted average temperature remains lowest when the air-conditioner is mounted in the west wall, in comparison with the other four cases.

6. Conclusions

In present work, the numerical study has been conducted in three-dimensional mixed radiation and convection through the sandwich panel classroom, The main observations from this work:

1. The numerical results show that the overall heat transfer is slightly influenced by air-conditioning position.
2. Due to the effect of both solar radiation and air condition the temperature contours for all cases are unsymmetrical around the classroom.
3. The velocity cell propagation strength approximately non equally and extended over hot zoon in the classroom due to solar radiation.
REFERENCES

[1] Lim, S. C. J., Husain, A., & Tee, B. T. (2002). Simulation of airflow in lecture rooms.
[2] Kumar and Bartaria “cfd analysis of room with air conditioner by using ansys workbench” JETIR July 2018, Volume 5, Issue 7
[3] M. Mahdy, M. Imam, A. Tahwia, and O. Youssf, “study on heat transfer of sandwich panels” 26 May 2017
[4] Moukalled, F., Verma, S., & Darwish, M. (2011). The use of CFD for predicting and optimizing the performance of air conditioning equipment. International journal of heat and mass transfer, 54(1-3), 549-563.
[5] Prof. Shankar Kumar and Rahaman “CFD Analysis of Air Conditioning Equipment of CAD Lab for Enhancing its Performance, ISSN 2347 - 6435 Volume 4, Issue 3
[6] Sarma, S., & Jakhar, O. P. (2016). Computational Analysis of Impact of The Air-Conditioner Location On Temperature And Velocity Distribution In An Office-Room. International Research Journal of Engineering and Technology (IRJET), 3(09).
[7] Hachim, D. M., & Abed, Q. A. Thermal Analysis Of Light Weight Wall Made From Sandwich Panels In The Aspect Of Thermal Insulation Design For Sustainable Built Environment.
[8] Suárez, C., Iranzo, A., Salva, J. A., Tapia, E., Barea, G., & Guerra, J. (2017). Parametric investigation using computational fluid dynamics of the HVAC air distribution in a railway vehicle for representative weather and operating conditions. Energies, 10(8), 1074.
[9] Qiao, H., Han, X., Nabi, S., & Laughman, C. R. (2019, February). Coupled simulation of a room air-conditioner with cfd models for indoor environment. In Proceedings of the 13th International Modelica Conference, Regensburg, Germany, March 4–6, 2019 (No. 157). Linköping University Electronic Press.
[10] Ali, B. H., Gilani, S. I., & Al-Kayiem, H. H. (2014). A Three Dimensional Performance Analysis of a Developed Evacuated Tube Collector using a CFD Fluent Solar Load Model. In MATEC Web of Conferences (Vol. 13, p. 06011). EDP Sciences.
[11] Matt Cable. “An evaluation of turbulence models for the numerical study of forced and natural convective flow in Atria” (Doctoral dissertation,2009).