Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

Simulation Of Multi-Component Gas Flow and Heat Flux in Fire Ventilation Systems Using Ansys Fluent

[1]Dr. G. Avinash, [2]Dr. P. Vinod Kumar Naidu, [3]M. Sunil Raj, [4]A. Yeswanth
[1]Associate Professor & HOD
Department of Mechanical Engineering
Pragati Engineering College (Autonomus)
(Affiliation to JNTUK)
Surampalem,Kakinada-533437
[2]Assistant Professor
Department of Mechanical Engineering
Pragati Engineering College (Autonomus)
(Affiliation to JNTUK)
Surampalem,Kakinada-533437
[3]Associate Professor
Department of Mechanical Engineering
Pragati Engineering College (Autonomus)
(Affiliation to JNTUK)
Surampalem,Kakinada-533437
[4]Assistant Professor
Department of Mechanical Engineering
Pragati Engineering College (Autonomus)
(Affiliation to JNTUK)
Surampalem,Kakinada-533437

ARTICLE INFO

Received: 11 Jan 2026

Revised: 21 Feb 2026

Accepted: 02 March 2026

ABSTRACT

Fire events in enclosed environments generate high-temperature multi-component gas mixtures consisting primarily of nitrogen, oxygen, carbon dioxide, water vapor, and toxic combustion products. Accurate prediction of gas dispersion, buoyancy-driven flow, and heat flux distribution is essential for safe ventilation system design. This study presents a Computational Fluid Dynamics (CFD) investigation of multi-component gas flow and thermal transport in fire ventilation systems using ANSYS Fluent. The governing equations for mass, momentum, species transport, and energy conservation are solved under buoyancy-driven turbulent conditions.

The numerical model evaluates:

- Temperature distribution
- Heat flux evolution
- Species concentration fields
- Buoyancy-induced circulation
- Ventilation efficiency

Results demonstrate strong coupling between temperature gradients and density variation, confirming buoyancy as the dominant driving mechanism in fire-induced ventilation flows.

Keywords: demonstrate, mechanism, essential

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

1. INTRODUCTION

Fire inside enclosed structures produces:

- Hot combustion gases
- Density variations
- Strong buoyancy forces
- Toxic species transport

Effective fire ventilation design must ensure:

- Rapid smoke removal
- Temperature control
- Safe evacuation conditions

CFD simulation provides a powerful tool to:

- Predict fire plume behavior
- Analyze gas mixture transport
- Optimize ventilation duct positioning

This study focuses on multi-component reacting/non-reacting gas flow modeling using ANSYS Fluent.

2. PHYSICAL MODELING OF FIRE-INDUCED FLOW

2.1 Buoyancy-Driven Convection

When gas temperature increases:

$$\rho = \frac{P}{RT}$$

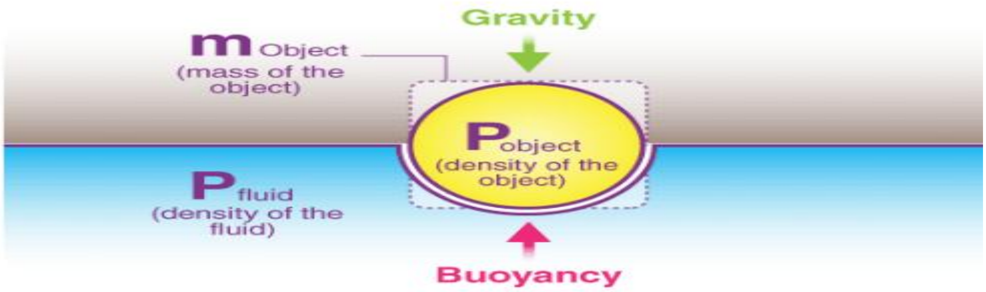
Density decreases → gas rises → natural convection forms.

Buoyancy force:

$$F_b = \rho g$$

Hot gases accumulate near ceiling → ventilation removes them.

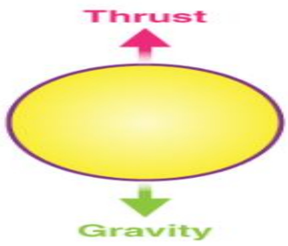
What causes Buoyant Force?



Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/
Research Article

When an object is immersed in water or any other fluid, we observe that the object experiences a force from the downward direction opposite to the gravitational pull, which is responsible for the decrease in its weight.

Demonstration of Buoyant Force



When we submerge an object in a fluid, an upward force is experienced by the object. The fluid applies this force on the object, which causes it to rise, and we call this force buoyant force. The magnitude of this force is precisely equal to the amount of weight of the liquid displaced.

2.2 Multi-Component Gas Mixture

Fire-generated gases typically include:

- O_2 (oxygen)
- N_2 (nitrogen)
- CO_2 (carbon dioxide)
- H_2O (water vapor)

Species transport is governed by:

$$\frac{\partial(\rho Y_i)}{\partial t} + \nabla \cdot (\rho v Y_i) = -\nabla \cdot J_i + R_i$$

Where:

- Y_i= mass fraction
- J_i= diffusion flux
- R_i= reaction source term

3. GOVERNING EQUATIONS

ANSYS Fluent solves:

3.1 Continuity Equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho v) = 0$$

3.2 Momentum Equation

$$\rho \frac{Dv}{Dt} = -\nabla p + \mu \nabla^2 v + \rho g$$

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article

Includes buoyancy source term.

3.3 Energy Equation

$$\rho C_p \frac{DT}{Dt} = \nabla \cdot (k\nabla T) + S_h$$

3.4 Species Transport Equation

$$\nabla \cdot (\rho v Y_i) = \nabla \cdot (D_i \nabla Y_i)$$

4. EQUATION OF STATE

Ideal Gas Law:

$$\rho = \frac{P}{RT}$$

Density variation directly couples temperature and flow field.

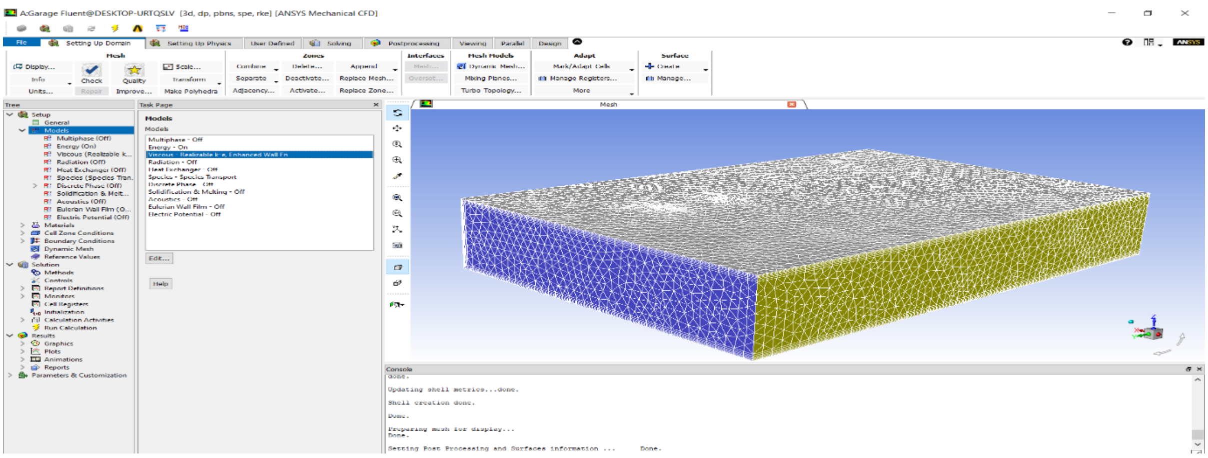
5. COMPUTATIONAL FLUID DYNAMICS APPROACH

5.1 Why CFD?

CFD enables:

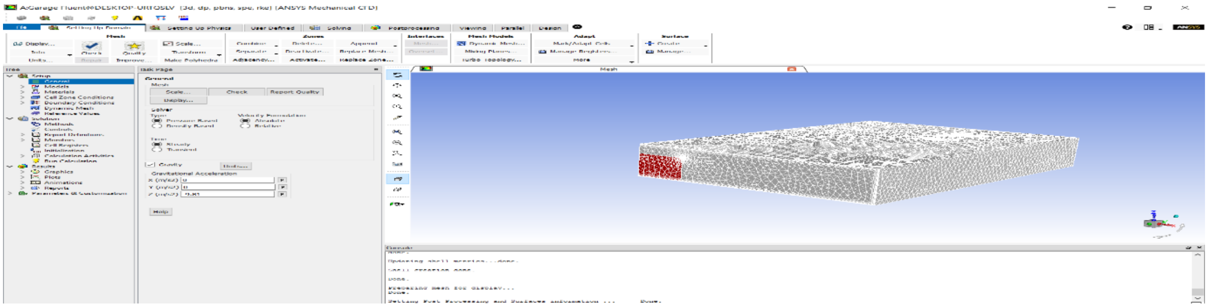
- Prediction of temperature layers
- Smoke propagation
- Toxic gas accumulation
- Ventilation performance evaluation

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows.

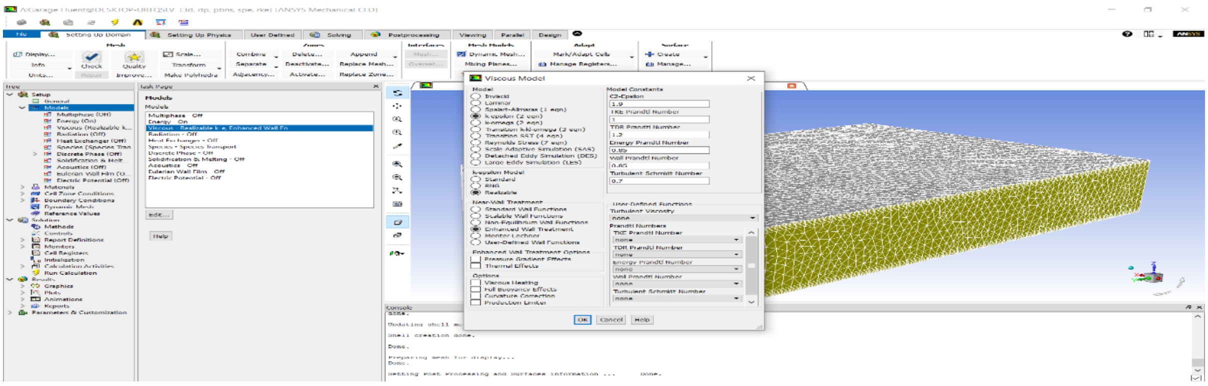


Inlet

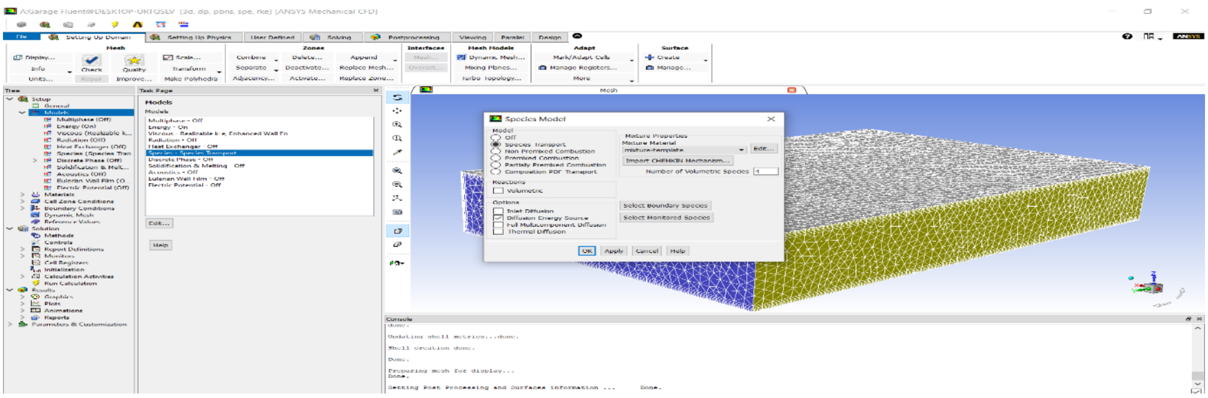
Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ Research Article



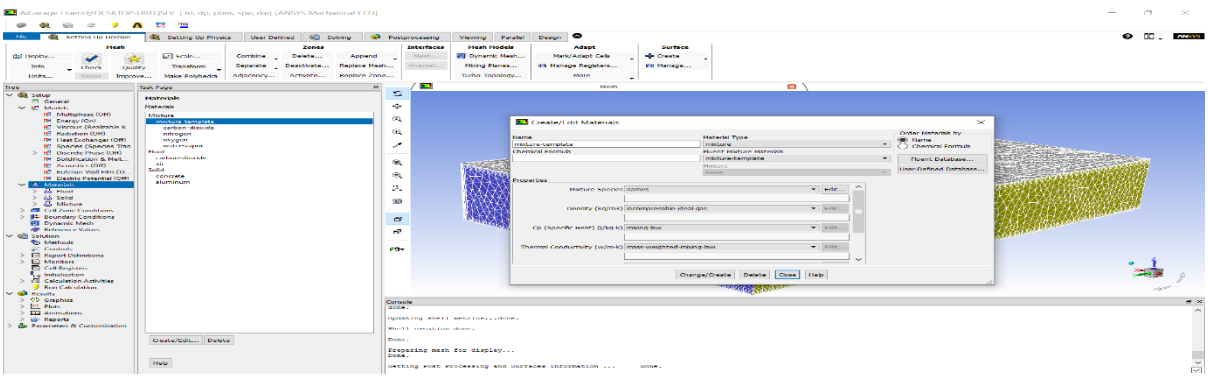
Outlet



Viscous



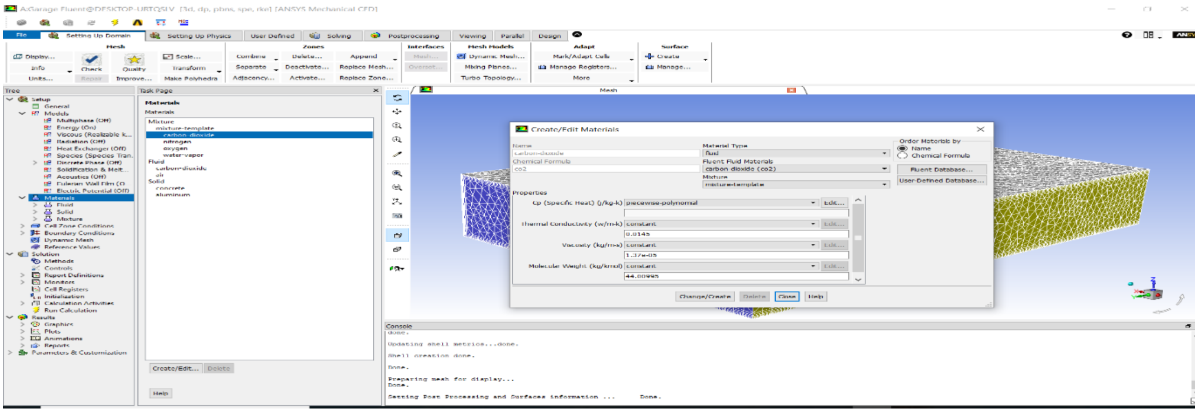
Species model



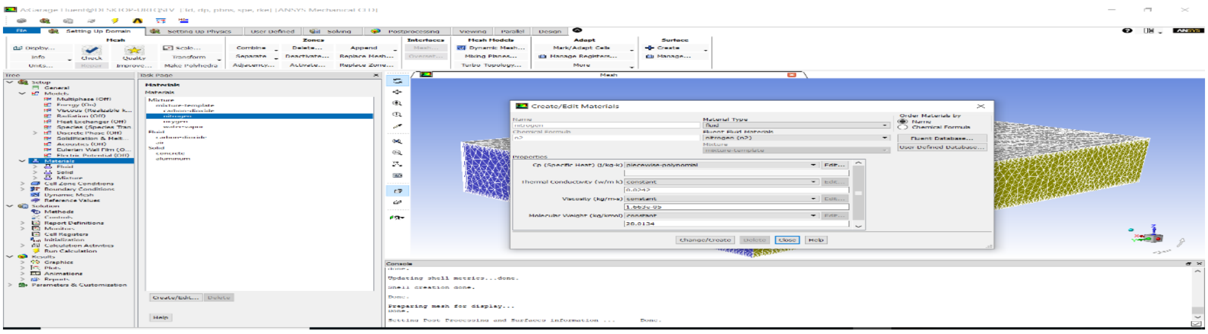
Mixture template

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

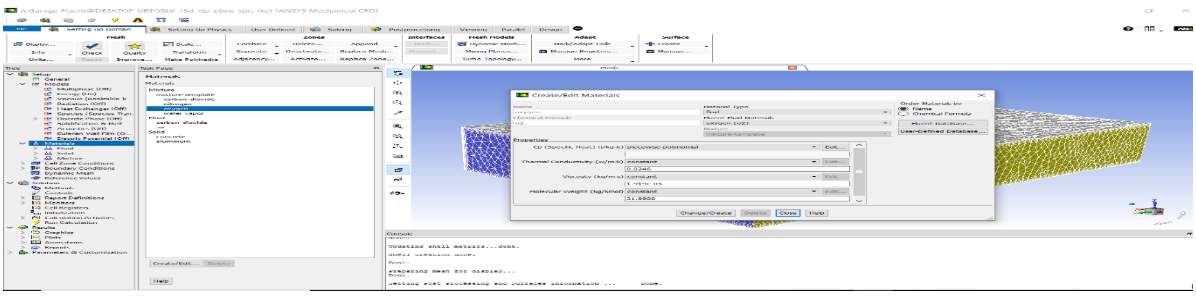
Research Article



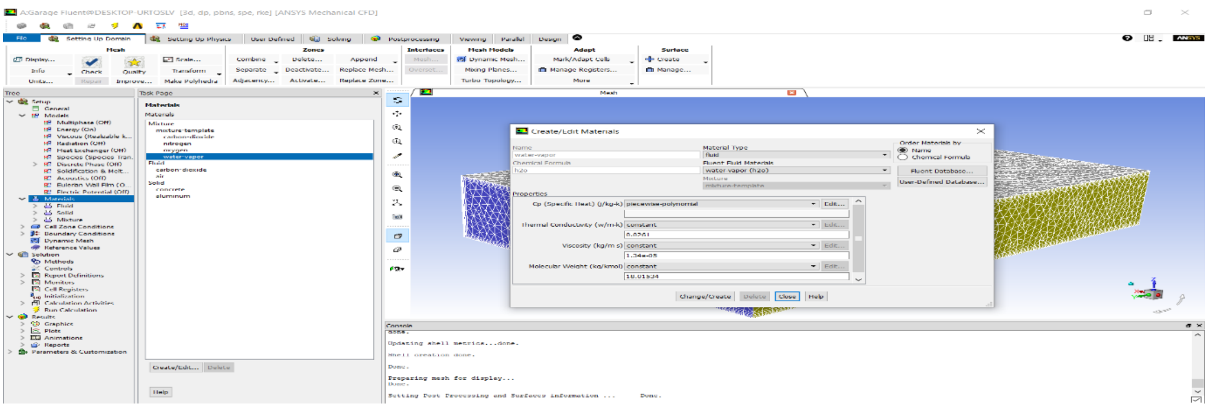
carbon dioxide



Nitrogen



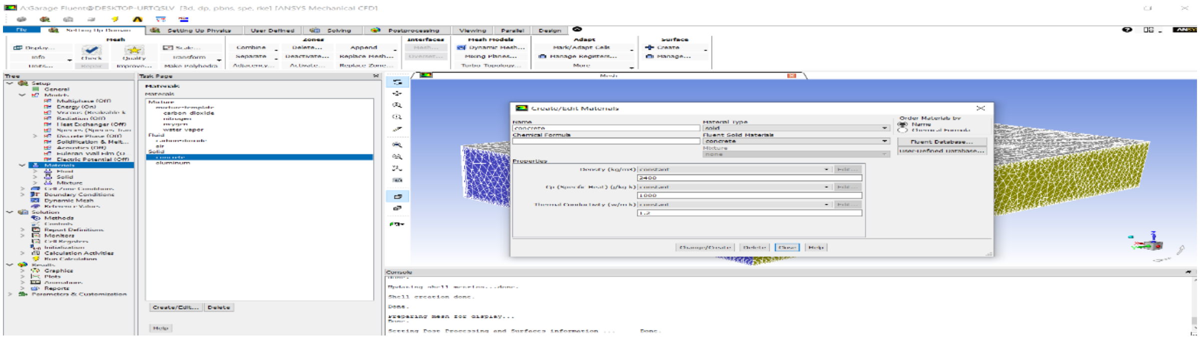
Oxygen



Water vapor

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

Research Article



Concrete

6. NUMERICAL METHOD

Finite Volume Method (FVM) used in ANSYS Fluent:

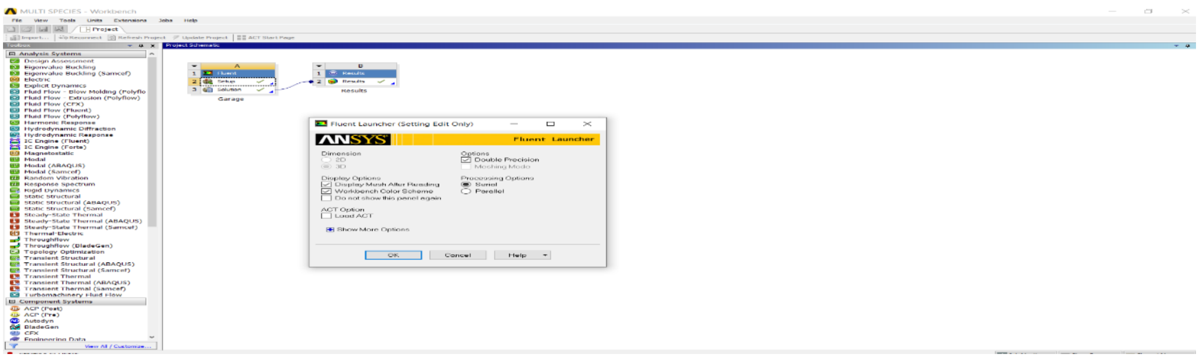
- Domain discretized into control volumes
- Governing equations integrated over each cell
- Algebraic equation system solved iteratively

7. GEOMETRY MODELING (SpaceClaim)

Geometry created in ANSYS SpaceClaim:

- Fire source region
- Ventilation ducts
- Outlet vents
- Enclosure walls

Structured/unstructured mesh applied.

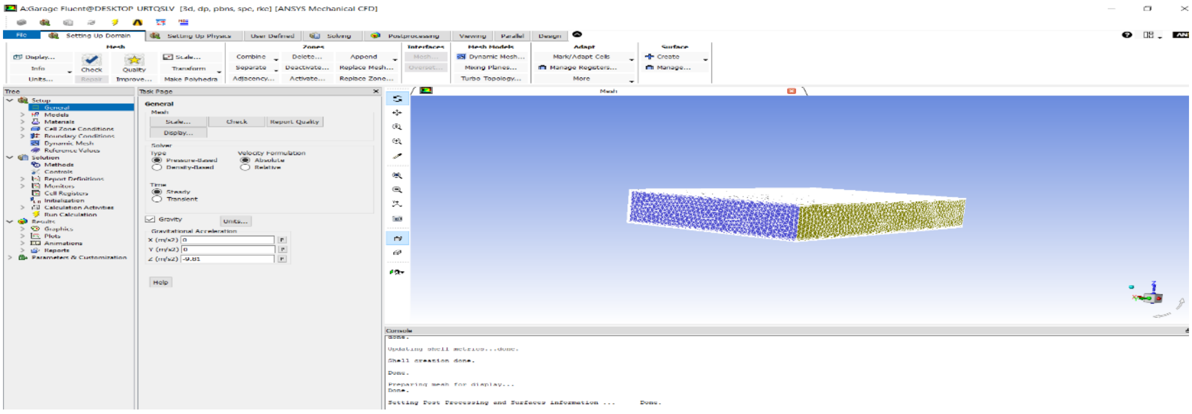


SpaceClaim Corporation markets SpaceClaim Engineer directly to end-user and indirectly by other channels. SpaceClaim also licenses its software for OEMs, such as ANSYS,[3] Flow International Corporation, CatalCAD, and Ignite Technology which markets a version of SpaceClaim for jewelry design.

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ **Research Article**

Finite element method:

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements.



8. MESH GENERATION

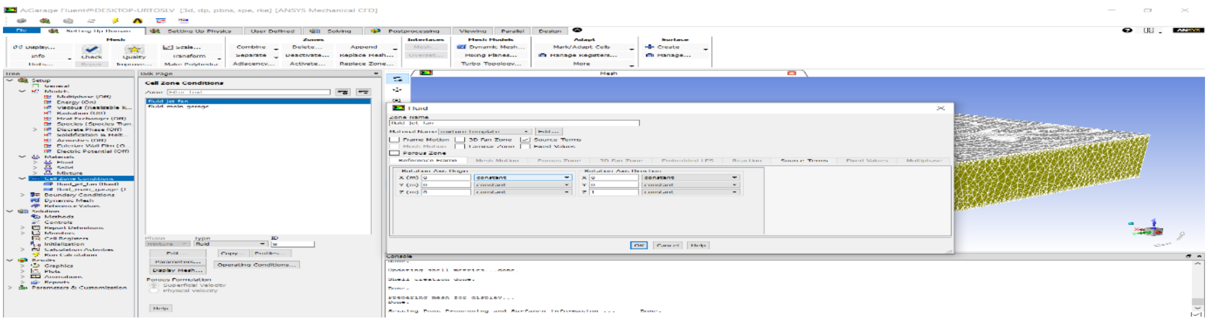
- Fine mesh near fire source
- Refined near ventilation openings
- Inflation layers near walls

Mesh independence study performed.

9. BOUNDARY CONDITIONS

Boundary	**Condition**
Fire Source	Heat flux / volumetric heat source
Inlet	Ambient air (22°C)
Outlet	Pressure outlet
Walls	No-slip + adiabatic

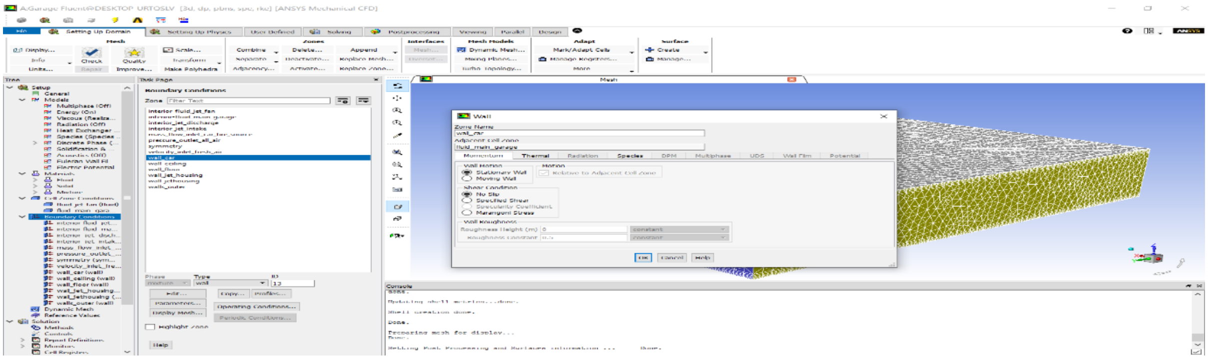
Gravity enabled for buoyancy modeling.



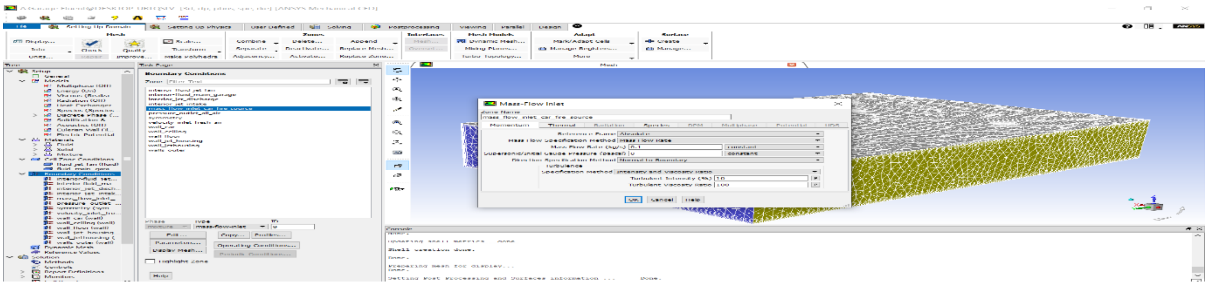
Fluid fan

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

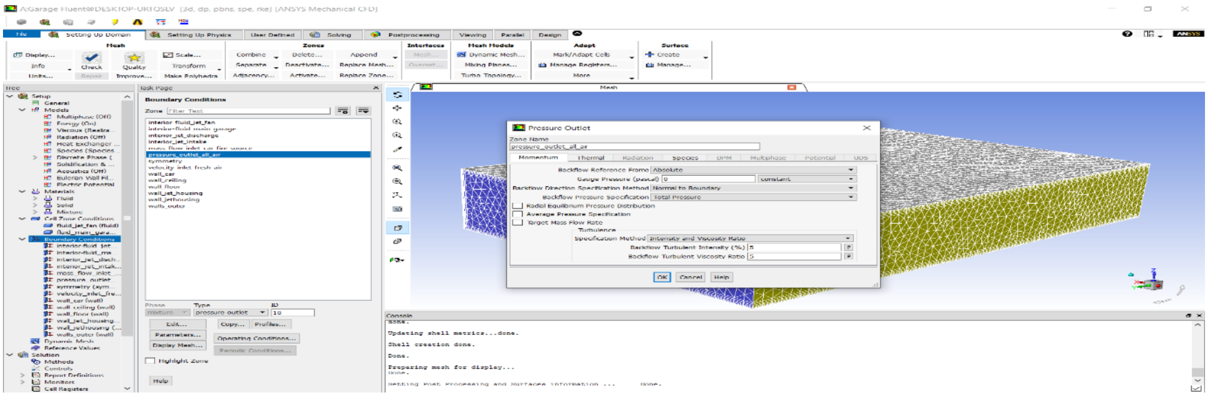
Research Article



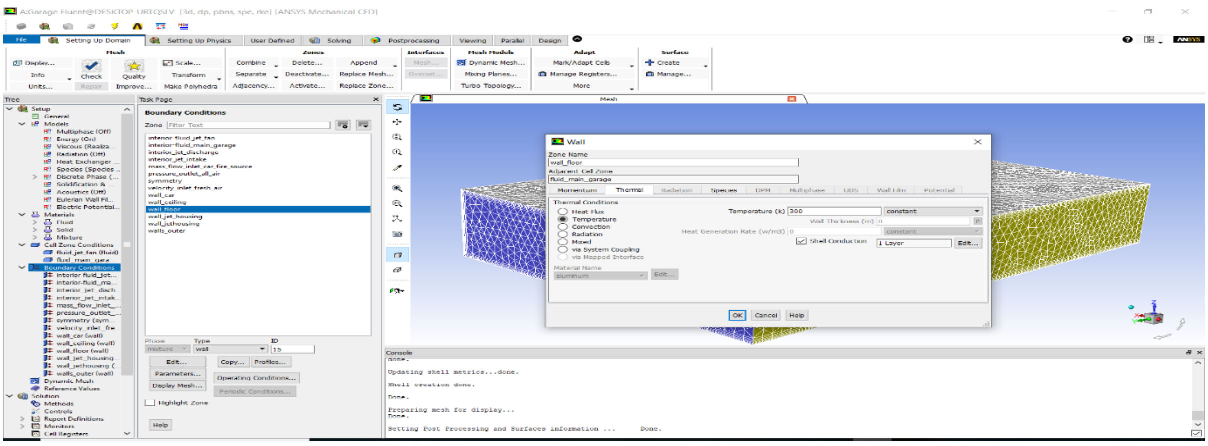
Inlet



Fire case

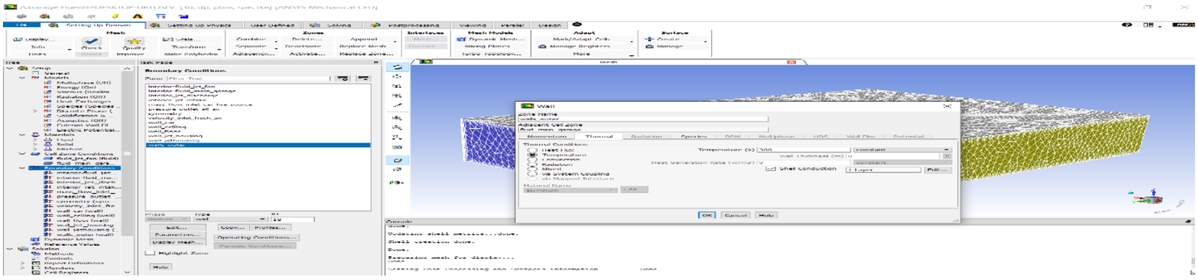


Pressure outlet

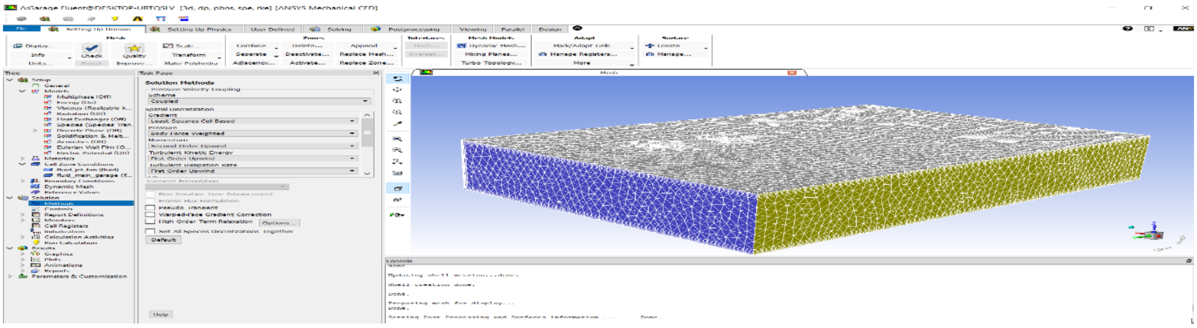


Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

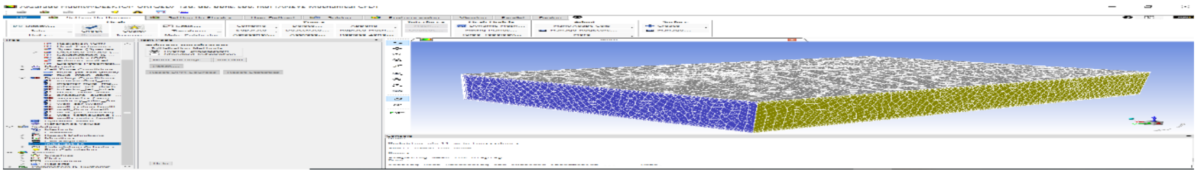
Research Article



Wall flow



Methods



Initialization

10. TURBULENCE MODEL

k–ε model used:

- Robust
- Suitable for buoyancy-driven turbulent flow
- Captures recirculation patterns

11. RESULTS – TEMPERATURE FIELD

- High-temperature plume forms above fire
- Stratified thermal layers observed
- Ceiling temperature highest
- Ventilation removes upper hot layer

12. SPECIES DISTRIBUTION

- CO_2 concentration increases near source

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ **Research Article**

- O_2 depletion observed
- Vertical stratification of gas mixture
- Ventilation significantly reduces concentration peaks

13. HEAT FLUX ANALYSIS

Heat transfer modes:

- Convection (dominant)
- Radiation (secondary)
- Conduction through walls

Heat flux highest near source and ceiling.

14. BUOYANCY EFFECT

As temperature increases:

$$\Delta\rho \rightarrow \Delta v$$

Strong upward plume formation.

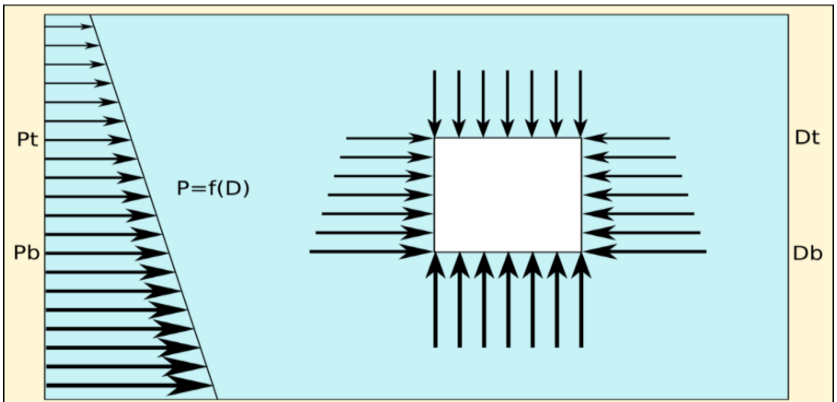
Recirculation zones form depending on vent placement.

Effect of buoyancy force on different profiles

A simplified explanation for the integration of the pressure over the contact area may be stated as follows:

Consider a cube immersed in a fluid with the upper surface horizontal.

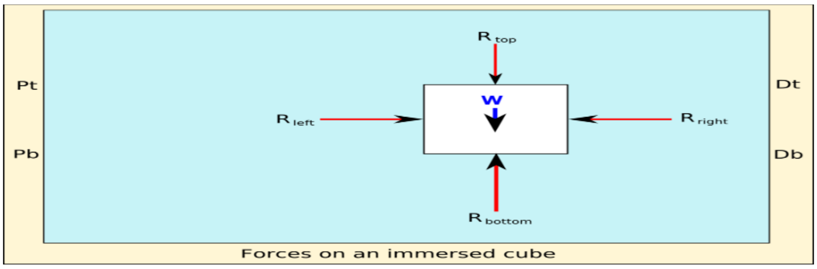
The sides are identical in area, and have the same depth distribution, therefore they also have the same pressure distribution, and consequently the same total force resulting from hydrostatic pressure, exerted perpendicular to the plane of the surface of each side.



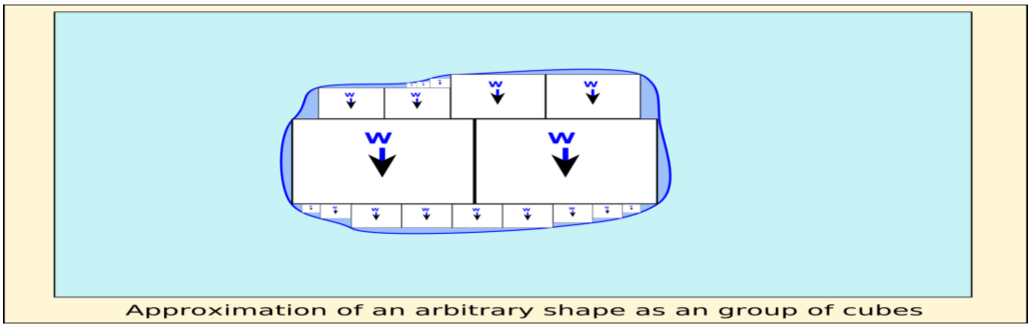
Pressure distribution on an immersed cube

Similarly, the downward force on the cube is the pressure on the top surface integrated over its area. The surface is at constant depth, so the pressure is constant. Therefore, the integral of the pressure over the area of the horizontal top surface of the cube is the hydrostatic pressure at that depth multiplied by the area of the top surface.

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ **Research Article**

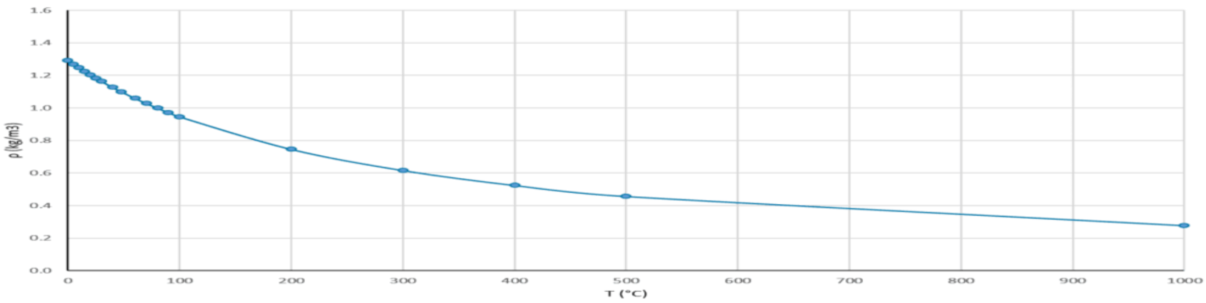


If two cubes are placed alongside each other with a face of each in contact, the pressures and resultant forces on the sides or parts thereof in contact are balanced and may be disregarded, as the contact surfaces are equal in shape, size and pressure distribution, therefore the buoyancy of two cubes in contact is the sum of the buoyancies of each cube. This analogy can be extended to an arbitrary number of cubes.



Applications of Buoyancy

As stated before, buoyancy should not be referred only to solid-in-liquid cases (e.g. ships in water). At least two big categories of applications can be directly linked to buoyancy effects: natural convection and multiphase flows. Natural convection is based on the fact that materials usually become less dense with an increase in temperature (see Figure 3 for example).

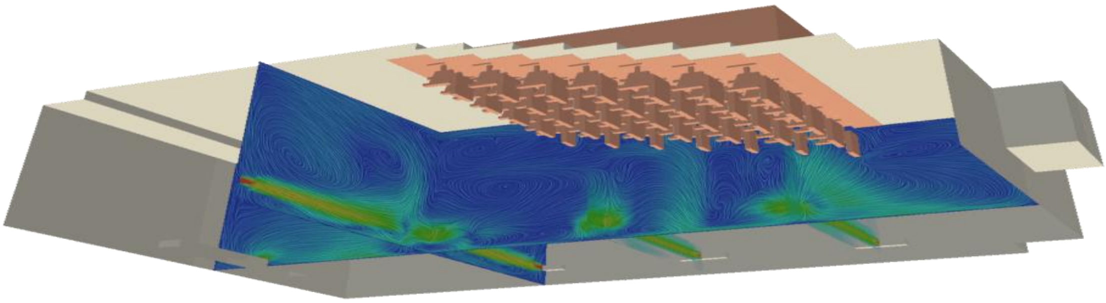


Example of density evolution of air with respect to temperature. Warmer fluids tend to have relatively lower densities and thus will tend to float.

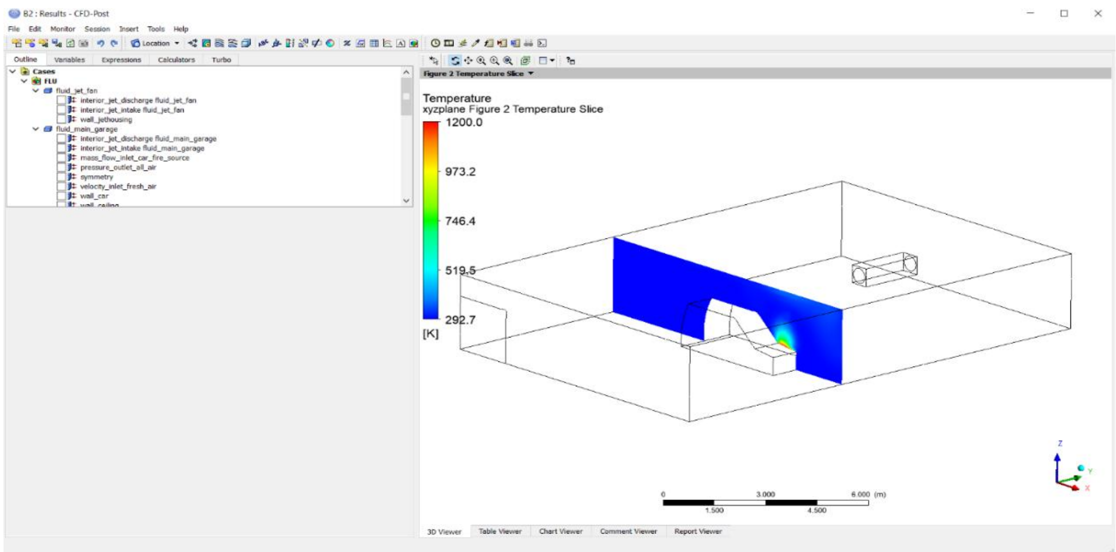
This means that a warm fluid will "float" when immersed in a region of the same fluid, but colder. In this case, we will not speak of "floating" or "sinking", but of upward streams of hot fluid and downward streams of cold fluid. This phenomenon is usually coupled with thermal analyses and it is the base of many applications such as meteorology, steel casting, and house warming/cooling. Natural convection is often enhanced by a forced flow imposition in order to obtain certain conditions. In this case, we speak of "forced convection", in which buoyancy forces are still present but less dominant. Figure 4 shows the natural convection induced by air conditioning in a theater space. The AC blows in cold air at the top of the space which streams down due to the upward buoyancy forces of the warm air at the bottom of the space. Thus, the cold air forms downward streams and the recirculation pattern is shown in the figure below.

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/

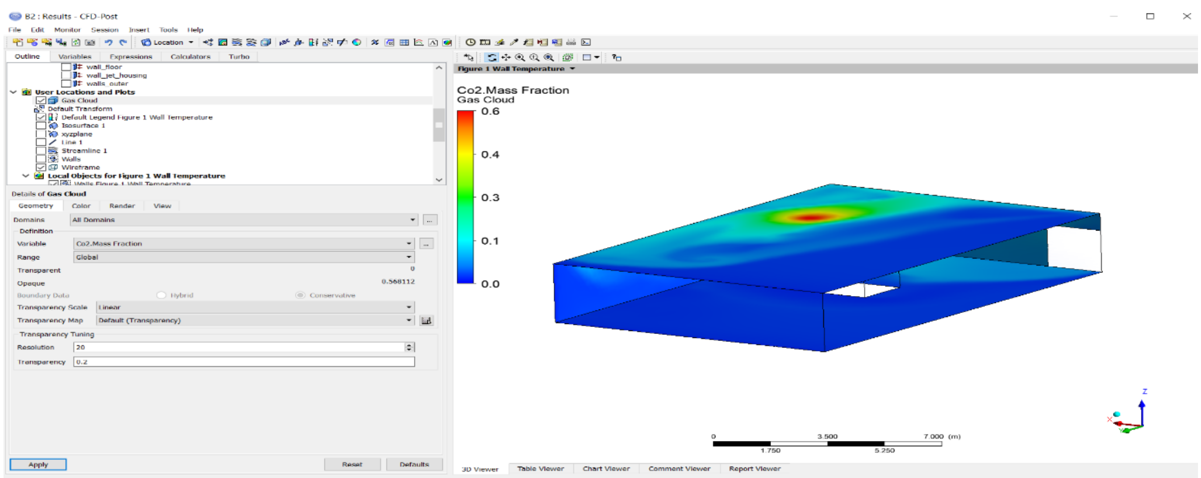
Research Article



Forced convection in a theater space with air flow recirculation patterns

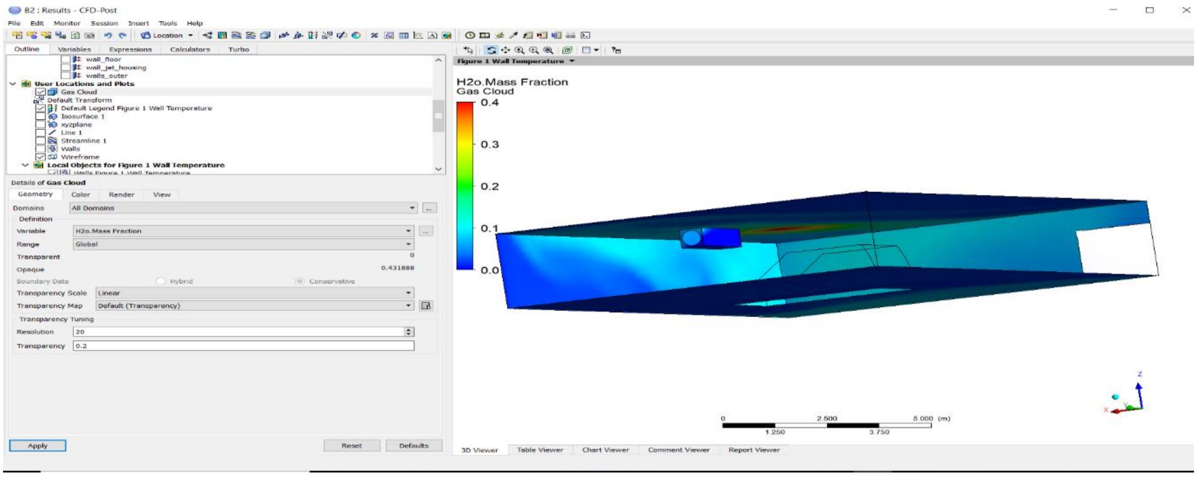


Temperature

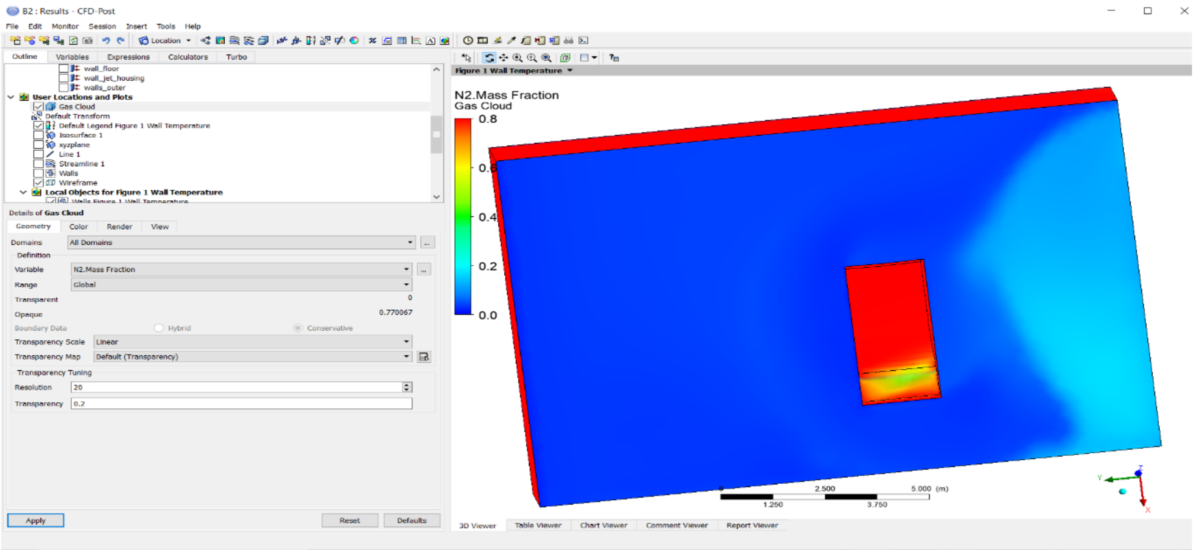


Co2 mass fraction

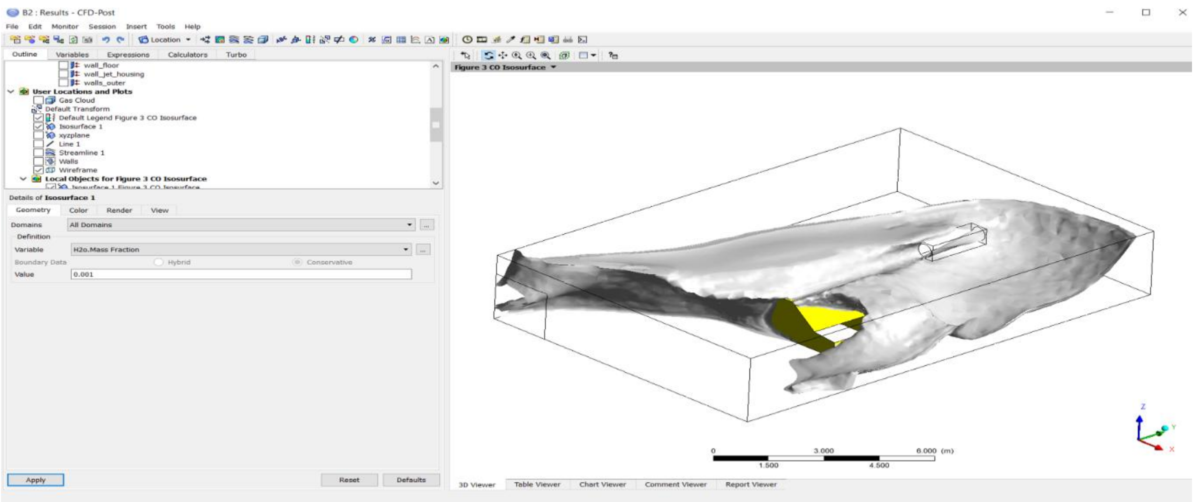
Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ Research Article



H2o mass fraction



N2 mass fraction



Iso surface – CO

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ Research Article

15. DISCUSSION

Key findings:

- Buoyancy dominates flow structure
- Multi-species transport strongly coupled to temperature
- Vent placement significantly affects smoke removal
- Heat flux concentrated near ceiling

16. ENGINEERING IMPLICATIONS

CFD analysis assists in:

- Smoke control system design
- Emergency ventilation optimization
- Fire safety regulation compliance
- Structural thermal assessment

17. CONCLUSION

This study successfully simulated multi-component gas flow and heat flux in a fire ventilation system using ANSYS Fluent.

Major conclusions:

1. Buoyancy is the primary driver of fire-induced flow.
2. Multi-species modeling is essential for realistic prediction.
3. Heat flux distribution strongly influences structural safety.
4. Ventilation geometry significantly affects smoke removal efficiency.
5. CFD provides a reliable and cost-effective design tool.

The developed framework can be extended to large industrial fire safety applications.

REFERENCES

[1] G Klár, T. Gast, A. Pradhana, C. Fu, C. Schroeder, C. Jiang, and J. Teran. 2016. Druckerprager Elastoplasticity for Sand Animation. ACM Trans Graph 35, 4 (2016), 103:1– 103:12. T.
[2] Lenaerts and P. Dutré. 2009. Mixing Fluids and Granular Materials. Comp Graph Forum 28, 2 (2009), 213–218. S. Liu, Z. Wang, Z. Gong, and Q. Peng. 2008. Simulation of Atmospheric Binary Mixtures Based on Two-fluid Model. Graph Mod 70, 6 (2008), 117–124.
[3] F. Losasso, J. Talton, N. Kwatra, and R. Fedkiw. 2008. Two-Way Coupled SPH and Particle Level Set Fluid Simulation. IEEE Trans Visu Comp Graph 14, 4 (2008), 797–804. P. Mackenzie-Helnwein, P. Arduino, W. Shin, J. Moore, and G. Miller. 2010.
[4] Modeling strategies for multiphase drag interactions using the material point method. Int J Num Meth Eng 83, 3 (2010), 295–322. C. Mast, P. Arduino, G. Miller, and M. Peter. 2014.
[5] Avalanche and landslide simulation using the material point method: flow dynamics and force interaction with structures.
[6] Comp Geosci 18, 5 (2014), 817–830. V. Mihalef, D. Metaxas, and M. Sussman. 2009. Simulation of two-phase flow with sub-scale droplet and bubble effects. Comp GraphForum 28, 2 (2009), 229–238. Ken Museth. 2013. VDB: High-resolution Sparse Volumes with Dynamic Topology.

Journal of Information Systems Engineering and Management
2026, 11(2s)
e-ISSN: 2468-4376
https://www.jisem-journal.com/ **Research Article**

[7] ACM Trans. Graph. 32, 3, Article 27 (July 2013), 22 pages. R. Narain, A. Golas, and M. Lin. 2010. Free-flowing granular materials with two-way solid coupling. ACM Trans Graph 29, 6 (2010), 173:1–173:10. M. Nielsen and O. Osterby. 2013.
[8] A Two-continua Approach to Eulerian Simulation of Water Spray. ACM Trans Graph 32, 4 (2013), 67:1–67:10. D. Peachey. 1986. Modeling Waves and Surf. SIGGRAPH Comput Graph 20, 4 (1986), 65–74. D. Ram, T. Gast, C. Jiang, C. Schroeder, A. Stomakhin, J. Teran, and P. Kavehpour. 2015.
[9] A material point method for viscoelastic fluids, foams and sponges. In Proc ACM SIGGRAPH/Eurograph Symp Comp Anim. 157–163. B. Ren, Y. Jiang, C. Li, and M. Lin. 2015. A simple approach for bubble modelling from multiphase fluid simulation. Comp Vis Media 1, 2 (2015), 171–181. B. Ren, C. Li, X. Yan, M. Lin, J. Bonet, and S. Hu. 2014.
[10] Multiple-Fluid SPH Simulation Using a Mixture Model. ACM Trans Graph 33, 5 (2014), 171:1–171:11. D. Robert and K. Soga. 2013. Soil-Pipeline Interaction in Unsaturated Soils.
[11] In Mechanics of Unsaturated Geomaterials, Lyesse Laloui (Ed.). Wiley Online Library, Chapter 13, 303–325. W. Rungjiratananon, Z. Szego, Y. Kanamori, and T. Nishita. 2008. Real-time Animation of Sand-Water Interaction. Comp Graph Forum 27, 7 (2008), 1887–1893.
[12] O. Song, H. Shin, and H. Ko. 2005. Stable but Nondissipative Water. ACM Trans Graph 24, 1 (2005), 81–97. A. Stomakhin, C. Schroeder, L. Chai, J. Teran, and A. Selle. 2013.
[13] A Material Point Method for snow simulation. ACM Trans Graph 32, 4 (2013), 102:1–102:10. A. Stomakhin, C. Schroeder, C. Jiang, L. Chai, J. Teran, and A. Selle. 2014.
[14] Augmented MPM for phase-change and varied materials. ACM Trans Graph 33, 4 (2014), 138:1– 138:11. D. Sulsky, S. Zhou, and H. Schreyer. 1995. Application of a particle-in-cell method to solid mechanics.
[15] Comp Phys Comm 87, 1 (1995), 236–252. T. Takahashi, H. Fujii, A. Kunimatsu, K. Hiwada, T. Saito, K. Tanaka, and H. Ueki. 2003. Realistic Animation of Fluid with Splash and Foam.
[16] Comp Graph Forum 22, 3 (2003), 391–400. N. Thürey, F. Sadlo, S. Schirm, M. Müller-Fischer, and M. Gross. 2007. Real-time Simulations of Bubbles and Foam Within a Shallow Water Framework.
[17] In Proc ACM SIGGRAPH/Eurograph Symp Comp Anim. Eurographics Association, 191–198. L. Yang, S. Li, A. Hao, and H. Qin. 2014. Hybrid Particle-grid Modeling for Multi-scale Droplet/Spray Simulation.
[18] Comp Graph Forum 33, 7 (2014), 199–208. T. Yang, J. Chang, B. Ren, M. Lin, J. Zhang, and S. Hu. 2015. Fast Multiple-fluid Simulation Using Helmholtz Free Energy.