

PAPER • OPEN ACCESS

Turbulent thermal characteristics and air flow patterns in a naturally ventilated tunnel with roof openings

To cite this article: Ashley A. Mason and R. Harish 2020 *J. Phys.: Conf. Ser.* **1716** 012026

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

A promotional banner for the 240th ECS Meeting. The banner features a colorful striped border at the top. On the left, the ECS logo is displayed in a green circle. To its right, the text reads "240th ECS Meeting" in large blue font, followed by "Oct 10-14, 2021, Orlando, Florida" in a smaller blue font. Below this, it says "Register early and save up to 20% on registration costs" in bold black text, and "Early registration deadline Sep 13" in a smaller black font. At the bottom left, there is a red "REGISTER NOW" button. On the right side of the banner, there is a photograph of a diverse group of people in a professional setting, with a man in a white shirt and tie clapping and smiling.

ECS **240th ECS Meeting**
Oct 10-14, 2021, Orlando, Florida
**Register early and save
up to 20% on registration costs**
Early registration deadline Sep 13
REGISTER NOW

Turbulent thermal characteristics and air flow patterns in a naturally ventilated tunnel with roof openings

Ashley A. Mason¹, R. Harish^{1*}

School of Mechanical Engineering, Vellore Institute of Technology, Chennai, India

Email: ^{1*} harish.r@vit.ac.in,

Abstract: Air flow is a major characteristic in the tunnel ventilation design, to avoid severe disasters like smoke suffocation problems in tunnel and fire accidents. Tunnels are best ways to reduce distance between two adjacent places. It may be underground or above ground, depending upon the usage of it. There are different types of tunnel ventilation, e.g. roof vents, side vents, mechanical ventilation, naturally ventilated etc. This paper majorly focuses on roof vents mass extraction with different sizes of the heat sources and it also records the horizontal and normal velocity for different sizes of heat sources. The Reynolds Number is low but still turbulence occurs when we deal with buoyant smoke keeping density as incompressible ideal gas. Ratio of height by width is taken into account and the transient air flow is simulated for 10 seconds, and mass flow is recorded at each time step of 0.1 seconds using Large Eddy turbulence Model to capture the flow physics. It is observed that with changing the heat source size, mass flow rate from each vent changes significantly and it drastically affects the air flow patterns and heat transfer characteristics inside the tunnel.

1. Introduction

As with growing population and demands, human workers are also needed more for analysis purpose or etc. E.g. taking a scenario of office, due to excessive heat produced by computer systems and heat released by human body, roof vents were needed to extract the heat of the room for optimum work condition of human workers and prevent working loss. Guo et al., [1] did a case study on optimizing of roof and night ventilation in office buildings in China. Their study majorly was for summer season to evaluate the thermal energy produced and devise an energy saving method for cooling it at night using roof vents. They used a model of six story building and considered different parameters like ventilation rate, air exchange rate and thermal mass level. Tomohiro et al, [2] did a research on ways to reduce the load for cooling purpose in summers. He focuses more on architectural feature for crowded places where pressure differences are low. His research promoted natural roof vents for air flow in crowded areas. Ameer et al., [3]. They used a C.F.D. approach and wind tunnel testing methods for evaluating different roof configurations for roof ventilations, using them as naturally ventilated vents. They concluded that roof vents are strongly influenced by wind velocity and air distribution method in the room.

Shaohua et al. [4], did a research on natural vents that were termed as hybrid as it contained both longitudinal as well as natural roof venting for getting a control over the smoke flow or fire inside a tunnel. It is the most effective method as it uses a hybrid method for smoke extraction method. They found out that the control over smoke is possible as well as its extraction is possible and the downstream temperature can be significantly decreased by using this methodology. Similar experiments can be seen from [5-9] on horizontal vent system for removing the rising smoke and cleanse the air. Zhong, et al. [10] did a study on the ventilations that are concerned with vertical shaft, that was investigated numerically by L.E.S. model. They aimed at studying the problem of smoke flow and tunnel fire with the combined effect and function of longitudinal wind, taking into account the effect of stack effect. They concluded that at high longitudinal velocity, the required force needed to drive out or exhaust smoke is weak. There is a specific value, also be called as critical value for the wind velocity, for which an optimum smoke exhaustion phenomenon is obtained.

Yang Ying Xing et al, [11] did their study on the various dimensions of dampers for propagation of smoke phenomenon in a naturally ventilated tunnel with vertical openings. The effect the harmful gas, majorly CO was taken into account and its marching behavior throughout the longitudinal direction of tunnel was



observed and how effective is the working of the dampers and vents. Karol et al., [12] performed their study on a check for a ventilation system operation of a road tunnel located in Southern Poland. They used an experimental approach by using anemometers at different place and recording different parameters at inlet and outlet for normal conditions and also for worst case scenarios. They concluded that the combination of the stack effect with the wind flow is very crucial for ventilation system. Similar studies [13-15] were reported with vertical opening for ventilation with respect to heat source location. Polat et al.

[16] did a numerical study on conjugate heat transfer on different modes of heat transfer like conduction and natural convection by solving different governing equations that govern the flow by numerically solving them. He calculated different isotherms and heat transfer by taking different aspect ratios of heated body (height / length). He realized that volume is an increasing function of area, aspect ratio, Rayleigh number. Jian Gong et al. [17] did a study on smoke flow to know whether the pattern or behavior of smoke marching will have an effect if the position and size of the heat source is varied. From their study they concluded that the floor as a fire source produced the largest smoke. They also concluded that smoke marching is effect by initial conditions and also ambient conditions are critically important for now the smoke flow behavior and also for time of evacuation. Lee et al. [18] carried out a numerical study to find a dependency of smoke flow with different aspect ratio of heat source. The simulations were carried out using F.D.S. Their research concluded that smoke deposition is maximum in the area below the ceiling. The final observation was a rate of decrease in the temperature due to reduced heat loss in longitudinal direction.

Ji et al. [19] studied the smoke distribution in near field of fire source. From their study, it was concluded that tunnel length also affected the heat and smoke distribution. They observed that smoke present on the lower side of ceiling slowly by slowly decreases both at longitudinal as well as transverse direction. The tunnel Size affects the mass flow rate and also the maximum temperature that can be reached by smoke accumulation. [20-32] analyzed the study parameters for aspect ratio of heat source using different vent, size of heat source, heat source location and different wind conditions. It is also observed that under the turbulence condition as it is a creeping flow still turbulence occurs in the governing flow [32-38]. Different turbulence models were used to solve N.V. equations. Jin et al., [38], observed that even multiple vents in tunnel in the country, China theoretical basis of tunnel for designing a tunnel ventilation system was incomplete. They realized that a governing equation for energy and an equation for natural vent were required for natural ventilation for calculating the drag flow of air. At the end of their experimentation, they found out that maximum air flow velocity is near tunnel entrance and decreases gradually along the tunnel, and along the direction of decreasing flow velocity, it makes a concave curve.

Rajan [39] did a research on a turbulent flow, that is generated from a heat source block which is inside a naturally ventilated cube shaped enclosure with different openings, one at the top and one at a vertical position. His results indicated the instability in the flow that is developed in the enclosure, especially at the vertical section of the opening was significantly higher than the horizontal opening. Rajan et al., [40] analyzed the performance roof vents and its effect with respect to tunnel fire keeping its flow characteristics as a concern.

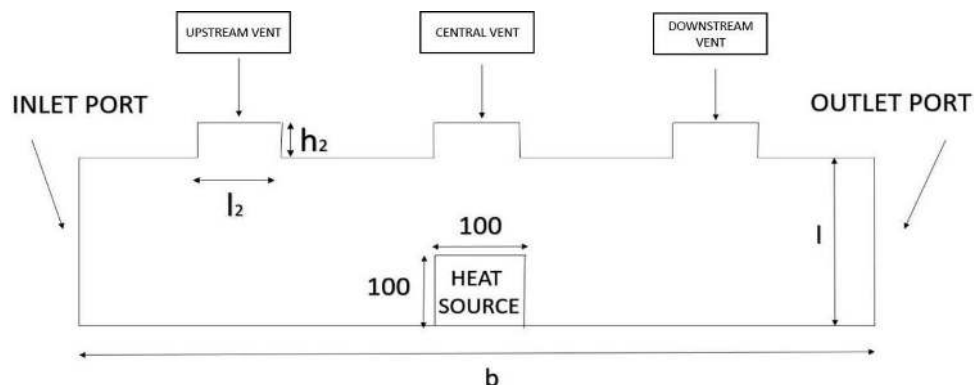


Figure 1. A 2D Schematic Diagram and Dimensions for CFD analysis (All dimensions in mm)

Their analysis contained a LES model for better results and they also varied the size of heat source for better flow physics visualization for better ventilation system. They concluded that the vent location with

respect to the heat source significantly affects the performance of the vent and also the smoke flow behavior.

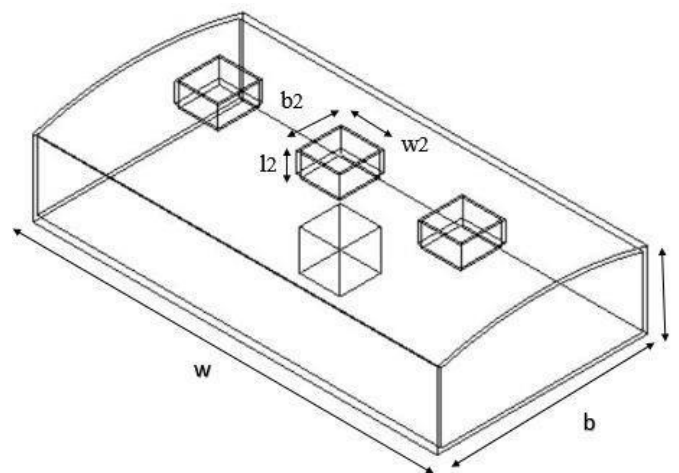


Figure 2. 3D Schematic Diagram for CFD analysis shown in Isometric projection

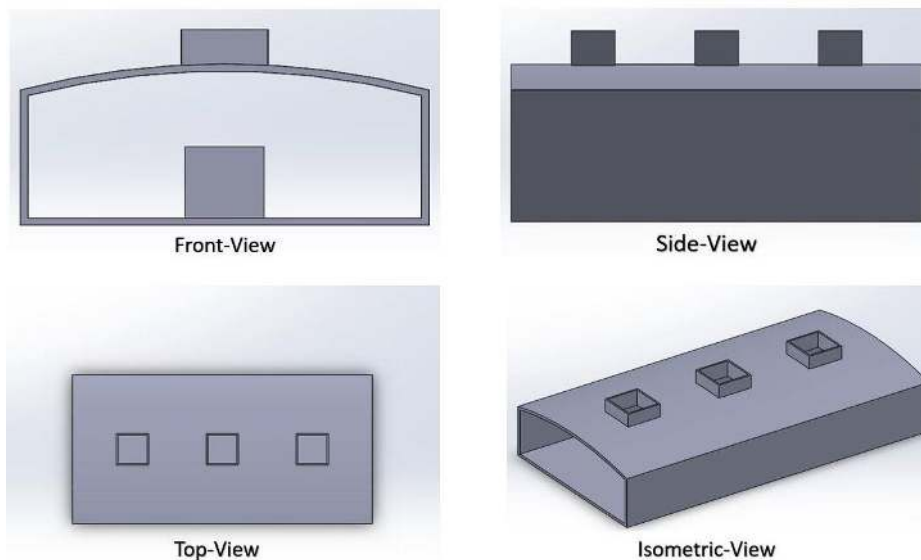


Figure 3 Different views Created in Solidworks for analysis, showing the ducts placement , height and other necessary views (Solid)

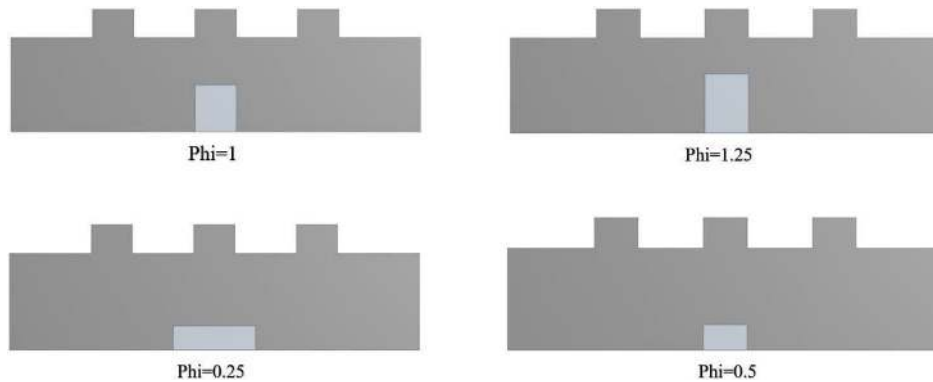


Figure 4 different aspect ratios for studying the effect of mass flow rate and other parameters with respect to size ratio.

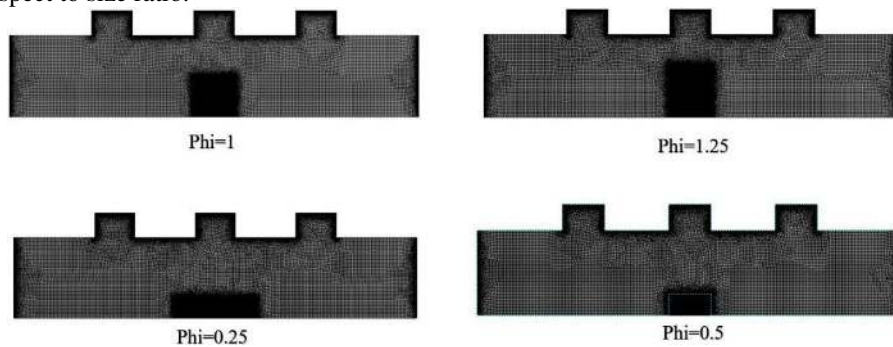


Figure 5 A uniform Quad-Mesh for better result predictions and inflation are provided wherever better flow physics needs to be captured.

2. Methodology

2.1 Governing Equations

The figure 1-4 shows the tunnel with heats source mounted on the lower wall with the positions of all the vents. The displayed heat source has a height (h), length (d) and breadth (w). The aspect ratio Φ is considered to be ratio of height to that of length. The different ratios (h/d) are 0.5, 0.25, 1, 1.25. The opening at the roof ($l_2 = b_2 = w_2 = 0.11$) as well as the opening at the different ports are naturally ventilated. The problem is modelled as a 3- dimensional problem using a L.E.S., Smagorinsky model as a turbulence model while treating density as incompressible ideal gas. The equations which govern the flow are Continuity, Energy, filtered Navier-Stokes equation. The energy equation is coupled with Navier-Stokes equation for computation as it is a natural convection problem.

- Continuity Eq.- (1)

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0$$

- Momentum Eq.- (2)

X momentum:

$$\frac{\partial \rho u}{\partial t} + \frac{\partial \rho u u}{\partial x} + \frac{\partial \rho u v}{\partial y} + \frac{\partial \rho u w}{\partial z} = -\frac{\partial \rho}{\partial x} + \frac{1}{\text{Re}} \left\{ \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right\}$$

Y momentum:

$$\frac{\partial \rho v}{\partial t} + \frac{\partial \rho v u}{\partial x} + \frac{\partial \rho v v}{\partial y} + \frac{\partial \rho v w}{\partial z} = -\frac{\partial \rho}{\partial y} + \frac{1}{\text{Re}} \left\{ \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right\}$$

Z momentum:

$$\frac{\partial \rho w}{\partial t} + \frac{\partial \rho w u}{\partial x} + \frac{\partial \rho w v}{\partial y} + \frac{\partial \rho w w}{\partial z} = -\frac{\partial \rho}{\partial z} + \frac{1}{\text{Re}} \left\{ \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right\}$$

- Energy Eq. (3)

$$\frac{\partial E}{\partial t} + \frac{\partial E u}{\partial x} + \frac{\partial E v}{\partial y} + \frac{\partial E w}{\partial z} = -\frac{\partial u p}{\partial x} - \frac{\partial v p}{\partial y} - \frac{\partial w p}{\partial z} - \frac{1}{\text{Re}(\text{Pr})} \left\{ \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right\} + \frac{1}{\text{Re}} \left\{ \frac{\partial p}{\partial x} [u(\tau_{xx}) + v(\tau_{xy}) + w(\tau_{xz})] + \frac{\partial p}{\partial y} [u(\tau_{yx}) + v(\tau_{yy}) + w(\tau_{yz})] + \frac{\partial p}{\partial z} [u(\tau_{zx}) + v(\tau_{zy}) + w(\tau_{zz})] \right\}$$

2.2 Boundary Conditions

The ambient temperature (T_∞) is set to be 300K while the temperature of the heat source is 1200K and is treated as an isotherm. The walls of the tunnels are also set at ambient temperature and the exit ports including the horizontal and vertical opening are set to be at pressure outlets.

2.3 Mesh Based study

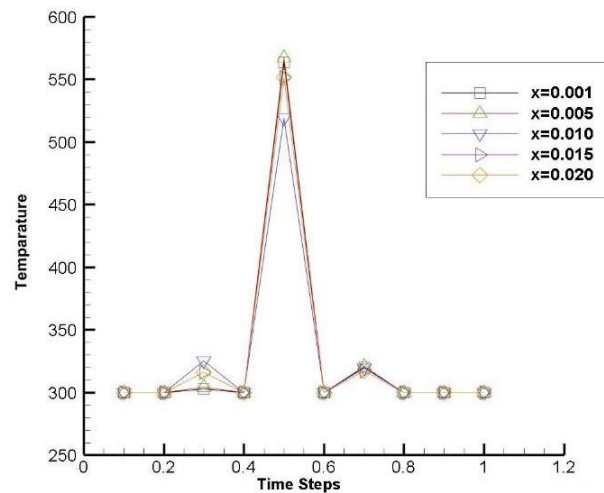


Figure 6. Grid study by using different element sizes

As seen from figure 6, different mesh of edge length 1mm to 20 mm has been used to study the grid dependence and significantly results of 1 mm and 5 mm overlap as in fig.6. Mesh based study has been performed as it is crucial parameters for capturing physics of flow. Large element size is not suitable for capturing very small scale phenomena as the cell volume may distort and continuum may get adversely effected and correct results could not be met.

2.4 Simulation and Set-Up

Table 1 The CFD solver setting and boundary conditions

Inlet	Pressure Outlet
Outlet	Pressure Outlet
Vent 1	Pressure Outlet
Vent 2	Pressure Outlet
Vent 3	Pressure Outlet
Gravity	-9.81(Y-Direction)
Upwinding Technique	Second Order
Discretizing Eq.	Continuity, Momentum, Energy
Solver	Pressure Based
Mesh Count	8,36,234
Pressure/velocity coupling	Simple

The mesh used is orthogonal with $x=0.0005$ due to system limitations, but is fine as lies in the satisfying range for analyzing. The orthogonality limit was found to be 0.85 and skewness was found out to be 0.05 and the Element Quality was 0.08. The total mesh count was 8 Lakhs. The Meshing was performed by using default Ansys workbench 2019, using a parallel meshing process using i5 processor with 4 core, to reduce the computing load as parallel processing takes more time to compute the cell grid for analysis. The total estimated time for grid developing was 25 minutes due to system limitations and also because of the fine mesh size of the cell to be created.

3. Results and Discussion

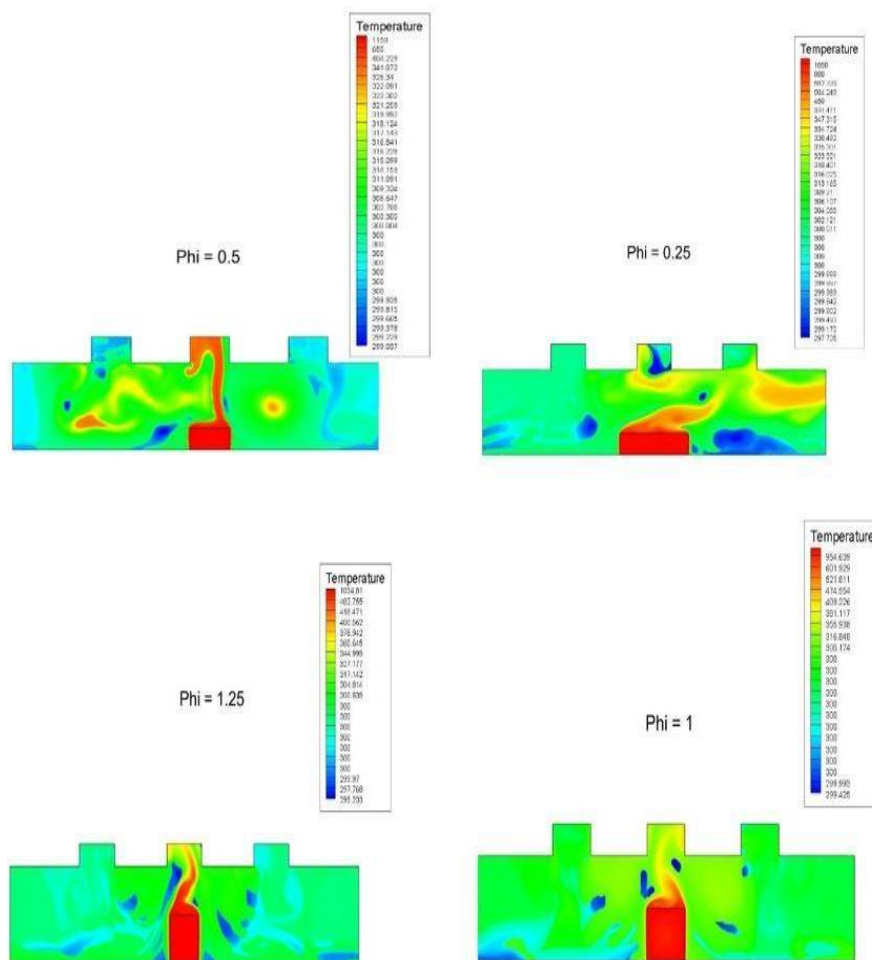


Figure 7 Temperature Contours of different body dimension ratios

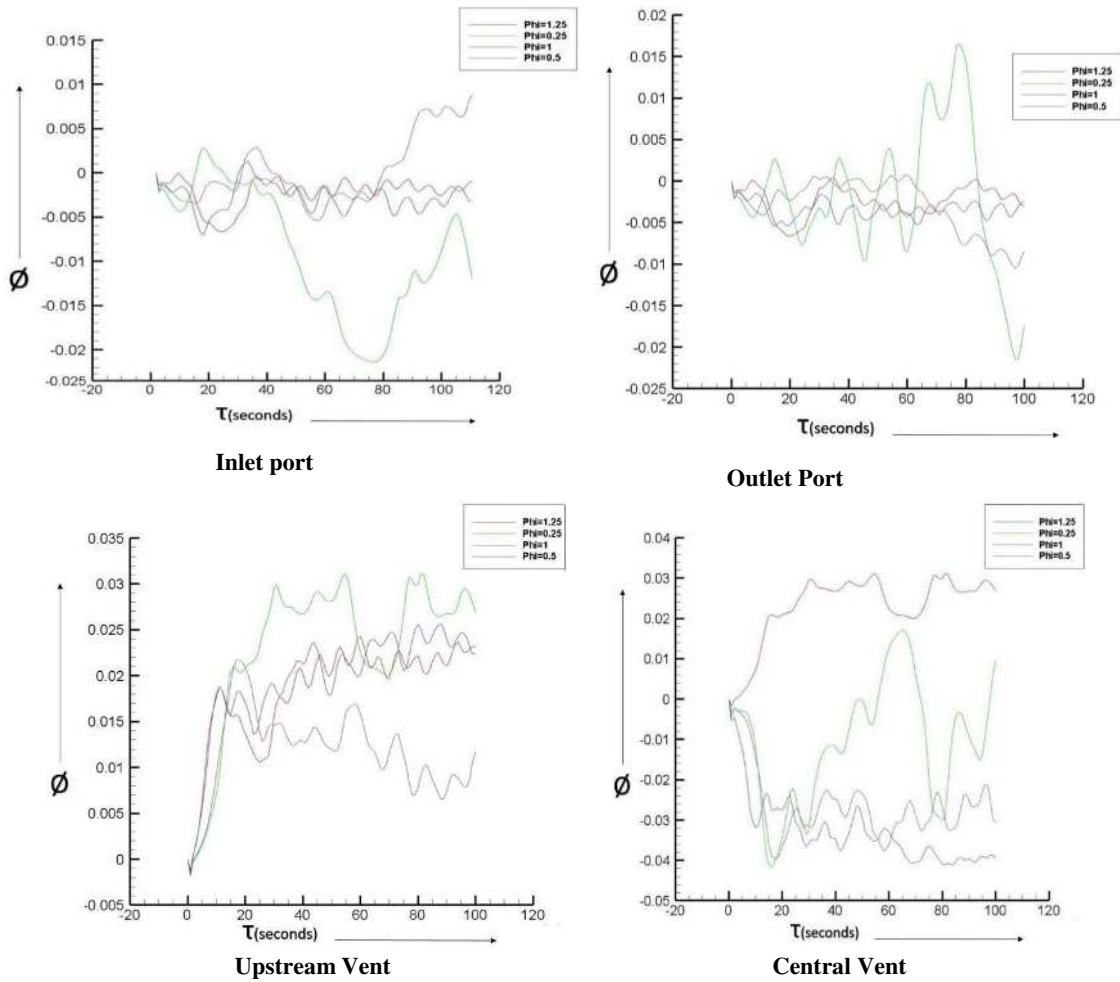


Figure 8. Flux Vs Time(seconds) graph from different vent positions and of different dimensional ratios. The graph shows ambient air entering and Hot air leaving from the provided ports.

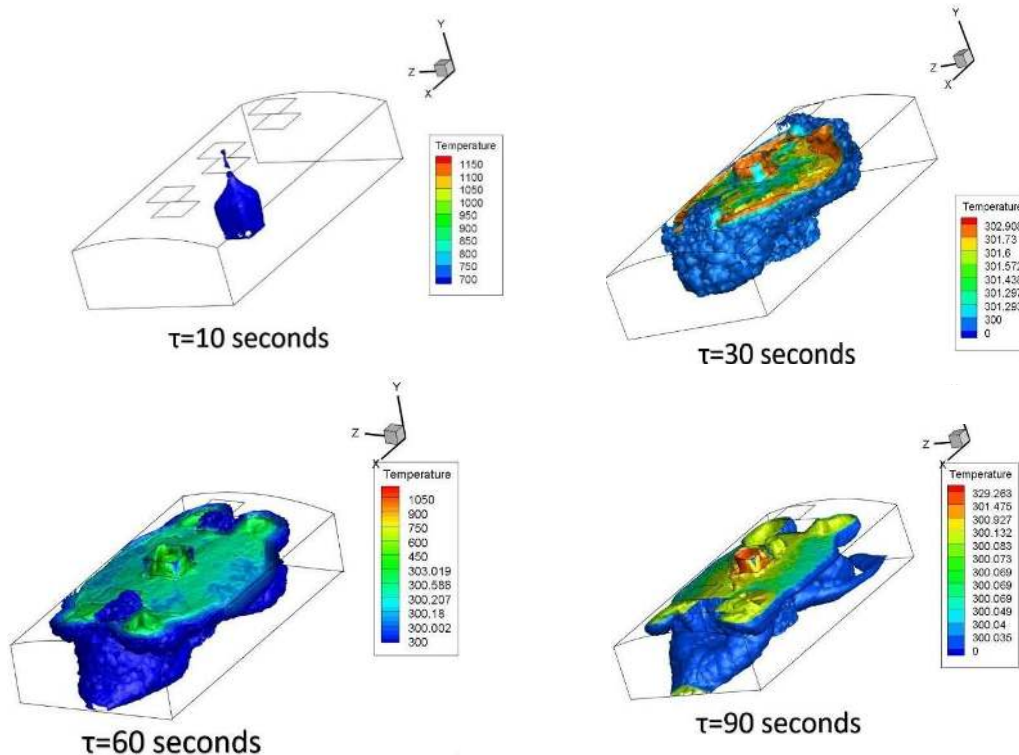


Figure 9. Iso-Surface plots showing the time dependent non-linear expansion effect of hot air inside the domain

In the above figures 6-8, ϕ represents the mass flow rate marching time as the simulation proceeds. Figure 7 show the temperature contours and figure 9 indicates iso-surface of $\phi=1$, and show the marching phenomena bi-directionally in lateral and longitudinal direction with respect to time step of 0.1 seconds. It is seen that stacking effect is predominant on the upper layers of the wall even when ambient air flows inside the room. Fig 8 show mass flow rate through inlet, outlet, Upstream, Central and Downstream vents which are located in vertical and horizontal direction, of different sizes of heat source. From the result it is clear that the distribution is non-linear within the room as the hot air leaves through the ports. The amount of the heated air also increases non-linearly as passage of time. The increment can be said as exponential as it enormously increases. The inlet and outlet port show negative mass flow rate which means ambient air enters through those ports, which turns out to be a time dependency factor. The upstream and downstream vent show positive mass flow rate which means air extraction is majorly through those ports. The Central vent show positive mass flow rate for $\phi=1.5$ while for rest it shows negative low rate. Therefore, we can incur from this knowledge that the air extraction central vent is a function of size of the iso-therm.

3.1 Transient Behaviors

It is seen that the temperature cloud rises and at $t = 10$ seconds and it reaches the topmost part of roof. As time proceeds, smoke distributes itself longitudinally in a non-linear behaviors and escapes from different ports that are available. From figure 8 and fig 9, it is visible that smoke accumulates at the position, which is the most bottom surface of roof and as time increases, it spreads bi-directionally in longitudinal direction as well as in transverse direction un-uniformly.

3.2 Influence of different Phi

For $\Phi = 0.25$, ambient air flows in from inlet and outlet port as time increases and smoke flows out majorly from vertical vents. Accumulation near the outlet vent can be seen. For $\Phi = 0.5$, smoke diffusion is more with increasing time. Major mass flow can be seen near the inlet and outlet ports, while the ambient air enters through vertical ports. For $\Phi = 1$, the ambient air enters from Inlet and Outlet port, whereas the mass accumulation takes place near Vent 1 and Vent 3. For $\Phi = 1.25$, accumulation occurs at Vent 1 and Vent 3, and least accumulation at Vent 2, therefore ambient air enters through Vent 2 and Inlet and Outlet ports.

3.3 Flow Behavior with Time Increment

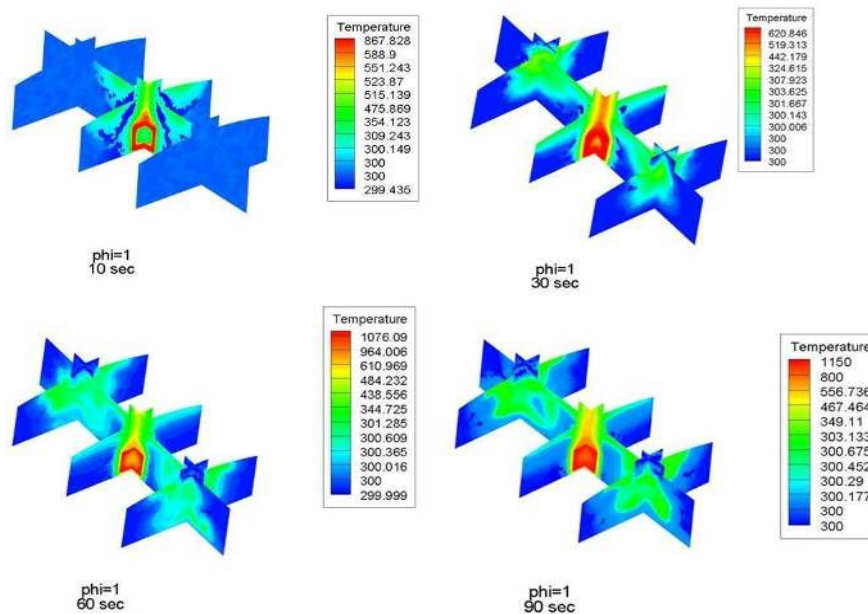


Figure 10. Contour Plots of $\phi=1$ show effect of temperature distribution in 3D with effect to marching time.

3.4 Mass Flow from different Vents for $\Phi = 1$

As visible in the figure 11, ambient air enters through central vent and while hot air is extracted by Upstream and Downstream vent. The magnitude of air entering increases with time and is also in equivalence with air removed, as in sum of air removed from Upstream and Downstream vent. The distribution is non-linear of ambient air entering as well as hot air leaving. The mass flow rate from Upstream and Downstream are nearly equal when $\Phi=1$.

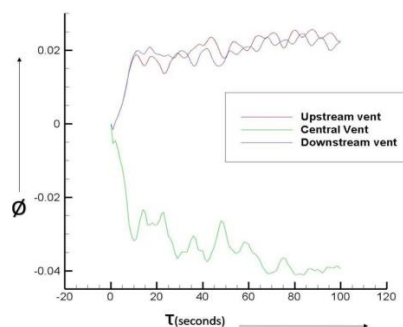


Figure 11. Mass Flow escaping from Vertical openings with respect to marching time

4. Conclusions

As the aspect ratio increases, discrepancy and smoke diffusion increases with time. The temperature cloud rises in the form of a plume and spreads across the domain transversally and longitudinally. It can also be visualized that the plume distribution is mainly on the upper edge of the tunnel, hence we can observe roof vents have better extraction chances than vertical vents while the ambient air is seen to enter majorly where smoke distribution or smoke accumulation is less, i.e. Inlet and Outlet port. As the ambient air enters the tunnel and interacts with the plume, more disturbance in plume is observed which force the air inside the tunnel to re-circulate and escape from the provided vents, reducing the accumulation of smoke within the tunnel. It is also clearly visible that mass flow rate slightly increases with the increase in the aspect ratio of the heat source. The mass being extracted from the vent and mass of ambient air entering through the vent can be seen to be in equivalence with each other. The distribution of air is seen to be non-linear and growth rate of the plume is exponential.

References

- [1] Rui Guo, Yafeng Gao, Chaoqun Zhuang, Per Heiselberg, Ronnen , Levinson, Xia Zhao, Dachuan Shi 2020 Optimization of cool roof and night ventilation in office buildings:A case study in Xiamen, China,*Renew. Energy* **147** 2279-2294.
- [2] Kobayashi T, Chikamoto T, Osada K, 2013 Evaluation of ventilation performance of monitor roof in residential area based on simplified estimation and CFD analysis ,*Build. Environ.* **63** 20-30.
- [3] Ameer S, Chaudhry H, Agha A, 2016 Influence of roof topology on air distribution and ventilation effectiveness of wind tower,*Energy and Build.* **130** 733-746.
- [4] Mao S, Yang D 2016 One-Dimensional analysis for optimizing smoke venting in tunnels by combining roof vents and longitudinal ventilation.,*Appl. Therm. Eng.* **108** 1288-1297.
- [5] Jaluria Y, Chiu K.S.W, Lee S.H.K 1995 Flow of smoke and hot gases across horizontal vents in room fires,*Combust. Sci. Technol.* **110** 197-208.
- [6] Tan Q, Jaluria. Y 2001 Mass flow through a horizontal vent in an enclosure due to pressure and density differences, *Int. J. Heat Mass Transfer* **44** 1543-1553.
- [7] Chow W.K, Gao Y 2009 Oscillating behavior of fire-induced air flow through a ceiling vent,*Appl. Therm. Eng.* **29** 3289-3298.
- [8] Chow W.K, Gao Y 2011 Buoyancy and inertial force on oscillations of thermal induced convective flow across a vent, *Build. Environ.* **46** 315-323.
- [9] Jiang Y, Chen Q 2003 Buoyancy-driven single-sided natural ventilation in buildings with large openings .*Int. J. Heat Mass Transfer* **46** 973-988.
- [10] Zhong W, Fan C, Ji J, Yang J. 2013 Influence of longitudinal wind on natural ventilation with vertical shaft in a road tunnel fire, *Int. J. Heat Mass Transfer* **57** 671-678.
- [11] Yang X, Xing Y, Shi Z, 2013 Analyse Smoke extraction efficiency on Point Exhaust System of side direction for Immersed Tunnel impact Shape of Smoke Dampers, *Appl. Mech.* **405** 1273-1277.
- [12] Krol M., Krol A., Koper Piotr, Wrona P 2017 Full scale measurement of the operation of fire ventilation in a road tunnel, *Tunn. Undergr Space Technol* **70** 204-213.
- [13] Chen L. F, L.H. Hu, Tang W, Yi L, 2013 Studies on Buoyancy driven two-Directional smoke flow layering length with combination of point extraction and longitudinal ventilation in tunnel fires, *Fire Saf J* **59** 94-101.
- [14] Chow W. K , 1998 On smoke control for tunnels by longitudinal ventilation, *Tunn. Undergr. Space Technol* **13** 271-275.
- [15] Fan C. G, J. Ji, Gao Z. H, Han J.Y, Sun J.H, 2013 Experimental study of air entrainment mode with natural ventilation using shafts in road tunnel fires, *Int. J. of Heat and Mass Transfer* **56**, 750-757,
- [16] Polat O, Bilgen E, 2003, Conjugate heat transfer in inclined open shallow cavities, *Int. J. of Heat and Mass Transfer* **46** 1563-1573.
- [17] Gong J, Li Y, 2010, CFD modelling of the effect of fire source geometry and location of smoke flow multiplicity, *Build Simul* **10** 205-214 .
- [18] Lee S, Sun Ryoung H, 2006 A numerical study on smoke movement in longitudinal ventilation fires for different aspect ratio, *Building and Environ*, **21** 719-725.
- [19] Ji J, Bi Y, Venkatasubbaiah K, Li K, 2016 Influence of aspect ratio of tunnel on smoke temperature distribution under ceiling in near field of fire source, *Appl. Therm. Eng.* **106** 1094-1102 .
- [20] Wojciench W, Grzegorz K, 2017 Combined wind engineering, smoke flow and evacuation Analysis for a design of a smoke and heat ventilation ,*Building Research Institute ITB* **172** 1243-1251.
- [21] Lin P, Tao Y, Li Z, You L. Si 2016 A Numerical Study on the Impact Of Vehicles' Blockage on the

- Performance of Semi-transversal Smoke Control System in Tunnel fire, Dept. of Fire Saf. Engg. **135** 248-260.
- [22] Hua G. Y, Wang W, Zhao Y.H, Li L, 2011, Urban Traffic Link Tunnel, *Tunn. Undergr. Space Technol* **26** 336-344.
- [23] Gilani S, H. Montazeri, B. Blocken, 2016 CFD simulation of stratified indoor environment in displacement ventilation: validation and sensitivity analysis *Build. Environ.* **95** 299–313.
- [24] Kosutova K, Van Hooff T, Blocken B 2018 CFD simulation of non-isothermal mixing ventilation in a generic enclosure: impact of computational and physical parameters, *Int. J. Therm. Sci.* **129** 343–357.
- [25] Iousef S, Montazeri H, Blocken B, Wesemael P.J.V. van, 2017 On the use of non-conformal grids for economic LES of wind flow and convective heat transfer for a wall mounted cube, *Build. Environ.* **119** 44–61.
- [26] Nardini G., M, Corvaro F 2014 Effect of heat transfer on natural convection in a square cavity with two source pairs, *Heat Tran. Eng.* **35** 875–886.
- [27] Chan Y.L, Tien C.L 1985 Numerical study of two dimensional laminar natural convection in shallow open Cavities, *Int. J. Heat Mass Tran.* **28** 612–630.
- [28] Krol M., Krol A., Koper P., Wrona P 2017 Full Scale Measurements of the operation of fire ventilation in a road tunnel, *Tunn. Undergr. Space Technol.* **70** 204-213.
- [29] Santolini E., Pulvirenti B. Benni S., Barbaresi L., Torreggiani D., Tassinari P 2018 Numerical study of a wind driven natural ventilation in a greenhouse with screens, *Comput Electron Agric* **149** 41-53.
- [30] Vauquelin O, Wu Y, 2006, Influence of tunnel width on longitudinal smoke control, *Fire Saf J.*, **41** 420-426.
- [31] Biswas N, Mahapatra P.S Manna N.K, Roy P.C 2016 Influence of heater aspect ratio on natural convection in a rectangular enclosure, *Heat Tran. Eng.* **37** 125–139.
- [32] Smagorinsky J 1963 General circulation experiments with primitive equations– I, *Meteorol Soc.* **91** 99-164.
- [33] Chen Yu-Jen, Shu Chi-Min, Ho San-Ping, Chien Shen-Wen 2019 Analysis of smoke in the Hsuehshan tunnel fire, *Tunn. Undergr. Space Technol.* **69** 142-150.
- [34] Jiang Y., Chen Q., 2001 Study of natural ventilation in buildings by large eddy simulation, *J. Wind Eng. Ind. Aerod-* **89** 1155–1178.
- [35] Hooff T. van, B. Blocken, Y. Tominaga, 2017 On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: comparison of RANS, LES and experiments, *Build. Environ.* **114**, 148–165.
- [36] Hooff T. van, Blocken B., G.J.F. van Heijst, 2013 On the suitability of steady RANS CFD for forced mixing ventilation at transitional slot Reynolds numbers, *Indoor Air* **23** 236–249.
- [37] Germano M, Piomelli U, Moin P 1991, A dynamic sub grid scale eddy viscosity model, *Phys. Fluids* **3** 1760– 1765.
- [38] Jin S, Jin J, Gong Y 2020 A theoretical explanation of natural ventilation at roof openings in urban road tunnels, *Tunn. Undergr. Space Technol.* **98** 103345.
- [39] Harish R, 2018 Effect of heat source aspect ratio on turbulent thermal stratification in a naturally ventilated enclosure *Build Environ* **143** 473-486.
- [40] Harish. R, Venkatasubbaiah K 2014 Effects of buoyancy induced roof ventilation systems for smoke removal in tunnel fires, *Tunn. And Undergr. Space Technol.* **42** 195-205.