DOI: https://doi.org/10.24113/ijoscience.v8i7.488

CFD Simulation of HVAC System in a Building using ANSYS - Fluent

Imran Ansari
Research Scholar
Master of Technology
Dept. of Mechanical Engineering
Corporate Institute of Science & Technology
Bhopal, Madhya Pradesh, India
ansariimran396@gmail.com

Atul Shankar Suman
Assistant Professor
Dept. of Mechanical
Engineering Corporate Institute of Science & Technology
Bhopal, Madhya Pradesh, India
sumanatul302@gmail.com

Abstract: The aim of the study is to simulate the functionality of a HVAC system in different situations, summer and winter time, using specialized software ANSYS-Fluent. A 2D building model was realized and simulating the internal conditions represented the main elements of the study. There are studied the indoor air temperature and air velocity in different conditions. The results are presented as graphs/plots and spectra of interest parameters. HVAC system functionality simulation using ANSYS-Fluent is providing important results for the studied scenario.
The study aimed to simulate HVAC system functionality in different situations, summer and winter time, using specialized software ANSYS-Fluent. The realization of 2D building model and simulation of external and internal conditions it represents the main elements of simulation. As general conclusion, it can be stated with certainty that the recently implemented HVAC system reaches its task and provides adequate comfort conditions inside the amphitheater during both seasons.

The average velocities, are slightly bigger during summer season, due to higher airflows required. However, this effect doesn't affect the occupants, because they are lower than the comfort ones. HVAC system functionality simulation using ANSYS-Fluent is providing important results for the studied scenario. This type of analysis can be used for pre-examination of the future projects in order to obtain functional systems and verify the all requirements for the HVAC installation. The results can be used in further analysis for determining the correlations to the main comfort indicators, PMV and PPD.

Keywords: *CFD analysis; indoor climate; HVAC system; 2Dbuilding: numerical simulation; ANSYS Fluent*

1. Introduction of HVAC system

In the present technological culture, peoples spend more than 90% of their time in air-conditioned houses, offices and vehicle. These artificially created indoor surroundings have contributed both their benefits and disadvantages to the human beings.
Air-conditioning is defined as simultaneous control of temperature, humidity, cleanliness and air motion .The widely used terms in air-conditioning systems are discussed here. According to American Society of Heating, Refrigerating and Air-conditioning Engineers (ASHRAE) standards, the thermal comfort is "the condition of mind which expresses satisfaction with the thermal environment". As per ASHRAE standards 62, acceptable IAQ is "air in which there are no known contaminants at harmful concentrations as determined by cognizant authorities and with which a substantial majority (80% more) of people exposed do not express dissatisfaction".
Addition of moistness to the air, without variation in its dry bulb temperature is identified as humidification. Better comfort conditions and IAQ can be maintained through optimum design, appropriate location of supply diffusers and optimum ventilations supply rates using C02 based Demand Control Ventilation (DCV). According to

International Energy Agency (IEA), DCV is defined as a ventilation system where the air flow rate is governed by a sensor detecting humidity or airborne pollutants, in order to keep the concentration levels of the detected substances below a preset value.

1.1 Parameters for a comfortable environment

The best possible environmental conditions in an enclosure/cabin are achieved by designing
HVAC system which provides right degree of comfort. The factors which contribute mainly in creating a comfortable environment are:

- Airflow rate
- Air quality
- Temperature
- Mean Radiant Temperature (MRT)
- Relative humidity

Apart from these other controllable environmental factors are proper ambience and noise. At design stage an HVAC engineer has to consider maximum parameters which affect the thermal comfort because a greater degree of comfort can be achieved by controlling the above mentioned factors. In order to achieve the necessity of a client; one has to also include the identification and enforcement of good ventilation practice with consideration of cost. The best suited values of relative humidity and temperature which are required for maintaining comfortable indoor environments vary as per the climatic conditions.

Table 1 Composition of the fresh air

Components	% by Volume
Nitrogen	78
Oxygen	21
CO_2	0.03
Others	0.97

Table 2 Optimum temperature and relative humidity values for thermal comfort

Temperature	21 – 26° C
Relative humidity	40 - 60 %
Airflow speed	0.1 - 0.15 m/s

The appropriate value of the in-cabin temperature for thermal comfort is a function of climatic conditions. During winter season the optimum value of in-cabin temperature lies in between 21 – 23 ℃; while during summer it varies from 21 – 26 ℃.

The variation of RH with air temperature is presented in figure 1. It is observed that RH value decreases with increasing temperature and vice-versa. Although human beings feel comfortable in a wide range of RH with different air temperature, high relative humidity (over 60%) create the discomfort to passengers and also leads to condensation of water vapour on the windshields/glasses of car. It can also cause shorting of electrical components.

The passenger feels discomfort with RH lower than 40% and it causes a dry sensation, nosebleeds, and respiratory problems.

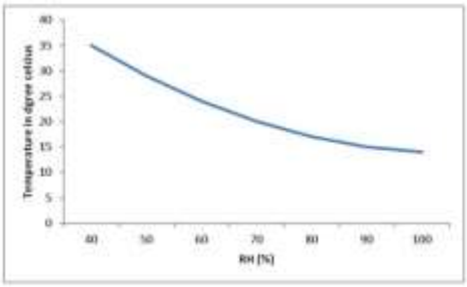

Figure 1 Relation between RH and temperature of air

1.2 Room Heat Gains

The heat gain components that contribute to the room-cooling load are indicated in Figure 2.

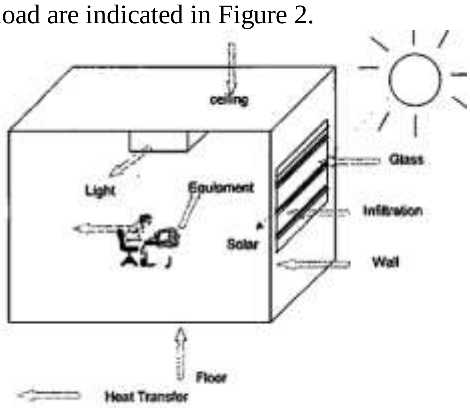

Figure 2. Components of cooling load

As shown, it consists of the following components.

1. Conduction through exterior walls, roofs, and glass
2. Conduction through interior partitions, ceilings and floors
3. Solar radiation through glass
4. Lighting
5. People
6. Equipment
7. Heat from infiltration of outside air through openings (windows, doors, cracks, etc.).These heat gains are categorized into two groups - those generated from external sources that is, from outside the room and those generated internally.

Items 1 through 3 and 7 are external heat gains and items 4 through 6 are internal heat gains. The heat gains are classified as sensible and latent heat gains. Sensible heat gains result in increasing the air temperature, whereas the latent heat gains are due to the addition of water vapour, which increase humidity. Items 1 through 4 are sensible heat gains. Items 5 and 7 are part sensible and part latent, and item 6 can fall in either category or both depending on the type of equipment.

2. Case description

The functionality of the HVAC system and further building services is a very significant objective for all type of buildings and even more for positions with high density of people. The present study is analyzing the HVAC system functionality, in steady state conditions, for a college amphitheater, being also available for a conference hall. The capacity of the audience is about 100 people, distributed as 12 people on 8 rows. Each row is placed on a higher step than previous one – Fig. 1.

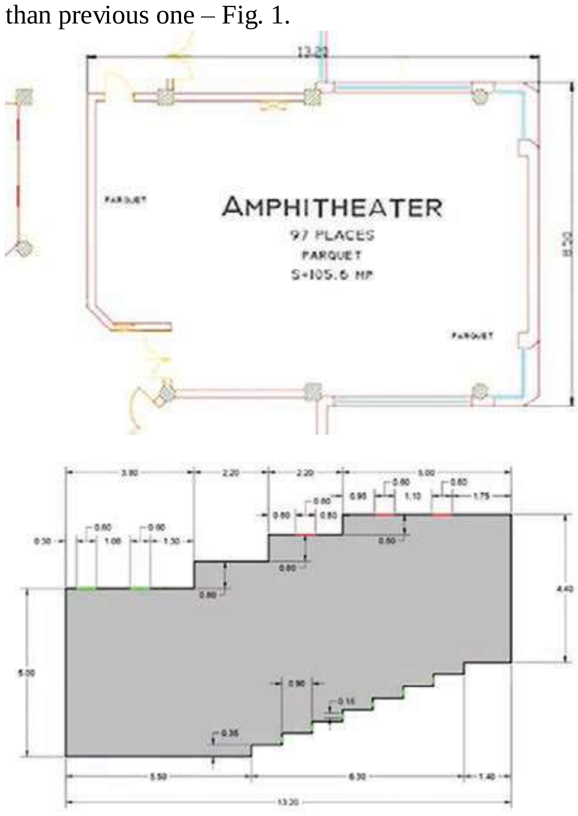

Figure 3 Geometry of the amphitheater a) Plane section; b) Longitudinal section

The preserved air is presented by the aid of a total 4 inlet grilles of 0.6mx0.6m placed at ceiling and 24 inlet grilles of 0.15mx0.5m placed at risers - Figure 3. The removal of the air is realized by 6 outlet grilles of 0.6mx0.6m placed at ceiling.

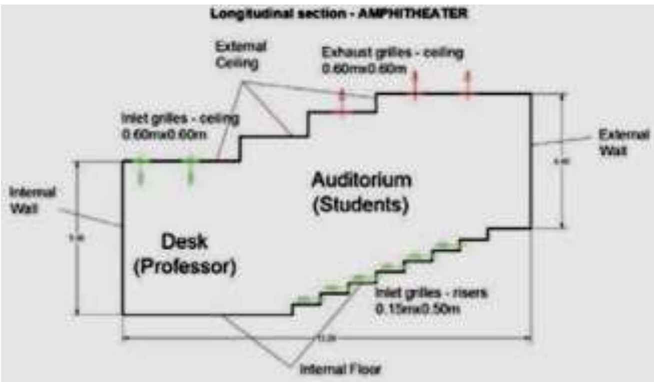

Figure 4 Longitudinal section – position of inlet and outlet grilles

3. Modeling HVAC system

The simulations are accomplished in steady state regime, by turbulent flow and k-ε model, suitable for estimation of airflow and heat transfer inside closed domains. The results obtained refers to the temperature and velocity data inside the amphitheater. Performing numerical simulations carried out by CFD tool, ANSYS-Fluent, the differential equations of heat transfer and fluid mechanics were resolved:

Conservation of mass or continuity equation:

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{v}) = S_m$$

Where S_m= mass added to the continuous phase or any user defined sources.

For 2D axisymmetric geometries, the continuity equation is given by

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\rho v_r}{r} = S_m$$

Where x is the axial coordinate, r is the radial coordinate, v_x is the axial velocity, and v_r is the radial velocity.

Momentum Conservation Equation

Conservation of momentum in an inertial reference frame is described by

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla.(\rho \vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F}$$

Where p= static pressure

$\bar{\bar{\tau}}$ = stress tensor,

$\rho \vec{g}$ = gravitational body force and

$\vec{F}$ = external body forces

The stress tensor $\bar{\bar{\tau}}$ is given by

$$\bar{\bar{\tau}} = \mu\left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3}\nabla.\vec{v}I\right]$$

where μ = molecular viscosity

I = unit tensor.

For 2D axisymmetric geometries, the axial and radial momentum conservation equations are

$$\frac{\partial}{\partial t}(\rho v_r) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_r) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_r)$$
$$= -\frac{\partial p}{\partial r} + \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(\frac{\partial v_r}{\partial x} + \frac{\partial v_x}{\partial r}\right)\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(2\frac{\partial v_r}{\partial r} - \frac{2}{3}(\nabla.\vec{v})\right)\right] - 2\mu\frac{v_r}{r^2}$$
$$+ \frac{2\mu}{3r}(\nabla.\vec{v}) + \rho\frac{v_z^2}{r} + F_r$$

Where

$$\nabla.\vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$$

Where v_x= Axial velocity

v_r= Radial velocity

v_z = swirl velocity

Energy Equation:

The energy equation for the mixture takes the following form:

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}(\alpha_k \rho_k E_k) + \nabla.\sum_{k=1}^{n}(\alpha_k \vec{v}_k(\rho_k E_k + p)) = \nabla.(k_{eff}\nabla T) + S_E$$

where k_{eff} = effective conductivity

S_E= volumetric heat sources

$$E_k = h_k - \frac{p}{\rho k} + \frac{v_k^2}{2}$$

Where

$E_k = h_k$ for an incompressible phase and h_k= sensible enthalpy for phase k

***k*−∈ model:**

The turbulence kinetic energy, k, and its rate of dissipation, ∈, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho \epsilon - Y_M + S_k$$

and

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

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients.

G_b is the generation of turbulence kinetic energy due to buoyancy.

Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.

$C_{1\epsilon}$, $C_{2\epsilon}$, and $C_{3\epsilon}$ are constant. σ_k and σ_ϵ are turbulent Prandtl numbers for k and∈.

S_k And S_ϵ are user-defined source terms.

The objective of the current study is to analyse the effect of the surface roughness on the hydraulic performance of the centrifugal pump. For the study of the above mentioned analysis a 2D CFD simulation is carried out by using ANSYS fluent 2020 R2. The CAD model, domain of fluid, meshing, applied boundary conditions and initial conditions will be discussed in detail.

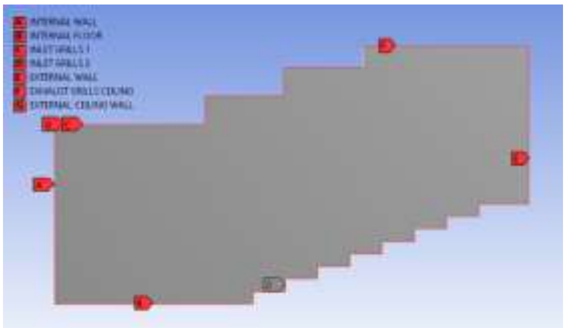

Figure 5 Name selection of fluid domain

Mesh Generation

The mesh generated in the domain quard mesh. For capturing the turbulence near the wall 7 inflation layers given domain model. For domain yzx mm element size is taken.

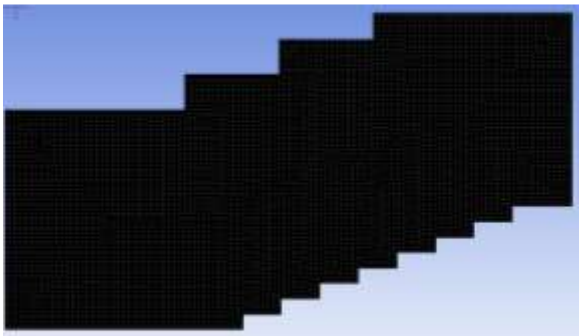

Figure 6 Mesh generation with standard settings.

Through the global mesh sizing settings, ANSYS Meshing® predictable that there were some curves around the body. However the meshing was very coarse and it was only the initial guess by the software. In order to capture more accurate data through solver we needed to improve the mesh. The first thing to do was changing the mesh sizing parameters & then change inflation layer setting.

Meshing of domain is done and total number of nodes and elements were found to be 121256 and 120446 respectively.

4. Results & Discussion

The computational results for the following cases are presented and discussed:

Table 3 Details of cases considered for present analysis is tabulated below:

Location	Variable	Case 1	Case 2	Case 3
Inlet Grills 1	Velocity Inlet	0.18	0.2	0.22
Inlet Grills 2	Velocity Inlet	0.15	0.167	0.18
External Ceiling Wall	Temp	44	44	44
Internal Wall	Temp	44	44	44
External Wall	Temp	44	44	44
Q(m^3/hr)		3500	3900	4300

All the results for different cases were obtained with the same meshing resolution, the same realization k-ϵ model, and also the same boundary conditions

4.1 CFD Analysis of Case 1

After performing CFD analysis using semi-empirical model k-ε. Velocity contour, pressure contour & velocity vector at different plane of case 1 (Inlet Grills 1- 0.18 m/s, Inlet Grills 2- 0.15m/s, External Ceiling Wall- 44℃, Internal Wall- 44℃, External Wall- 44℃, Q- 3500 m3/hr) as shown in figure below.

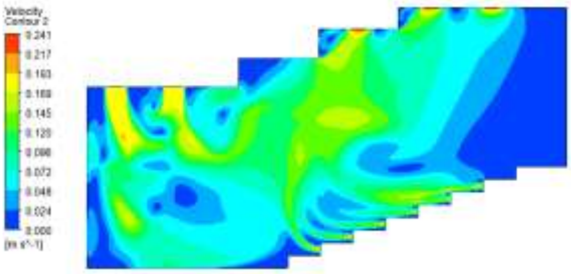

Figure 7 Velocity contour of case 1

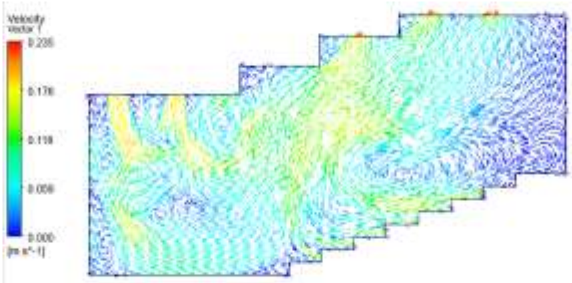

Figure 8 Velocity vector of case 1

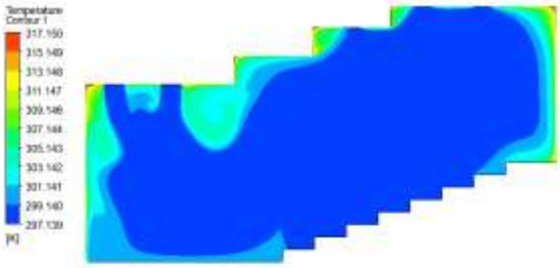

Figure 9 Temperature contour of case 1

The velocity contour in the fig 7 above clearly represents the velocity variation within the domain for inlet treated air velocity. It can be seen that velocity variation has maximum limit of 0.241 m/s which is at top where comfort is not much desired as per velocity value which further validates the scheme in terms of velocity. Fig 8 & 9 shows velocity vector & temperature contour respectively of case 1.

4.2 CFD Analysis of Case 2

After performing CFD analysis using semi-empirical model k-ε. Velocity contour, pressure contour & velocity vector at different plane of case 2 (Inlet Grills 1- 0.2 m/s, Inlet Grills 2- 0.167 m/s, External Ceiling Wall- 44℃, Internal Wall- 44℃, External Wall- 44℃, Q- 3900 m3/hr) as shown in figure below.

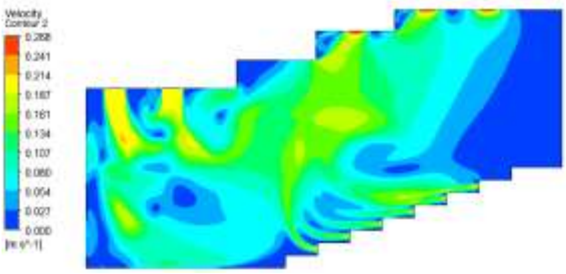

Figure 10 Velocity contour of case 2

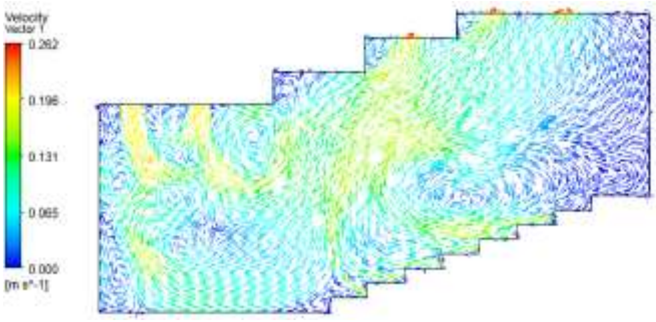

Figure 11 Velocity vector of case 2

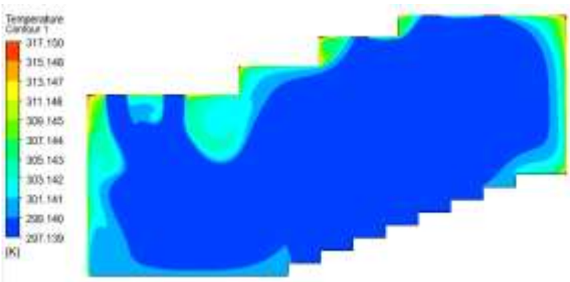

Figure 12 Temperature contour of case 2

The velocity contour in the fig 10 above clearly represents the velocity variation within the domain for inlet treated air velocity. It can be seen that velocity variation has maximum limit of 0.268 m/s which is at top where comfort is not much desired as per velocity value which further validates the scheme in terms of velocity. Fig 11 & 12 shows velocity vector & temperature contour respectively of case 2.

4.3 CFD Analysis of Case 3

After performing CFD analysis using semi-empirical model k-ε. Velocity contour, pressure contour & velocity vector at different plane of case 3 (Inlet Grills 1- 0.22 m/s, Inlet Grills 2- 0.18 m/s, External Ceiling Wall- 44°C, Internal Wall- 44°C, External Wall- 44°C, Q- 4300 m3/hr) as shown in figure below.

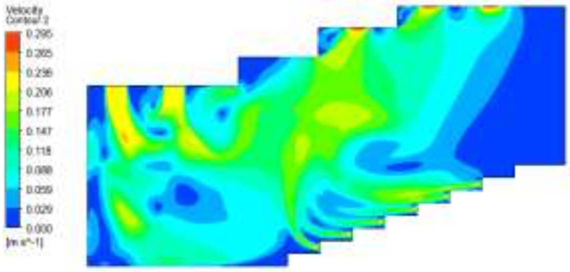

Figure 13 Velocity contour of case 3

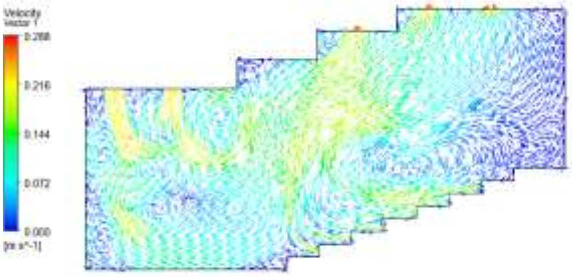

Figure 14 Velocity vector of case 3

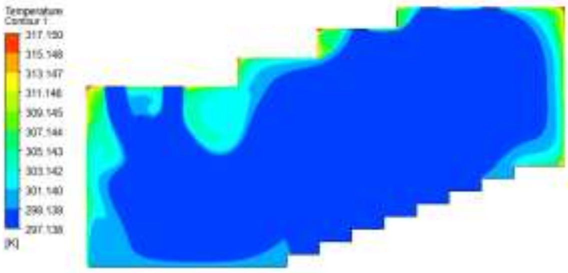

Figure 15 Temperature contour of case 3

The velocity contour in the fig 13 above clearly represents the velocity variation within the domain for inlet treated air velocity. It can be seen that velocity variation has maximum limit of 0.295 m/s which is at top where comfort is not much desired as per velocity value which further validates the scheme in terms of velocity. Fig 14 & 15 shows velocity vector & temperature contour respectively of case 3.

4.4 Influence of maximum velocity on different HVAC system

The graph shown above is plotted for highest velocity inlet case for different height of 1m, 2m, 3m and 4m as series height 1 to 4 respectively. It is clearly shown that the location at 4 m height has a maximum point of 0.22 m/s velocity which can be accepted for human comfort perspective.

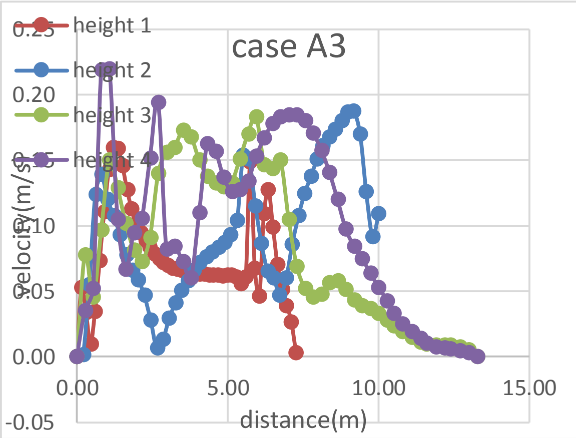

Figure 16 Plot for highest velocity inlet case for different height of 1m, 2m, 3m and 4m.

4.5 Influence of Discharge on different HVAC system performance parameters

Table 4 Values of different performance parameters at different discharge.

Case detail	Discharge (m^3/hr)	Inlet Velocity 1 (m/s)	Inlet Velocity 2 (m/s)
Case #1	3500	0.180	0.150
Case #2	3900	0.200	0.167
Case #3	4300	0.221	0.184

This study has mainly focused on to show the location of maximum velocity for human comfort perspective. In this study we have analysed three cases at different discharge rate. As we increase the discharge rate the velocity also increases at case 3 the discharge rate is 4300 (m^3/hr), find the maximum velocity which is 0.22 m/s. It is clearly shown that at the 4 m height maximum point of 0.22 m/s velocity which can be accepted for human comfort perspective.

5. Conclusions:

The study aimed to simulate HVAC system functionality in dissimilar circumstances, summer time, using specialized software ANSYS-Fluent. The realization of 2D building model and simulation of external and internal surroundings it signifies the chief elements of simulation.

As per common conclusion, it can be indicated with certainty that the newly applied HVAC system reaches its task and provides adequate comfort conditions inside the amphitheater during both seasons.

1. The average velocities, are somewhat larger during summer season, due to higher airflows required. Though, this effect doesn't affect the inhabitants, since they are lower than the comfort ones. HVAC system functionality simulation by means of ANSYS-Fluent is providing important results for the studied scenario.
2. If we increased discharge rate from 3100 to 3500 m3/hr (case 1) it is observed that as we increase the discharge rate the velocity is increased by 13.20% from 0.159 to 0.18 m/s.
3. If we increased discharge rate from 3500 to 3900 m3/hr (case 2) it is observed that as we increase the discharge rate the velocity is increased by 11.1% from 0.18 to 0.20 m/s.
4. If we increased discharge rate from 3900 to 4300 m3/hr (case 3) it is observed that as we increase the discharge rate, the velocity is increased by 10% from 0.20 to 0.22 m/s.
5. In case 3 at location of 4 m height, it has a maximum point of 0.22 m/s velocity which can be accepted for human comfort perspective. This results shows the most optimum result in current study.

This type of study can be used for pre-examination of the upcoming projects in order to obtain functional systems and prove the all necessities for the HVAC installation. The results can be used in advance analysis for determining the correlations to the main comfort indicators, PMV and PPD.

References

1. Cătălin George Popovici. HVAC system functionality simulation using ANSYS-Fluent, Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 October 2016, Bucharest, Romania; Energy Procedia 112 (2017) 360 – 365.
2. Yunqing Fan, Masaki Toyoshima, Makoto Saito, "Energy conservation and thermal environment analysis of room air conditioner with intermittent supply airflow", International Journal of Low-Carbon Technologies 13, 2018, 84–91.
3. Bamodu, Xia, Tang, "A numerical simulation of air distribution in an office room ventilated by 4-way cassette air conditioner", ICAE, Elsevier, 2016.
4. Ding, Guo, Chen, "Design and simulation of an air conditioning project in a hospital based on computational fluid dynamics", Archives of civil engineering, Volume 13, Issue 2, 2017.
5. DU, Lei, Chena, "The indoor thermal environment simulation and testing validation of a power plant turbine room in extreme cold area", 8th International Cold Climate HVAC Conference, Elsevier, 2016.
6. Satyam, Jagtap, Archana, "Design and Development of Portable Air Conditioner", International Journal for Research in Engineering Application & Management, Volume 02, Issue 7, 2016.
7. Sudhangshu Sarma, O. P. Jakhar,"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), Volume: 03, Issue: 09, Sep - 2016.
8. D prakash, "Transient analysis and improvement of indoor thermal comfort for an air-conditioned room with thermal insulations", Ain Shams Engineering Journal, Elsevier, 2015.
9. Thakur,Patel, Parth, "Quantification of Air Flow Pattern in Air Conditioned Room – A Review",

International Journal of Advance Engineering and Research Development Volume 1,Issue 12, December - 2014.
10. Calautit, Hughes, "Wind tunnel and CFD study of the natural ventilation performance of a commercial multi-directional wind tower", Building and Environment, Volume 80, October 2014, PP 71-83.
11. Mallikarjun, Malipatil, "CFD Analysis of Air-Cooled Condenser by Using Copper Tubes and Aluminum Fins", International Journal for Research in Applied Science & Engineering Technology, Volume 2 Issue X, October 2014.
12. Quadri, Jomon Jose, "Computational Analysis of Thermal Distribution within Passenger Car Cabin", International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME), Volume-2, Issue-2, 2013.
13. Taher, Mahmoud, Essam, "Numerical Investigation of Flow Patterns and Thermal Comfort in Air-Conditioned Lecture Rooms", International Journal of Mechanical and Mechatronics Engineering, Volume 7, Issue 5, 2013.
14. Hassan, Khan, Rasul, "Temperature monitoring and CFD Analysis of Data Centre", Procedia Engineering, Elsevier, 56, 2013, PP. 551 – 559.
15. Sihwan Lee, Mai Nogami, Satomi Yamaguchi, "Evaluation of Heat Transfer Coefficients In Various Air conditioning Modes By Using Thermal Manikin", 13th Conference of International Building Performance Simulation Association, 2013
16. Tengfang T. Xu, Francois R. Carrie, Darryl J Dickerhoff , William J. Fisk.―Performance of thermal distribution systems in large commercial building‖. Building and Environment 34 (2002) 215-226.
17. M. Krajcík, A. Simonea, B. W. Olesena. Air distributionand ventilation effectiveness in an occupied room heated by warm air‖. Energy and Buildings 55 (2012) 94–101.
18. .L. Pang, J. Xu, L. Fang, M. Gong, H. Zhang, Y. Zhang. Evaluation of an improved air distribution system for aircraft cabin. Building and Environment 59 (2013) 145-152
19. Ramponi R, Blocken B. CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. Building and Environment (2012)53: 34-48.
20. Rameshkumar et al, CFD Analysis of Air Flow and Temperature Distribution in an Air Conditioned Car, International Refereed Journal of Engineering and Science, (2013), 2(4) , pp. 01-06.
21. M Raman ME, Computational fluid dynamics Analysis of HVAC system in auditorium, International Journal of Advanced Research and Development, vol. 1(2016), pp. 68-72.
22. Alireza Kermani, CFD modeling for ventilation system of a hospital Room. Excerpt from the Proceedings of the 2015 COMSOL Conference in Boston
23. T.T. Chow, Z. Lin, Q.W.Wang and J.W.Z. Lu, Studying Thermal Performance of Split-Type Air-Conditioners at Building Re-Entrant via Computer Simulation, City University of Hong Kong Tat Chee Avenue, Kowloon Tong, Hong Kong (2011), 31(2).