# Tunnel Ventilation System Design Using Computational Fluid Dynamics

**AG.MALLIKARJUNA, Associate Professor & Head of H&S Dept., St.Johns College of Engineering and Technology, Yemmiganur, Kurnool(Dist.). E_mail Id: mallipraveena1023@gmail.com**

**S VEERA REDDY, a Research scholar of Rayalseema University.,Associate Professor & H&S Dept., St.Johns College of Engineering and Technology, Yemmiganur, Kurnool(Dist.)KURNOOLE_mail Id: veerakuppagallu@gmail.com**

## Abstract:

Computational Fluid Dynamics is used to examine a realistic model of a naturally ventilated tunnel with side ducts. Diverse heat sources and different domains are taken into consideration in this research as the phenomena of smoke marching longitudinally is caught. According to the findings, the first domain is air, followed by CO. The study examines the impact of altering the heat intensity with the moving of smoke. Smoke velocity and airflow direction are taken into consideration. As a convective issue, the study focuses primarily on temperature distribution.

## Keywords:

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

## I. INTRODUCTION

Urbanization and the utilisation of sophisticated structures and buildings, such as subways, are on the rise as a result of population expansion. As we know, long-distance transportation and travel need a region that may either be open or closed. Tunnel design is a difficult process that needs research into air movement and, on occasion, the analyses of different species or gases to be completed.

Because it is a difficult process that relies on several assumptions, there aren't many studies dedicated to smoke control and fire safety in tunnels.

To regulate smoke in a tunnel, Yang, Xing et al. [1] used a variety of sizes and forms of dampers. To better understand how the damper regulates CO (a hazardous gas), they tracked its spread and how well it is reduced. According to the findings, flat dampers are more efficient in capturing sound waves. According to Wojciench et al [2], wind conditions may have an impact on the design of natural smoke and heat ventilation systems. They also performed a transient CFD study to account for the different and significant design flaws. The study also looked at fire-related concerns, such as a shopping centre fire in Warsaw, Poland. Ansys Fluent was used for the simulation.

Road ventilation systems in southern Poland were the primary focus of Krol et al. [3]. They ran numerical simulations in Ansys fluent to figure out how fast the train would go after it emerged from the tunnel. Using the bucklering impact of smoke and the stack effect, they came up with their findings. Chen, Shu, and colleagues [4] This study was prompted by a 2012 fire mishap in the Hsuehshah road tunnel, that resulted in two deaths and a large number of injuries. Fire Dynamic Simulator was used to simulate the fire (F.D.S.). Doors or tunnel connections should be automatically closed during an outbreak, so that fewer accidents occur and the rescue effort can be carried out effectively.

Peng, Zhang et al. [5] investigated the influence of vehicle obstruction on the performance of smoke control in a tunnel. For their simulation, they used Ansys Fluent 14 and F.D.S. Their main concern was the pace at which smoke was extracted. A semi-transverse smoke management system they created, based on their models of the tunnel's smoke patterns, allowed them to effectively maintain safe smoke extraction procedures.

## II. EXPERIMENTAL SETUP

A 8000x600x523 mm tube serves as the basis for the design. A train, side vents, and a heat source make up the tunnel. In addition, six ducts with a combined width and depth of 450 mm and 110 mm have been installed in the tunnel. Each duct is separated from the next by 950 millimetres. The tunnel is hollow, and a train of 1920x138.43x362.56 mm is located within. Smoke will be expelled from the train via windows of 130x50 mm. It is located 1000mm from the intake of the train. The heat source measures 1045x100x20 millimetres.

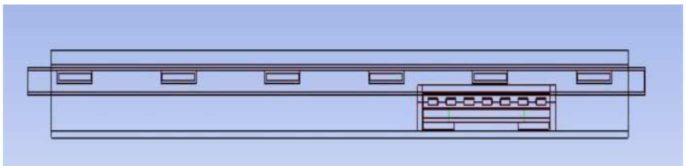


**Figure 1. Schematic diagram of the geometry**

Figure 1 describes the train geometry with a train andheat source in it.

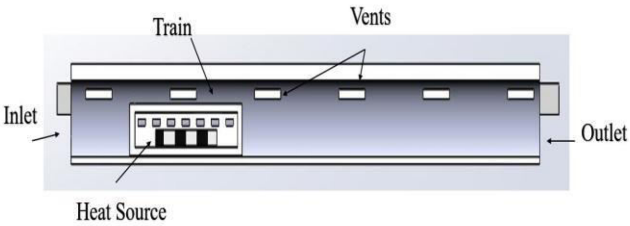



**Figure 2. Cross Section of the geometry**

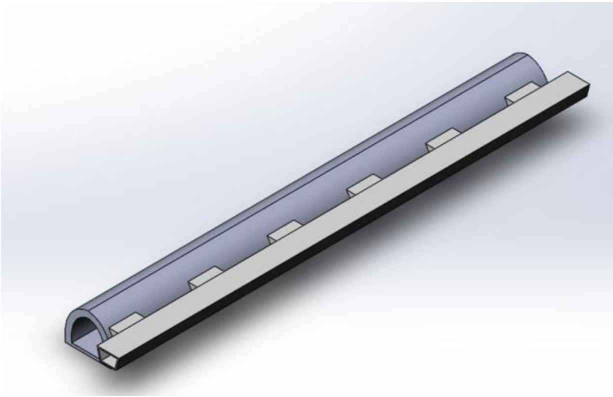


**Figure 3. Solid geometry of tunnel**

## III. MATHEMATICAL MODELING AND NUMERICAL METHOD

The Finite volume approach is utilised in the following studies for calculations and simulations. To get the time-average energy equation and the temperature field, the Reynolds averaged Naiver-Stokes (RANS) method is used. The turbulence is modelled using the L.E.S. Smagorensky turbulence model. Surface and surface radiation energy are included in the governing equation:

1. $X - Momentum$ :

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}= -\frac{\partial p}{\partial x}+\frac{1}{Re}\left\{\frac{\partial \tau xx}{\partial x}+\frac{\partial \tau xy}{\partial y}+\frac{\partial \tau xz}{\partial z}\right\}$$

2. $Y - Momentum$ :

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho vu)}{\partial x}+\frac{\partial(\rho vv)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}= -\frac{\partial p}{\partial y}+\frac{1}{Re}\left\{\frac{\partial \tau yx}{\partial x}+\frac{\partial \tau yy}{\partial y}+\frac{\partial \tau yz}{\partial z}\right\}$$

3. $Z - Momentum$ :

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho wu)}{\partial x}+\frac{\partial(\rho wv)}{\partial y}+\frac{\partial(\rho ww)}{\partial z}= -\frac{\partial p}{\partial z}+\frac{1}{Re}\left\{\frac{\partial \tau zx}{\partial x}+\frac{\partial \tau zy}{\partial y}+\frac{\partial \tau zz}{\partial z}\right\}$$

4. $Energy\ Equation$ :

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

$$-\frac{1}{Re(\mathrm{Pr})}\left\{\frac{\partial qx}{\partial x}+\frac{\partial qy}{\partial y}+\frac{\partial qz}{\partial z}\right\}$$

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 is represented by a Ts, the heat source by a Ts, infinity by a Ts, and the wall temperature by a Tw. A one-second time step may provide a good picture of the temperature distribution across the course of a simulation lasting 100 seconds. The convergence criteria are ten times four. No-slip boundary conditions are applied to the walls and the inner walls are linked and the temperature of each face is determined. To check that B.C. was working correctly with the model, a cold flow study was performed before applying transient analysis.

## IV. MESHING AND PREPROCESSING

As an IGES file, Solidworks 2019 was used to produce the geometry, which was then imported into Ansys Workbench 2019. When it comes to post modelling and meshing, everything is done in Ansys Design Modular. The orthogonality and skewness thresholds were set at 0.7 and 0.2, respectively. When all of these requirements are satisfied, you know that the meshing has been judged to be a success. The smallest piece had a length of 10 mm, while the overall mesh count was over 5 lakh. As a result, the simulation is solved using a coarse mesh made up of tetragonal pieces. To get better outcomes, the meshing is thicker in the Heat source zone. Post-processing is done using CFD Post-processing once the computation is completed using Ansys Fluent

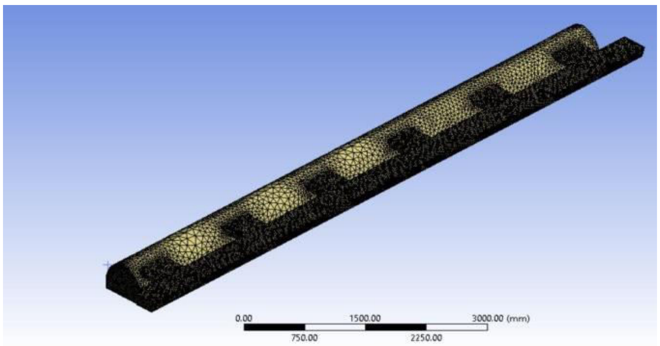



**Figure 4. Mesh of the geometry**

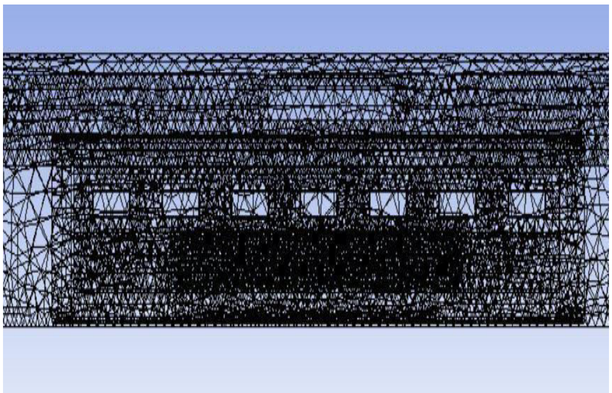


**Figure 5. Wireframe view of Train and Heat source**

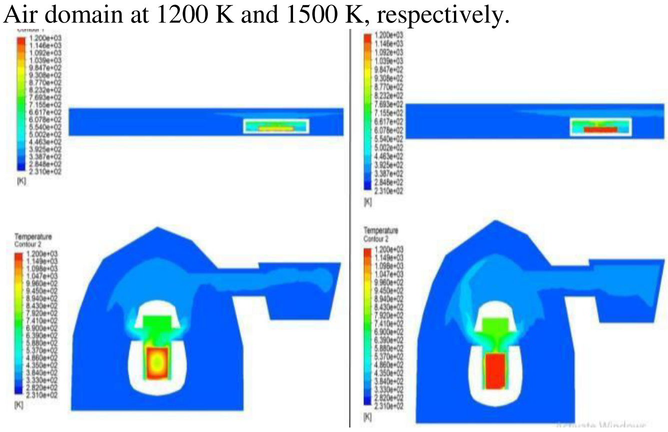
## V. SIMULATION SET UP AND COMPUTATION

| | |
|---|---|
| *Inlet* | *Velocity Inlet* |
| *Outlet* | *Outflow* |
| *Density* | *Incompressible ideal gas* |
| *Walls* | *No Slip* |
| *Turbulance 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 run the steady flow simulation in which the gravity is acting in the y-axis direction. The wall area does not have a slide condition.

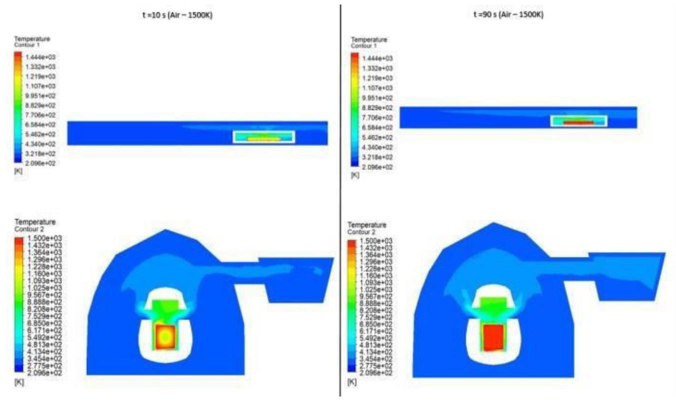
## VI. RESULTS AND DISCUSSION

Contours and variations in velocity, temperature, and density have been shown at 10s and 90s for the CO domain and the Air domain at 1200 K and 1500 K, respectively.



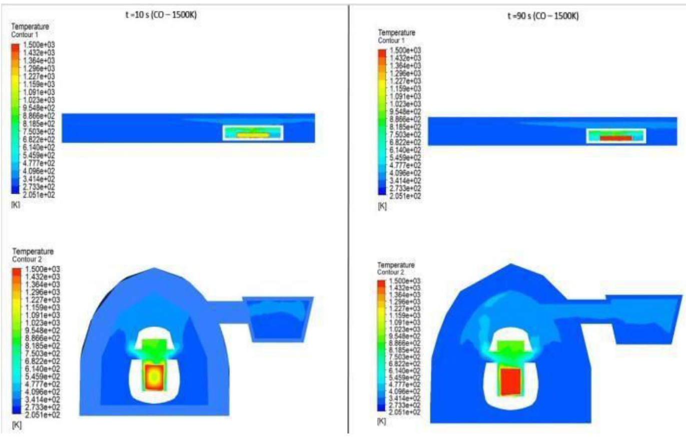
T=10 T=90

**Figure 6. Air- 1200 K**



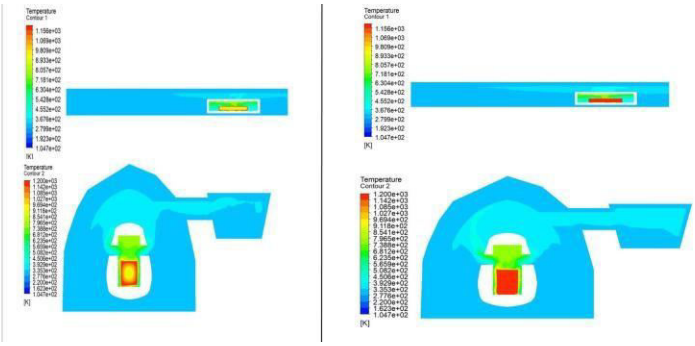
T=10 T=90

**Figure 7. Air- 1500 K**



T=10 T=90

**Figure 8. CO- 1500 K**



T=10 T=90

**Figure 9. CO- 1200 K**

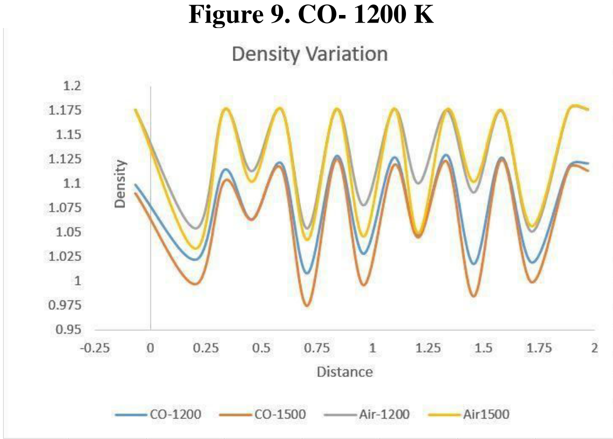



**Figure 10. Density Variation**

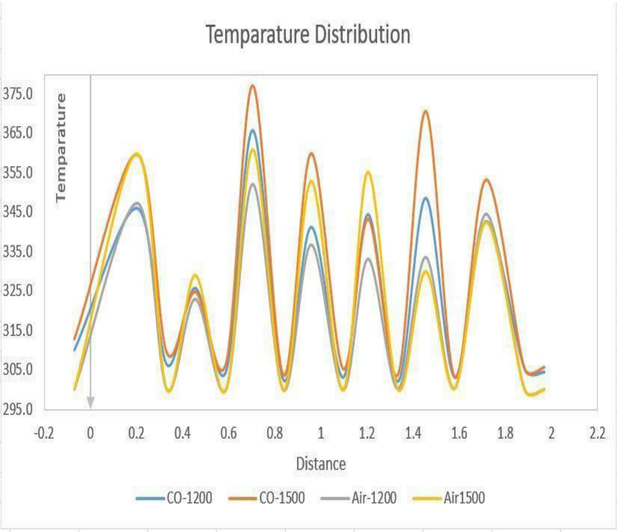



**Figure 11 Temperature Distribution**

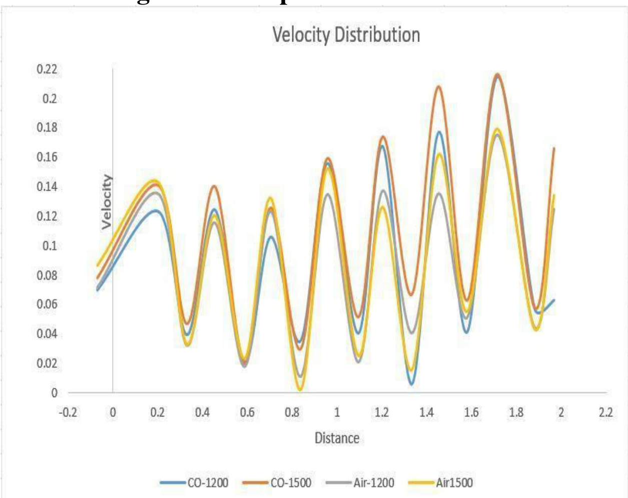



**Figure 12. Velocity Distribution**

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

## VII. CONCLUSION

As a result, the influence of marching CO and air with altering heat source is seen. The critical velocity for dissipation in the CO domain is greater than the critical velocity in the velocity Air domain. Temperature marching in the CO domain is likewise faster than in the Air domain. As can be seen, heat distributes itself longitudinally and enters the ducts, therefore good duct placement, in addition to velocity and temperature, is a vital element in successful tunnel ventilation. The fluctuation in density may be noticed when the heat source is increased.

## 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. Wojcienkh, 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, Tunnelling and Underground Space Technology, Volume 84, February 2019, Pages 142-150, Tunneling and Underground Space Technology pg. :142-150, 2019.
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.