

Design of Effective Tunnel Ventilation System using Computational Fluid Dynamics

Ashley A. Mason, M. Sairam, Jacob J. Vettikattil, R. Murthy, R. Harish

Abstract: A realistic model of a naturally ventilated tunnel with side ducts is analyzed by Computational Fluid Dynamics. This study accounts for the different temperatures of heat sources and the different domains as the phenomenon of smoke marching in the longitudinal direction is captured. The study accounts the domain to be air domain, and then CO domain. The effect of varying the heat intensity with the marching of smoke is analyzed in the research. Critical parameters like air flow direction and velocity of smoke are taken into account. The paper mainly focuses on temperature distribution, modelling the problem as a convection problem.

Keywords: Large Eddy Simulation, Turbulence Flow, Computational fluid dynamics, Boundary Conditions.

I. INTRODUCTION

As with the growth of population, urbanization is also increasing and so is the use of complex structures and building and subways. As we know that, for transporting and travelling for long distances, an area to move is required which may be open or maybe closed. Sometimes the automobile of passenger cars or trains are to go through tunnels and tunnels design is a complex procedure which requires air flow study and sometimes study of different species or gases.

Not many investigations that focus on smoke control and fire safety in tunnels and it still remains a field of research as it is a complex procedure and involves a lot many assumptions.

Yang, Xing et al, [1] performed their study based on the different sizes and shapes of dampers for smoke regulation in a tunnel. They traced the effect of CO (harmful gas) and captured its effect of spreading and how efficiently it is regulated by the damper. The study concluded that flat dampers have better capture efficiency. Wojciench et al, [2] put forward the very basic methods that can be used for natural smoke and heat ventilation system design, considering the effect of wind conditions. They also carried out a transient CFD analysis accounting the various and major problems in the design. Fire related issues have also been focused in the research e.g. shopping mall in Warsaw, Poland. The simulation was carried on Ansys Fluent.

Revised Manuscript Received on April 15, 2020.

*Correspondence Author

Ashley.A.Mason, School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamilnadu-600127, India.

M. Sairam, School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamilnadu-600127, India.

Jacob J. Vettikattil, School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamilnadu-600127, India.

R. Murthy, School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamilnadu-600127, India.

R.Harish*, School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamilnadu-600127, India.. Email: harish.r@vit.ac.in

Krol et al, [3] had their prime focus on the road ventilation system that was conventionally operated in southern Poland. They performed numerical simulations using Ansys fluent to determine the velocity resulting from the tunnel. Their research was based on the buckling effect of smoke and stack effect. Chen, Shu et al. [4] An accident in 2012, a fire accident in the Hsuehshah road tunnel, in which two fatal and a significant number of injuries occurred triggered this research. Their simulation was carried out on Fire Dynamic Simulator (F.D.S.). They researched that during an outbreak, doors or connectivity to tunnel should be shut automatically so that fewer accidents occur and rescue operation can be carried out successfully.

Peng, Zhang et al, [5] their investigation focused on the impact of blockage made in a tunnel by the action of vehicles to test the performance of smoke control. Their simulation was carried out on Ansys Fluent 14, and F.D.S. Their prime interest was extraction rate of smoke. Their simulations were made by study of the smoke patterns in the tunnel and later they proposed a semi-transverse smoke control system, that was able to efficiently maintain safe smoke extraction methods.

Hua, Zhao et al, [6] conducted a research of UTLT (Urban Traffic Link Tunnel), a optimum underground transportation system which consists of different linkages linked together. It consisted of a main tunnel as the main linkage connected to different tunnels. This research was used to investigate a smoke flow control strategy for controlling a fire scenario in case there is a firebreak in Beijing Centre Business District (C.B.D). The parameters that were taken into account for the calculation are critical velocity, minimal smoke, spreading area. The devised a total of six different strategies of smoke control. The spreading of smoke within the area with its temperature were analyzed and the major concern was CO₂ concentration.

II. EXPERIMENTAL SETUP

The geometry consists of a tunnel of dimension 8000x600x523 mm. The tunnel consists of a train, side vents and a heat-source. Six ducts have also been placed in the tunnel of width 450mm and depth 110 mm. Each duct is at a distance of 950mm from the adjacent one. The tunnel is hollow and a train is placed inside the tunnel which is of dimension 1920x138.43x362.56 mm. Windows of dimension 130x50 mm are present on the train from which the smoke will escape. The train is placed at a distance of 1000mm from the inlet. The dimensions of the heat source are 1045x100x20 mm.



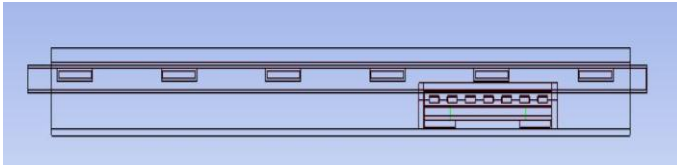


Figure 1. Schematic diagram of geometry

The figure 1 represents the train geometry with a train and heat source inside it.

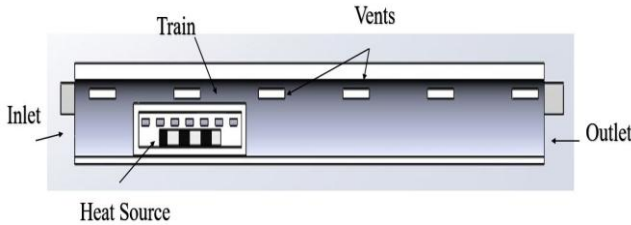


Figure 2. Cross Section of the geometry

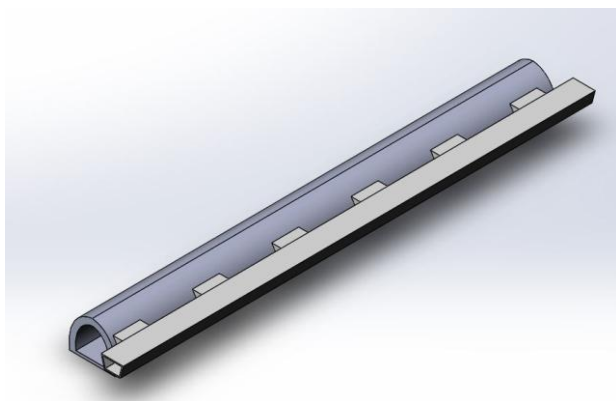


Figure 3. Solid geometry of tunnel

III. MATHEMATICAL MODELING AND NUMERICAL METHOD

In the following research, Finite volume method is used for calculations and simulation. The Reynolds averaged Navier-stokes (RANS) for getting the time average energy equation and temperature field. L.E.S. Smagorensky turbulence model is applied for modelling the turbulence. For the energy that includes surface and surface radiation, then the governing equation as to solve:

1. *X – Momentum :*

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

2. *Y – Momentum :*

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

3. *Z – Momentum :*

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

$$-\frac{\partial p}{\partial z} + \frac{1}{Re} \left\{ \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right\}$$

4. *Energy Equation :*

$$\frac{\partial(E)}{\partial t} + \frac{\partial(Eu)}{\partial x} + \frac{\partial(Ev)}{\partial y} + \frac{\partial(Ew)}{\partial z} = -\frac{\partial up}{\partial x} - \frac{\partial vp}{\partial y} - \frac{\partial wp}{\partial z}$$

$$-\frac{1}{Re(Pr)} \left\{ \frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right\}$$

5. *Continuity :*

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

The ambient temperature and the heat source is denoted by T_s and that of infinity is T_∞ and temperature of walls is denoted by T_w . A small time step of 1 second is provided and total simulation is of 100 seconds and temperature distribution can be seen clearly within 100 seconds. The convergence criteria is set to be 10^{-4} . No slip boundary condition is applied to the walls and the inner walls are coupled and temperature is specified on the faces of heat source. First a cold flow analysis is done to check the working of B.C. with the model, later transient analysis was applied to the simulation

IV. MESHING AND PREPROCESSING

The geometry was created in Solidworks 2019 and was imported to Ansys Workbench 2019 as a IGES format. All the post modelling is done in Ansys Design Modular and later meshing is done using Ansys Meshing itself. The orthogonality limit was 0.7 and skewness was 0.2. By judging these criteria, the meshing can be called a good mesh as it qualified all mesh check parameters. The minimum element length was 10 mm and the total mesh was approximately 5 lakhs. therefore, a course mesh of tetragonal elements is used to solve the simulation. The meshing is denser near the Heat source region for better results.

The calculation is done using Ansys Fluent and all the post processing is done using CFD Post-processing

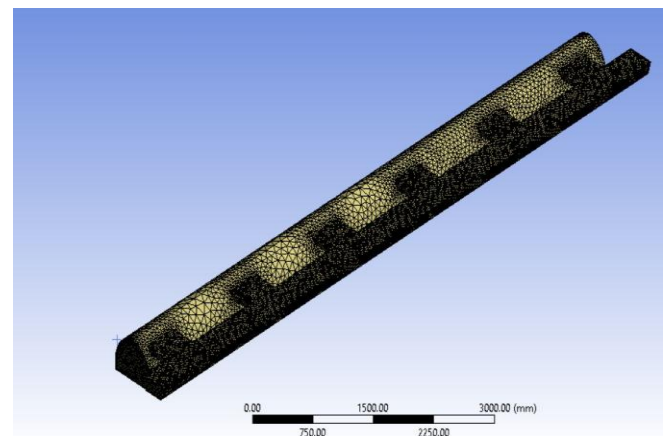


Figure 4. Mesh of the geometry

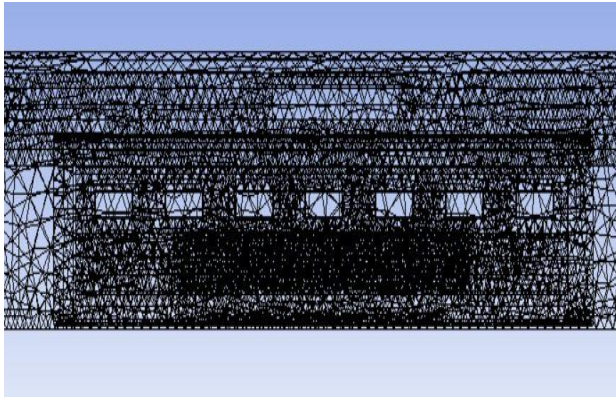


Figure 5. Wireframe view of Train and Heat source

V. SIMULATION SET UP AND COMPUTATON

Inlet	Velocity Inlet
Outlet	Outflow
Density	Incompressible ideal gas
Walls	No Slip
Turbulence Model	LES ,Smagorinsky
Gravity	- 9.81 (y - direction)
Discretization	pressure ,energy , momentum
Upwind techniques	Second Order
Solver	Pressure Based
Pressure/Velocity Coupling	Simple

The fluent solver is used to execute the simulation in which the absolute velocity and pressure based steady flow is executed with gravity acted along y-axis direction. No slip condition is used for the wall region.

VI. RESULTS AND DISCUSSION

Contours and variation in velocity, temperature , density have been plot at 10s and 90s for CO domain and Air domain with 1200 K and 1500 K

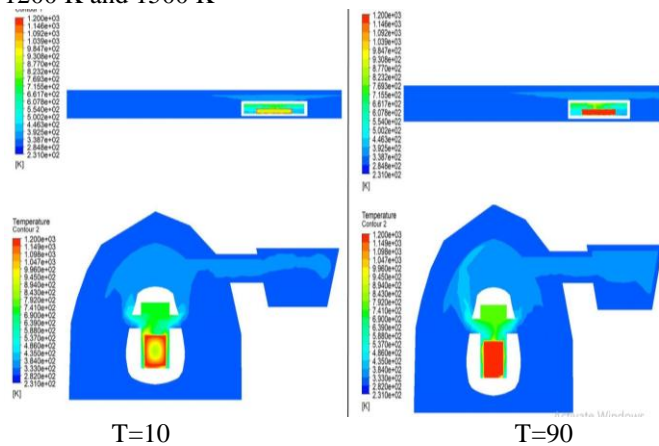


Figure 6. Air- 1200 K

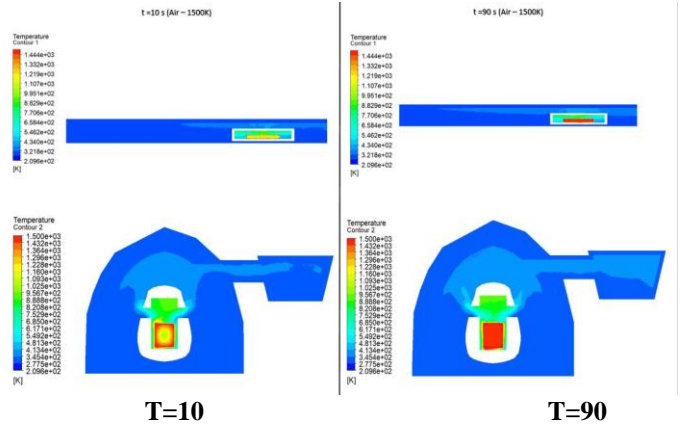


Figure 7. Air- 1500 K

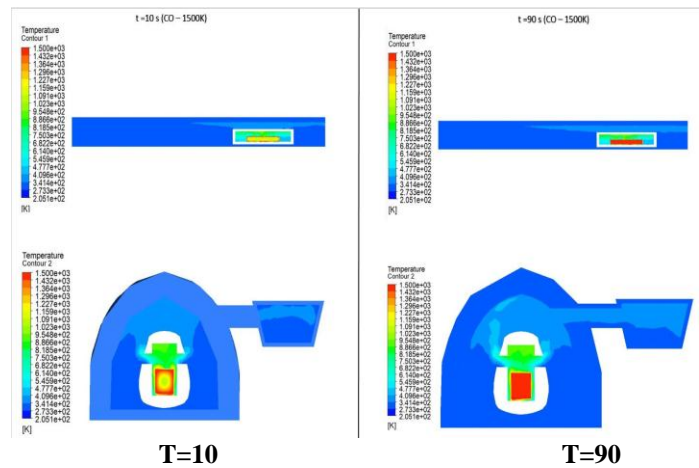


Figure 8. CO- 1500 K

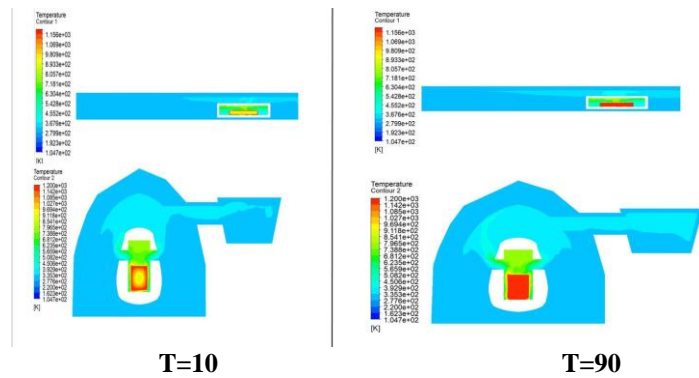


Figure 9. CO- 1200 K

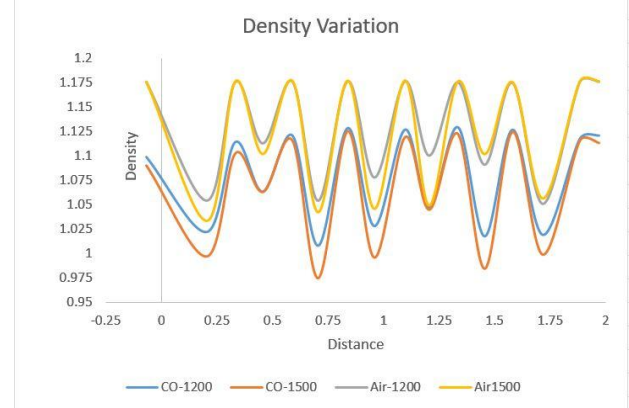


Figure 10. Density Variation

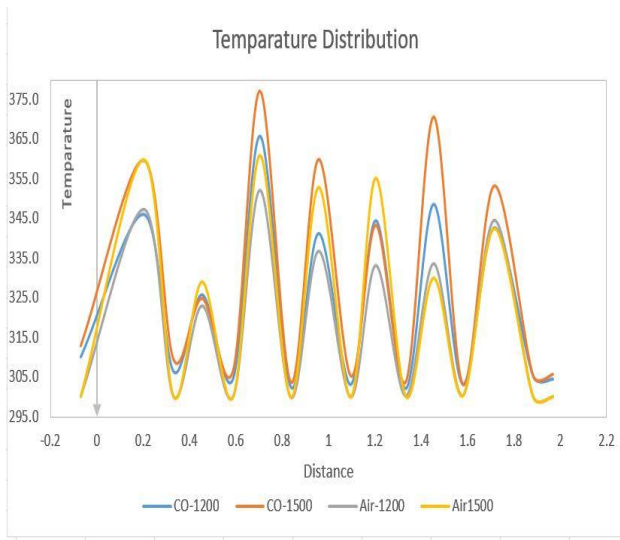


Figure 11 Temperature Distribution

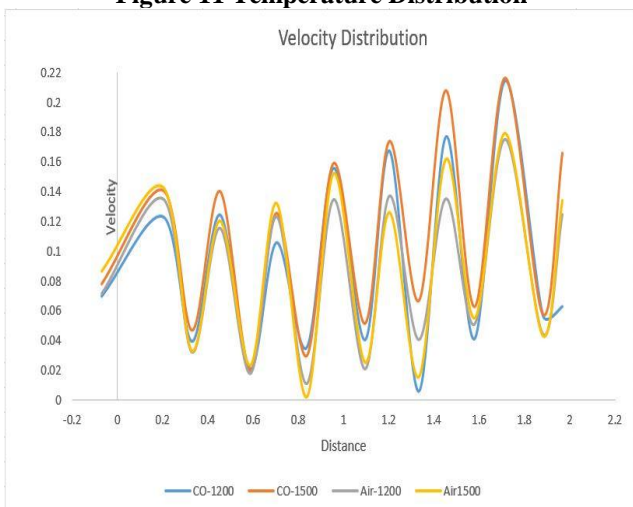


Figure 12. Velocity Distribution

A line taking coordinates (0.13 ,0.0549412, -0.00950586) and (0.13 ,0.0549412, 2.00494) was used to collection 16 sample points to plot data accordingly.

VII. CONCLUSION

Hence the effect of marching of CO and air with varying the heat source can clearly be seen. The critical velocity for dissipation in CO domain is higher than critical in velocity Air domain. Whereas the temperature marching in CO domain is also fast than Air domain. It can be clearly seen the heat distributes itself in the longitudinal direction and enters the ducts therefore proper placement of ducts is also a critical parameter other than velocity and temperature in effective tunnel ventilation. The variation of density can be seen, as and when there is an increase in heat source.

REFERENCES

1. X. Yang, Y. Xing, Z. Shi. Analyze Smoke extraction efficiency on Point Exhaust System of side direction for Immersed Tunnel impact Shape of Smoke Dampers, Applied Mechanics and Materials Vol . 405-408 pp 1273-1277, 2013.
2. W. Wojciench, K. Grzegorz Combined wind engineering, smoke flow and evacuation Analysis for a design of a smoke and heat ventilation, Procedia Engineering Volume 172, 2017, Pages 1243-1251.
3. M. Krol, A. Krol, Piotr Koper, P. Wrona , Full scale measurement of the operation of fire ventilation in a road tunnel, Tunneling and Underground Space Technology, Pg:204-213 ,2017.

4. Yu-Jen Chen, Chi-Min Shu, San-Ping Ho, Shen-Wen Chien, Analysis of smoke in the Hsuehshan tunnel fire, Tunneling and Underground Space Technology, Volume 84, February 2019, Pages 142-150.
5. P. Lin, Y. Z. Tao Li, You L. Si, A Numerical Study on the Impact Of Vehicles' Blockage on the Performance of Semi-transversal Smoke Control System in Tunnel fire , Procedia Engineering Volume 135 2016, Pages 248-260.
6. G. Y. Hua , W. Wang , Y.H. Zhao and L. Li , Urban Traffic Link Tunnel , Tunneling and Underground Space Technology ,Volume 26, Issue 2, March 2011, Pages 336-344.

AUTHORS PROFILE



Ashley A. Mason is pursuing his M.Tech degree in CAD/CAM at VIT Chennai Campus. His research Interest are in the field of Computational fluid dynamics and Computer aided design.



M. Sairam is pursuing his M.Tech degree in CAD/CAM at VIT Chennai Campus. His research Interest are in the field of Computational Fluid Dynamics and Manufacturing.



Jacob J. Vettikattil is pursuing his M.Tech degree in CAD/CAM at VIT Chennai Campus. His research Interest are in the field of Computational Fluid Dynamics and Manufacturing.



R. Murthy is pursuing his M.Tech degree in CAD/CAM at VIT Chennai Campus. His research Interest are in the field of Computational Fluid Dynamics and Manufacturing.



Dr. R. Harish is working as an Assistant Professor in the school of Mechanical and Building Sciences at VIT Chennai campus. His research interests are in the field of computational fluid dynamics, Heat Transfer and Turbulent flows.