

CFD analysis of thermal plume behavior under diagonal heat source orientation

Received: 6 April 2026

Accepted: 2 July 2026

Published online: 10 July 2026

Cite this article as: Deshmukh S.M., Naidu M., Pal J.S. *et al.* CFD analysis of thermal plume behavior under diagonal heat source orientation. *Sci Rep* (2026). <https://doi.org/10.1038/s41598-026-61194-2>

Sagar Mane Deshmukh, Mithul Naidu, Jitendra Singh Pal, Sanjeet Kanungo, Ajaj Attar, Ravi Prakash Singh, Sachin Salunkhe, Robert Čep & Emad Abouel Nasr

We are providing an unedited version of this manuscript to give early access to its findings. Before final publication, the manuscript will undergo further editing. Please note there may be errors present which affect the content, and all legal disclaimers apply.

If this paper is publishing under a Transparent Peer Review model then Peer Review reports will publish with the final article.

ARTICLE IN PRESS

CFD Analysis of Thermal Plume Behavior under Diagonal Heat Source Orientation

¹Sagar Mane Deshmukh, *² Mithul Naidu, ³Jitendra Singh Pal, ⁴ Sanjeet Kanungo ⁵Ajaj Attar, Ravi Prakash Singh⁶, Sachin Salunkhe^{7*}, Robert Čep⁸, Emad Abouel Nasr^{9*}

¹Sr. Assistant Professor, Mechanical Engineering Department, Tolani Maritime Institute, Induri, Pune-410507

² Associate Professor, Mechanical Engineering Department, Tolani Maritime Institute, Induri, Pune-410507

³Sr. Assistant Professor, Marine Engineering Department, Tolani Maritime Institute, Induri, Pune-410507

⁴ Professor, Marine Engineering Department, Tolani Maritime Institute, Induri, Pune-410507

⁵Sr. Assistant Professor, Mechanical Engineering Department, Tolani Maritime Institute, Induri, Pune-410507

⁶Mathematics Institute, University of Warwick, Coventry CV4 7AL, UK
ravi.p.singh@warwick.ac.uk

^{7a}Department of Biosciences, Saveetha School of Engineering, Saveetha Institute of Medical and Technical Sciences, Chennai, India

^{7b}Department of Mechanical Engineering, Gazi University Faculty of Engineering, Maltepe, Ankara, Turkey

⁸Department of Machining, Assembly and Engineering Metrology, Faculty of Mechanical Engineering, VSB-Technical University of Ostrava, 17. Listopadu 2172/15, 708 00 Ostrava, Czech Republic;

⁹Department of Industrial Engineering, College of Engineering, King Saud University, P.O. Box 800, Riyadh 11421, Saudi Arabia,
eabdelghany@ksu.edu.sa

*Corresponding Author: Sachin Salunkhe sachinsalunkhe@gazi.edu.tr;
Mithul Naidu mjn2979@gmail.com; Emad Abouel Nasr
eabdelghany@ksu.edu.sa

Abstract:

Confined fire environments in marine and industrial compartments present serious safety challenges due to rapid heat accumulation, restricted ventilation, and complex buoyancy-driven flow behavior. Accurate prediction of propagation of heat and development of plume is important for the effective and efficient design of thermal detection and mist-based fire suppression systems. In the present study, a 2D transient Computational Fluid Dynamics (CFD) analysis has performed to investigate buoyancy-driven heat transfer generated by heat source of 0.5 m diameter maintained at 450 °C inside a closed container of dimensions 2.6 m × 2.6 m. The air was considered as the working fluid, and natural convection was modeled to capture plume rise, entrainment, and thermal stratification effects. The heat source was sequentially positioned at four diagonal locations: D1 (0.65 m, 0.65 m), D2 (1.95 m, 0.65 m), D3 (1.95 m, 1.95 m), and D4 (0.65 m, 1.95 m) to assess the influence of source orientation relative to enclosure (Container) boundaries. The numerical methodology incorporates mesh-independence verification, transient time-step control with Courant numbers maintained below 2, appropriate turbulence modeling, and strict mass conservation to ensure numerical accuracy. The simulation reveals the formation of strong buoyancy-driven thermal plumes with peak velocities in the range of 0.9 – 1.3 m/s. Lower heat source configurations produce tall vertical plumes, while upper source placements result in early plume impingement and the development of ceiling jets, significantly enhancing lateral heat transport. Temperature and density fields indicate localized high-temperature zones near the heat source, with air density reducing to approximately 0.5 – 0.7 kg/m³ within plume cores. Turbulent kinetic energy reaches peak values of the order of 10⁻² m²/s² along plume shear layers and ceiling interaction regions, reflecting intense mixing. Pressure and mass imbalance analyses confirm stable transient behavior, with mass imbalance remaining within $\pm 10^{-7}$ kg/s. The findings present design trends under idealized enclosed conditions, which may support future development of fire detection and suppression strategies for marine and industrial compartments after further validation under realistic operating conditions.

.Keywords:

Confined fire dynamics, Buoyancy-driven thermal plume, Computational fluid dynamics (CFD), Thermal sensor optimization, Water mist fire suppression

Nomenclature

ρ – Density of the Fluid, kg/m³

μ – Dynamic viscosity of the fluid, Pa.s

u – Velocity of the fluid, m/s

h – Enthalpy of the fluid, kJ/kg

k – Thermal Conductivity of the fluid, W·m⁻¹·K⁻¹

t - Time involved in the process, Sec
 T_s - Source temperature, K
 T_∞ - Air temperature inside the container, K
 β - Thermal expansion coefficient, K^{-1}
 ν - Kinematic viscosity, m^2/s
 L - Characteristic length, m
 Gr - Grashof's number
 Pr - Prandtl number
 Ra - Rayleigh Number
 P - Pressure in the fluid, Pa
 u - X- directional velocity, m/s
 v - Y- directional velocity, m/s
 μ_t - Turbulent eddy viscosity, $Pa \cdot s$
 P_k - Turbulence production by mean shear
 C - CFL constant
 Δx - Mesh size
 u - Velocity
 y^+ - Desired non-dimensional wall distance
 ν - Kinematic viscosity
 u_τ - Friction velocity
 g_y - Y-directional gravity
 k - Turbulent Kinetic Energy
 ω - Specific Dissipation Rate
 k_t - Turbulent thermal conductivity
 Pr_t - Turbulent Prandtl number
 $D\omega$ - Cross-diffusion term
 σ_k - Turbulent Prandtl number for 'k'
 σ_ω - Turbulent Prandtl number for ' ω '
 F_1 - Near-wall blending function
 F_2 - Free-stream blending function
 α - Production coefficient in the ω -equation
 β^* - Dissipation coefficient in the turbulent kinetic energy equation
 β - Dissipation coefficient in the specific dissipation rate equation
 a_1 - Eddy viscosity limiter constant
CFD - Computational fluid dynamics
FDS - Fire dynamics simulator

1. Introduction and literature survey

Fire-induced buoyancy-driven heat transfer in confined enclosures remains a critical safety concern in marine, industrial, transportation, and energy

systems due to rapid temperature rise, smoke accumulation, restricted ventilation, and limited evacuation possibilities [1]–[3]. Once ignition occurs within an enclosed domain, interactions among heat release, buoyant plume development, turbulence generation, radiation effects, and enclosure boundaries govern the evolution of temperature, pressure, density, and velocity fields [1], [2], [16]. These coupled processes strongly influence fire growth, smoke propagation, thermal stratification, and performance of detection and suppression systems [3], [6].

Fundamental theories of turbulent transport and thermal-fluid interactions have established that convection dominates heat transfer in buoyancy-driven flows, while radiative contributions become increasingly important at elevated temperatures [15], [16]. Early analytical and experimental studies on compartment fires demonstrated that buoyant plumes dominate heat transfer during the initial combustion stages, with convective transport greatly exceeding conductive mechanisms within gaseous phases [1], [2]. Under confined conditions, plume characteristics deviate from classical free-plume behavior because of reduced entrainment, wall confinement, and ceiling interactions, producing accelerated plume rise and earlier ceiling-jet formation [7], [36]. Smoke accumulation and hot gas movement in enclosed spaces strongly depend on enclosure geometry and source location [3], [6].

Understanding transient heat transfer and turbulent flow behavior in confined geometries has attracted increasing attention beyond conventional fire safety research. Recent investigations involving turbulent wall jets, impinging jets, and heat transfer over heated surfaces highlighted the complex coupling among turbulence intensity, boundary interactions, and localized thermal gradients [58]–[62]. The studies of turbulent wall jet impingement and twin swirling jets demonstrated that recirculation zones and impingement effects substantially modify heat transfer mechanisms [58], [62]. Similar phenomena may occur in confined fire environments where buoyant plumes interact with enclosure boundaries and ceiling jets [38], [40].

Advances in computational resources have enabled Computational Fluid Dynamics (CFD) to become the primary tool for analyzing transient fire behavior in confined spaces. Governing equations for conservation of mass, momentum, and energy, combined with turbulence and buoyancy modeling, allow detailed resolution of plume structures and thermal stratification [12], [14]. Theoretical formulations of CFD and turbulence transport remain central to enclosure fire modeling [12], [15]. Quintiere [1] and Jones [2] emphasized that accurate representation of buoyancy-induced flow is essential for predicting plume rise and ceiling temperatures, while recent reviews highlighted CFD as indispensable for compartment fire prediction and optimization of fire safety systems [57].

Several numerical investigations focused on thermal plume dynamics within cubical and rectangular enclosures, emphasizing effects of enclosure dimensions and heat-source positioning [7], [17], [18]. Hosseini and Fakhri [7] demonstrated significant variations in plume height, entrainment rate, and temperature decay with changing heat source placement. Similar findings were reported by Zheng and Wang [10] and Yu et al. [17], indicating that offset and corner-mounted heat sources generate asymmetric temperature fields and intensified thermal stratification. Recent studies on compartment heat propagation further confirmed strong dependence of temperature evolution on enclosure geometry and source placement [18], [34], [37]. Corner fires exhibit accelerated plume growth owing to restricted entrainment [36], [43], [44].

Thermal transport enhancement due to localized turbulence and geometric modifications has also been explored in studies involving phase change materials (PCM) and impinging jets [59]–[61]. Investigations reported that inclined surfaces and jet impingement significantly alter melting rates and transient thermal responses [59]–[61]. Such findings are relevant to confined fire scenarios where enclosure geometry and internal structures alter plume trajectories and heat accumulation patterns.

Turbulence modeling remains critical for accurate prediction of buoyancy-driven plume behavior. Comparative studies on k - ε and k - ω models demonstrated satisfactory prediction of natural convection plumes under appropriate mesh and timestep conditions [8], [39]. Although Large Eddy Simulation (LES) provides improved representation of transient turbulence and combustion phenomena [13], [46], computational expense restricts its use in extensive parametric studies. Recent investigations into transient turbulent plume modeling have therefore continued to employ RANS-based approaches for engineering applications [45]. Consequently, transient Reynolds-Averaged Navier-Stokes approaches remain widely used in enclosure fire simulations [45], [57].

Numerical stability and solution accuracy are essential considerations in transient CFD analysis. Studies highlighted the importance of Courant number control, grid independence assessment, and convergence criteria in buoyancy-dominated thermal simulations [9], [20], [21]. Mesh quality metrics strongly influence plume stability and numerical accuracy [49]. Verification and validation frameworks have been proposed to minimize numerical uncertainty and improve confidence in simulation predictions [47], [48]. Coupled conduction-convection simulations also require careful accuracy assessment under transient conditions [51].

Thermal stratification and ceiling-layer formation constitute important characteristics of compartment fires. Experimental and numerical investigations demonstrated rapid formation of hot gas layers beneath enclosure ceilings, especially under elevated or offset heat sources [19], [31], [32]. Ceiling jets strongly influence lateral heat transport, resulting in non-uniform temperature distributions and delayed heating of lower regions [33], [38]. Similar jet-induced heat transfer enhancement mechanisms have also been observed in turbulent impinging flow studies [58], [62]. Boundary layer behavior near heated surfaces additionally affects thermal transport and plume development [40].

Optimization of thermal sensor placement has therefore emerged as an active research area. CFD-based investigations reported that sensor activation time strongly depends on sensor location relative to plume trajectories and local thermal gradients [11], [27], [55]. Sensors positioned within regions of high temperature gradient respond substantially faster than those located in thermally uniform regions [35]. Sensor performance evaluation using CFD has become increasingly important for enclosed fire safety applications [53], [55].

Parallel to fire detection systems, water-mist suppression technologies have gained significant attention in marine and confined applications due to superior cooling capability and regulatory acceptance [5], [22], [26], [52]. Studies demonstrated that interactions between mist droplets and fire-induced plumes depend strongly on turbulence intensity, plume velocity, and density gradients [4], [41], [42]. Investigations on marine engine-room and machinery-space fires showed suppression efficiency to be highly sensitive to nozzle positioning and spray orientation [23]–[25]. Full-scale experiments further investigated thermal control characteristics of water mist systems under single- and multiple-source fire conditions [56].

Recent CFD analyses revealed that mist nozzles aligned with dominant plume trajectories and ceiling jets exhibit improved cooling and suppression performance [29], [30]. Hyper-mist deployment frameworks emphasized targeting regions with elevated turbulence kinetic energy to maximize droplet-gas interaction [28]. However, suppression efficiency may reduce considerably when droplets are introduced into recirculation-dominated zones [41]. Marine fire safety regulations and suppression standards provide performance-based criteria but offer limited guidance regarding optimized sensor and nozzle placement under transient conditions [5], [26], [52].

Recent benchmark datasets and validation studies highlighted the growing need for integrated CFD-based methodologies supporting performance optimization in confined compartments [50], [54], [65]. Benchmarking

against experimental data is increasingly recognized as essential for validating transient fire simulations [54]. Reference thermophysical property data and engineering standards continue to support fire safety design methodologies [50]. Recent numerical studies have extensively investigated buoyancy-induced flow and heat transfer behavior in enclosed cavities under different thermal boundary conditions [63]. For instance, studies on magnetohydrodynamic buoyancy-driven flow in non-isothermally heated square enclosures demonstrated the significant influence of Rayleigh and Hartmann numbers on temperature distribution and flow circulation patterns. Similarly, three-dimensional investigations of buoyancy-induced flow in nanofluid-filled cavities with internal obstacles highlighted the effects of enclosure geometry and obstacle configuration on heat transfer and entropy generation[64].

From the reviewed literature, it is evident that buoyancy-driven plume dynamics, thermal sensor response, turbulence modeling, and water-mist suppression have largely been investigated independently [1]-[57]. Furthermore, recent developments in turbulent impinging jets, wall jets, and phase change heat transfer reveal additional mechanisms governing transient thermal behavior in confined systems [58]-[62]. Nevertheless, a significant research gap persists regarding integrated transient analyses linking heat source location with coupled evolution of temperature, pressure, density, velocity, turbulence characteristics, sensor activation, and Hyper Mist suppression effectiveness within confined enclosures. In particular, effects of corner and offset heat sources on optimized sensor-nozzle placement strategies remain insufficiently quantified under transient conditions. The present study addresses this gap through comprehensive transient CFD simulations aimed at systematically connecting plume dynamics with practical guidance for thermal sensor and Hyper Mist nozzle placement in confined marine and industrial environments.

2. CFD Methodology and Model Setup

2.1 Geometry Description

Figure 1 illustrates the geometric configuration of the computational domain and the prescribed locations of the heat source within the confined container. The enclosure is a square container of internal dimensions $2.6 \text{ m} \times 2.6 \text{ m}$, representing a simplified two-dimensional confined compartment. The square boundary denotes the rigid, sealed walls of the container used in the CFD simulations. The star symbols indicate the centers of the circular heat source (0.5 m diameter) placed at four distinct diagonal locations inside the enclosure to study the effect of source positioning on heat propagation and plume development. These locations are intentionally chosen to be offset from both the geometric center and the walls, ensuring symmetric yet non-central placement.

Table 1: Positions of the diagonally mounted heat sources in container

Diagonals	X-Coordinate (m)	Y-Coordinate (m)	Heat source position in the container
D1	0.65	0.65	located near the lower-left quadrant
D2	1.95	0.65	located near the lower-right quadrant
D3	1.95	1.95	located near the upper-right quadrant
D4	0.65	1.95	located near the upper-left quadrant

Each position lies midway between the container center and the respective corner, maintaining equal offset distances of 0.65 m from the adjacent walls and 1.95 m from the opposite walls. This arrangement enables a systematic assessment of diagonal asymmetry effects on buoyancy-driven thermal plume formation, temperature stratification, and wall interactions. By keeping the enclosure geometry and heat source characteristics constant while varying only the location, the figure establishes a controlled framework for isolating the influence of source position on transient heat spread, plume height, and thermal field evolution - critical parameters for optimizing thermal sensor placement and hyper-mist nozzle distribution in confined fire safety applications.

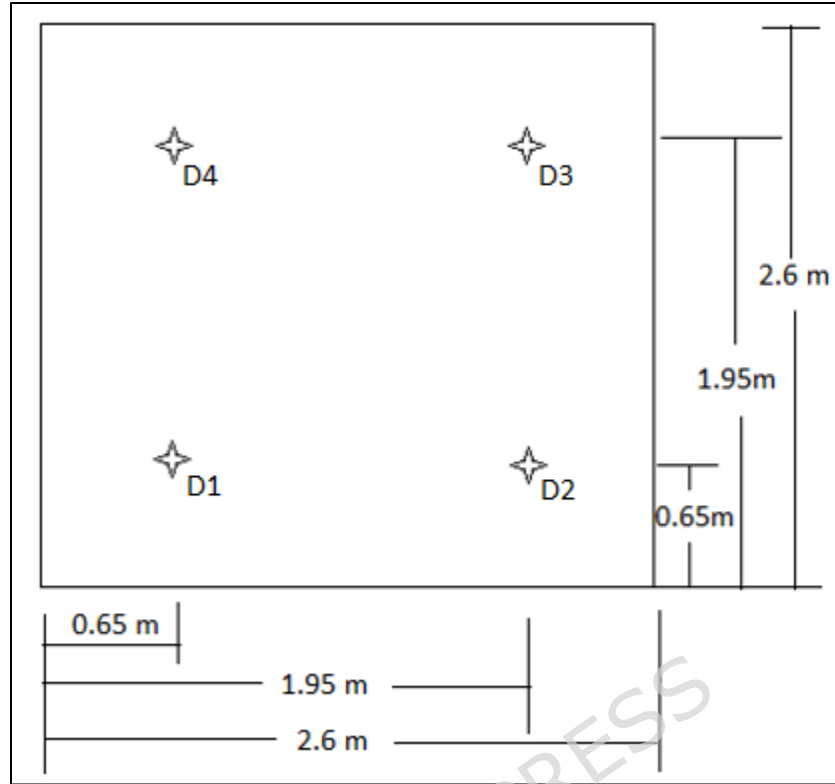


Figure 1: Positions of the heat source in the container.

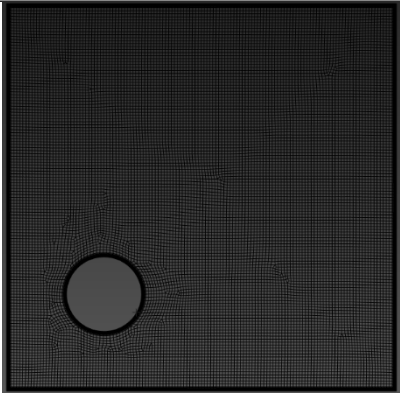
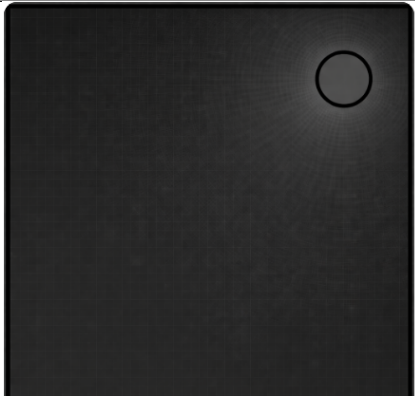
Note: Coordinates of the heat source position are as follows:

D1(0.65m,0.65m, D2(1.95m, 0.65m), D3(1.95m,1.95m), D4(0.65m,1.95m)

The domain contains only air initially at ambient conditions (27 °C, 1 atm). No openings exist on walls, simulating an enclosed real shipboard space.

2.2 Meshing Strategy and Grid Sensitivity

Table 2: Mesh files

Diagonal positions	Mesh file	Diagonal positions	Mesh file
D1 (x=0.65m, y=0.65m)		D2 (x=1.95m, y=0.65m)	

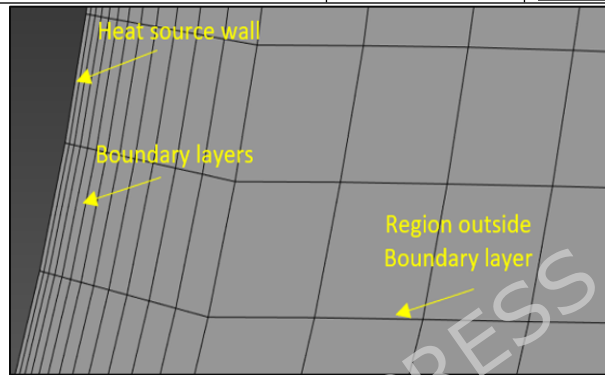
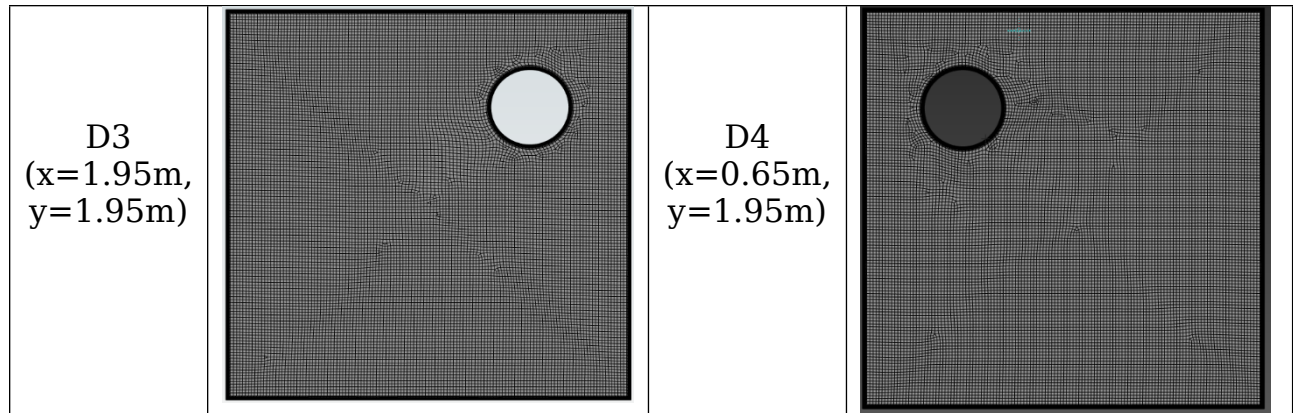


Figure 2: Boundary layers (12 in numbers) applied on heat source wall

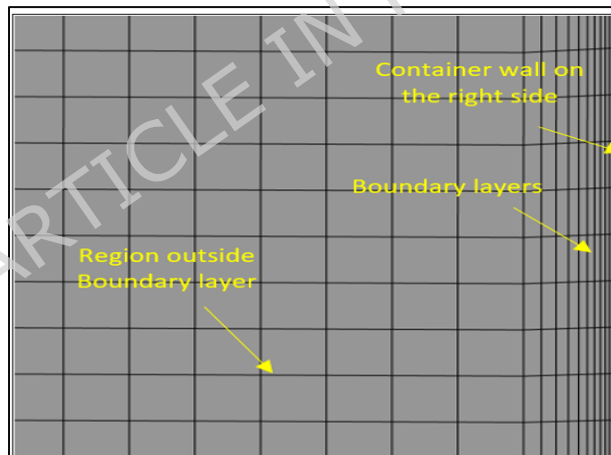


Figure 3: Boundary layers (12 in numbers) applied on container wall (right), (Similarly, it is added on container left wall, top wall and bottom wall which is not shown here)

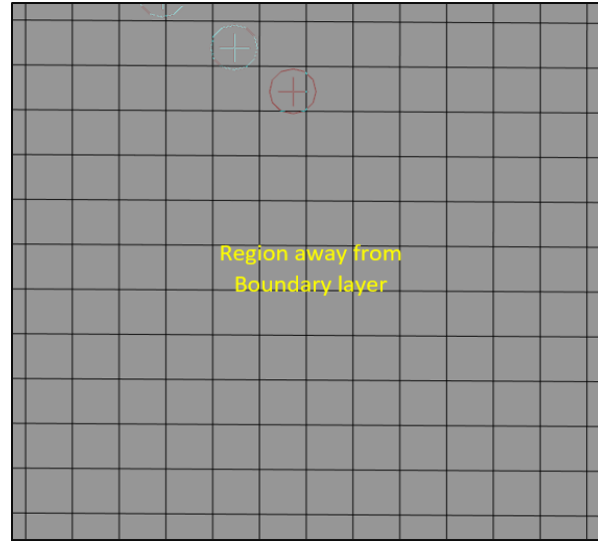


Figure 4: Quad mesh in the air domain away from boundary walls of heat source and container wall

Table 3: Mesh independence study parameters

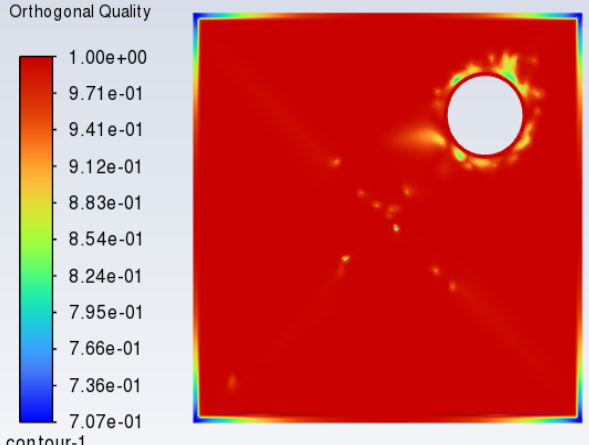
Sr. No.	Mesh with global element size	Number of nodes	Number of Elements	Inflation layers	Transition ratio	Growth rate	Area weighted Temperature at probe point (0.3,0), °C	Temp at point
1	0.03	18174	17827	12	0.272	1.2	47	102
2	0.02	27659	27299	12	0.272	1.2	46.3	101.5
3	0.01	79630	79041	12	0.272	1.2	46	101

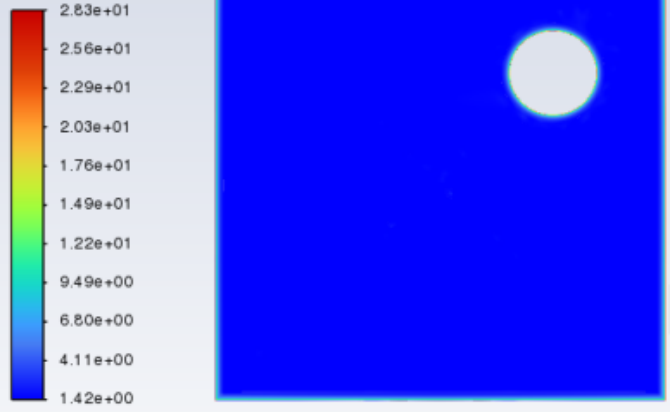
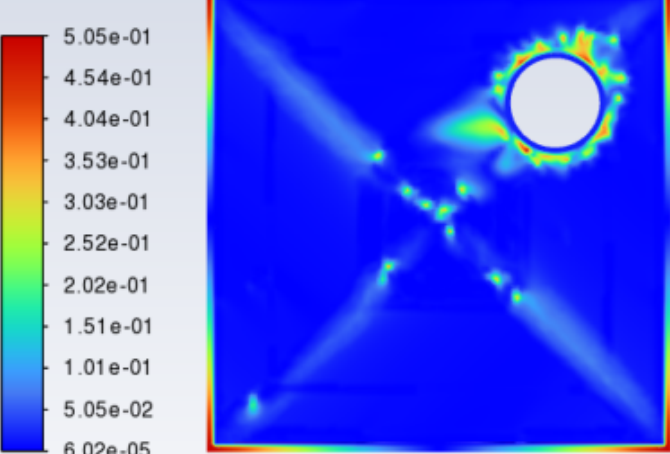
The mesh independence study was performed using three global element sizes, 0.03 m, 0.02 m, and 0.01 m (as shown in Figure 2 to 4), to evaluate the sensitivity of the numerical solution to mesh refinement. Temperature data were extracted at two probe locations, (0.3, 0) and (0, 0.5), which represent regions of strong convective effects respectively. The results showed that as the mesh was refined from 0.03 m to 0.01 m, the temperature variations at both probe points decreased significantly: the temperature at (0.3,0) varied from 47 °C → 46.3 °C → 46 °C, while the temperature at (0,0.5) changed from 102 °C → 101.5 °C → 101 °C. The maximum deviation between the 0.02 m and 0.01 m meshes is less than 2%, indicating that further refinement does not produce meaningful improvements in accuracy.

The probe-point temperature-versus-time plots (refer to figures 6 and 7) also show nearly identical transient trends for the 0.02 m and 0.01 m meshes, confirming that the flow and thermal structures are captured consistently. In contrast, the 0.03 m mesh shows slightly larger fluctuations due to under-resolution of near-wall gradients. Based on these observations, the 0.02 m global element size provides an optimal balance between numerical accuracy and computational cost, achieving mesh-independent behavior while maintaining simulation efficiency. Thus, the mesh size of 0.02 m was selected for all further simulations to ensure reliable results within the targeted 2% accuracy margin.

Figures 2 to 5 illustrate the meshing strategy adopted for the two-dimensional heat-spreading simulation. A structured quadrilateral mesh was generated throughout the square container domain to maintain element regularity and minimize numerical diffusion. The mesh is globally refined with an element size of 0.03 m, providing uniform resolution in the core flow region while enabling efficient computation. To accurately resolve the steep thermal gradients around the circular heat source and along the container walls, an inflation layer comprising 12 prism layers was applied on all solid-fluid interfaces. A transition ratio of 0.272 and a growth rate of 1.2 were used to progressively expand the layer thickness away from the walls. This combination ensures a smooth transition from the near-wall fine mesh to the coarser bulk mesh, thereby improving stability and reducing discretization error. The inflation region captures the near-wall heat transfer phenomena with high fidelity, which is critical for predicting heat spreading behavior.

Table 4: Mesh quality check through different contours

Sr. No.	Description	Contours
1.	Orthogonal quality	

2.	Aspect ratio	Aspect Ratio  2.83e+01 2.56e+01 2.29e+01 2.03e+01 1.76e+01 1.49e+01 1.22e+01 9.49e+00 6.80e+00 4.11e+00 1.42e+00 contour-1
3.	Mesh poor elements	Mark Poor Elements  0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 0.00e+00 contour-1
4.	Cell-Equiangular skewness	Cell Equiangle Skew  5.05e-01 4.54e-01 4.04e-01 3.53e-01 3.03e-01 2.52e-01 2.02e-01 1.51e-01 1.01e-01 5.05e-02 6.02e-05 contour-1

The mesh quality check was performed to ensure the correctness of the results and details have mentioned in the Table. The orthogonal quality contour indicates that the majority of the computational cells possess values close to unity, with only minor local reductions near the curved heat-source boundary where geometric refinement is necessary. Since orthogonal quality values greater than 0.2 are considered acceptable for transient natural convection simulations, the obtained distribution confirms excellent cell alignment and minimal numerical diffusion. This ensures stable pressure-velocity coupling and accurate resolution of buoyancy-driven flow structures.

The cell equiangle skewness contour shows that most cells lie within the low-skewness range, with values well below the critical threshold of 0.85 recommended by ANSYS FLUENT. Slightly elevated skewness is observed only in highly refined regions around the circular heat source, which is expected due to curvature-induced grid transitions. These localized values remain within permissible limits and do not adversely affect solver convergence or solution accuracy.

The “Mark Poor Elements” contour confirms the absence of critically poor-quality cells across the computational domain. No cells violate ANSYS FLUENT’s mesh-quality constraints, indicating that the grid does not contain inverted, highly skewed, or degenerate elements. This validates the robustness of the mesh for transient thermal and flow simulations.

The aspect ratio contour demonstrates that the majority of cells exhibit low and uniform aspect ratios, remaining within the acceptable range recommended for natural convection problems. Slightly higher aspect ratios near boundary layers are intentional and beneficial for resolving thermal gradients adjacent to walls and the heat source without compromising numerical stability.

Overall, all mesh-quality parameters—orthogonal quality, skewness, aspect ratio, and poor element indicators—remain well within the ranges specified by ANSYS FLUENT best-practice guidelines for transient buoyancy-driven heat transfer simulations. The selected mesh therefore ensures numerical stability, accurate plume resolution, reliable temperature prediction, and solver convergence, making it appropriate and correct for the present confined fire heat-propagation study.

2.3 Governing Equations:

Assumptions used in Two dimensional (2-D) Simulation

- Two-dimensional, transient (unsteady) flow was considered in the x- and y-directions.

- Air was modeled as a **variable-density Newtonian fluid** using the ideal gas law.
- The flow was assumed to be **low-Mach-number turbulent flow** and modeled using the **k- ω SST turbulence model**.
- Gravity acted in the negative y-direction with $g = -9.81 \text{ m/s}^2$
- No-slip boundary conditions were applied at the heat source and container walls.
- Buoyancy effects were captured through **temperature-dependent density variations** rather than the Boussinesq approximation.

Continuity Equation (2-D)

For incompressible flow:

Continuity equation has been presented in equation (1),

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \text{ ----- (1)}$$

Where,

ρ = density

u = velocity in x-direction

v = velocity in y-direction

Momentum Equations (2-D RANS):

'X' and 'Y' directional momentum equations have been presented in equations (2) and (3) respectively.

X-Momentum Equation:

$$\left(\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u u)}{\partial x} + \frac{\partial(\rho v v)}{\partial y} \right) = - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}[(\mu + \mu_t) \frac{\partial u}{\partial x}] + \frac{\partial}{\partial y}[(\mu + \mu_t) \frac{\partial u}{\partial y}] \text{ -----(2)}$$

Y-Momentum Equation:

$$\left(\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho u v)}{\partial x} + \frac{\partial(\rho v v)}{\partial y} \right) = - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x}[(\mu + \mu_t) \frac{\partial v}{\partial x}] + \frac{\partial}{\partial y}[(\mu + \mu_t) \frac{\partial v}{\partial y}] + \rho g_y \text{ -----}$$

(3)

Where,

μ = is molecular dynamic viscosity,

$\mu_t = \rho \cdot \nu_t$ is turbulent (eddy) viscosity,

$P = \rho RT$ (ideal gas law)

P = Pressure,
 R = gas constant
 T = temperature

Energy Equation (2-D):

The energy equations have been presented in equations (4) and (5)

$$\left(\frac{\partial(\rho C_p T)}{\partial t} + \frac{\partial(\rho u C_p T)}{\partial x} + \frac{\partial(\rho v C_p T)}{\partial y} \right) = \frac{\partial}{\partial x} \left[(k + K_t) \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[(k + K_t) \frac{\partial T}{\partial y} \right] \text{----- (4)}$$

with turbulent thermal conductivity:

$$K_t = \frac{\mu_t C_p}{Pr_t} \text{----- (5)}$$

Turbulence Model:

k- ω SST (2-D Form):

The turbulent kinetic energy equations have been presented in equations (6) and (7)

Turbulent Kinetic Energy (k) Equation:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u k)}{\partial x} + \frac{\partial(\rho v k)}{\partial y} = P_k - \beta^* \rho k \omega + \frac{\partial}{\partial x} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x} \right] + \frac{\partial}{\partial y} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial y} \right] \text{--- (6)}$$

Specific Dissipation Rate (ω) Equation:

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u \omega)}{\partial x} + \frac{\partial(\rho v \omega)}{\partial y} = \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + \frac{\partial}{\partial x} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x} \right] + \frac{\partial}{\partial y} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial y} \right] + D_\omega \text{--(7)}$$

Where, D_ω is the **cross-diffusion term** used in SST formulation.

Turbulent Viscosity Closure:

The equation for turbulent viscosity closure has been mentioned in equation (8)

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \omega, S F_2)} \text{----- (8)}$$

Buoyancy force in y-momentum equation:

The equation for the buoyancy force used in the simulation has been presented in equation (9)

$$\rho g_y = \left(\frac{P}{RT} \right) g \text{----- (9)}$$

Table 6: Fluid and Physical Constants (Air) at 27 deg C

Parameter	Symbol	Value / model used	Unit
Density	ρ	Temperature dependent, (P/RT)	$\text{kg}\cdot\text{m}^{-3}$
Dynamic viscosity	μ	1.7894×10^{-5}	$\text{kg}\cdot\text{m}^{-1}\cdot\text{s}^{-1}$
Specific heat	cp	1006.43	$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$
Thermal conductivity	k	0.0242	$\text{W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$
Gravity (Y-direction)	gy	-9.81	$\text{m}\cdot\text{s}^{-2}$
Gas constant	R	287	$\text{J}\cdot\text{kg}^{-1}\cdot\text{K}^{-1}$

Turbulence Model Constants

k- ω SST Model (Default Fluent Values):

Table 7: Model Constants

Constant	Symbol	Value
Turbulent kinetic energy production coefficient	α	0.31
Dissipation coefficient (k-equation)	β^*	0.09
Dissipation coefficient (ω -equation)	β	0.075
Eddy viscosity limiter constant	α_1	0.31

Table 8: Diffusion Coefficients

Quantity	Symbol	Value
Turbulent Prandtl number for 'k'	σ_k	0.85
Turbulent Prandtl number for ' ω '	σ_ω	0.50

Table 9: Blending Function Constants (SST):

Constant	Value
F1	Near-wall blending (k- ω)
F2	Free-stream blending (k- ε behavior)

These values were calculated automatically by the software based on the wall distance

Table 10: Turbulent Viscosity and Heat Transfer Parameters:

Parameter	Symbol	Value
Turbulent viscosity	μ_t	Computed by SST model
Turbulent Prandtl number	Pr_t	0.85
Turbulent thermal conductivity	k_t	$Kt = \frac{\mu_t C_p}{Pr_t}$

Pressure-Velocity Coupling (SIMPLE Scheme)

Table 11: Under-Relaxation Factors (Transient Defaults):

Variable	Relaxation Factor
Pressure	0.3
Density	1.0
Body forces	1.0
Momentum	0.7
Energy	1.0
Turbulent kinetic energy (k)	0.8
Specific dissipation rate (ω)	0.8

Table 12: Temporal Discretization Parameters:

Parameter	Value
Time formulation	Implicit
Time step size	User-defined (e.g. 0.002 s)
Courant number limit	< 2
Iterations per time step	10

Table 13: Convergence Criteria (Residual Limits)

Equation	Residual Target
Continuity	10^{-5}

X-velocity	10^{-5}
Y-velocity	10^{-5}
Energy	10^{-6}
Turbulent kinetic energy (k)	10^{-5}
Specific dissipation rate (ω)	10^{-5}

2.4 Theoretical basis for selection of the specific flow model

The heat spreading inside the container is driven by natural convection. The buoyancy-induced motion of air transports thermal energy from the source to the surrounding fluid. The buoyancy force responsible for natural convection. The Grashof number characterizes the relative strength of buoyancy forces compared to viscous forces, whose equation (10) has mentioned below:

$$Gr = \frac{g \beta (T_s - T_\infty) L^3}{\nu^2} \text{-----(10)}$$

A high value of Gr ($>10^9$) indicates strong buoyancy-driven convection and the onset of turbulence in natural convection.

The Rayleigh number governs the nature of convective flow, presented has been mentioned in equation (11).

$$Ra = Gr \times Pr \text{-----(11)}$$

For all studied temperatures $Ra > 10^9$, confirming the turbulent and highly fluctuating nature of the thermal plume, with well-mixed hot fluid rising and cooler fluid recirculating.

A two-dimensional simulation is considered appropriate due to the geometric symmetry of the container and the corner-mounted positions of the heat sources. The dominant thermal stratification and plume development occur primarily along the vertical planes extending from each heat source, allowing a 2-D model to effectively capture the essential flow structures while significantly reducing computational effort. This simplification is widely accepted in cases where lateral heat losses are negligible compared to the vertical plume motion. To assess the flow regime (laminar or turbulent) within the container, preliminary calculations were performed based on the characteristic flow parameters of the present simulation.

$$\text{Ambient/air temperature} = T_\infty = 27 + 272 = 300 \text{ K}$$

$$\text{Surface / source temperature} = T_s = 200 + 450 = 723 \text{ K}$$

$$\text{The coefficient of thermal expansion} = \beta = \frac{1}{T_\infty} = \frac{1}{300} = 0.0032$$

$$\text{Characteristic length} = L = 0.5 \text{ m}$$

$$\text{Kinematic viscosity} = \nu = 1.57 \times \frac{10^{-5} \text{ m}^2}{\text{s}}$$

$$\text{Grashof Number} = \text{Gr} = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2} = \frac{9.81 \times 0.0032 \times (723 - 300) \times 0.5^3}{(1.57 \times 10^{-5})^2} = 0.6944 \times 10^{10}$$

$$\text{Prandtl Number} = \text{Pr} = 0.7 \text{ (for the given temperature conditions)}$$

$$\text{Rayleigh Number} = \text{Ra} = \text{Gr} \times \text{Pr} = (0.6944 \times 10^{10}) \times 0.7 = 4.8608 \times 10^9$$

As the observed value of the Rayleigh number (Ra) was found to be greater than 10^9 , the flow is turbulent, and highly fluctuating, well-mixed thermal plumes will be produced.

2.5 Theoretical Basis for First-Layer Height and Time-Step Calculation

Accurate prediction of near-wall heat transfer in buoyancy-driven or mixed convection flows requires resolving the viscous sublayer with adequate mesh refinement. The first-layer thickness and total inflation-layer height were therefore calculated based on the non-dimensional wall distance, y^+ , which ensures proper integration of the turbulence model and enhanced wall treatment. The time step calculation is mentioned in equation (12).

$$\text{Time Step Calculation (CFL-Based): } \Delta t = C \times \frac{\Delta x}{u} \text{ -----(12)}$$

Mesh Settings: $y^+ = 1$ (near-wall mesh requirement),

Growth rate = 1.2,

Boundary Layer & First Cell Height Calculation:

Total Prism Layer Thickness: Total thickness $\approx 10 \times$ first cell height,

Smooth prism layers required.

First cell height calculation has mentioned in equation (13).

$$\text{Wall Distance (First Cell Height) Formula} = y = \frac{y^+ \nu}{u_\tau} \text{ -----(13)}$$

Friction velocity calculation has mentioned in equation (14).

$$\text{Friction Velocity} = u_\tau = U \sqrt{\frac{C_f}{2}} \text{ -----(14)}$$

Free stream velocity =

Pseudo Courant number =

Target wall distance =

$$\text{Kinematic viscosity} = \nu = 1.5 \times 10^{-5} \text{ m}^2/\text{s}$$

Characteristic length =

Reynolds Number =

Skin Friction Coefficient =

Final First Layer Height =

Friction Velocity =

First Layer Height (for $y^+ = 1$) =

$$\text{Time Step from CFL Condition} = \Delta t = C \times \frac{\Delta x}{U} = 2 \times \frac{0.00052708}{0.5} = 0.0021082s$$

The mesh size and time-step calculations were performed to ensure numerical stability and accurate resolution of near-wall thermal and velocity gradients in the natural convection simulations. The first-layer height was determined using the non-dimensional wall distance requirement ($y^+ = 1$) to properly resolve the viscous sublayer under enhanced wall treatment. Based on the characteristic length $L = 0.5$ m, free-stream velocity $U = 0.5$ m/s, and air kinematic viscosity $\nu = 1.5 \times 10^{-5}$ m²/s, the Reynolds number was computed as $Re \approx 1.67 \times 10^4$. The corresponding skin-friction coefficient ($C_f = 0.006487$) was obtained using the empirical relation $C_f = 0.026Re^{-1/7}$. From this, the friction velocity was calculated as $u_\tau = 0.02846$ m/s, giving a first-layer thickness of $y \approx 5.27 \times 10^{-4}$ m. This ensures that the near-wall mesh meets the turbulence-model requirements and captures the sharp thermal gradients occurring around the heat source and container walls.

The time-step size was derived using the CFL (Courant-Friedrichs-Lewy) condition to maintain numerical stability for transient flow simulation. Using a pseudo Courant Number, $C = 2$, the first-cell height $\Delta x \approx 5.27 \times 10^{-4}$ m, and free-stream velocity $U = 0.5$ m/s, the time step was found to be $\Delta t = 0.002108$ s. This time-step value allows the solver to capture the evolution of the thermal plume without violating stability constraints, while ensuring smooth convergence throughout the transient simulation.

2.6 Solver Settings

Table 13 : Numerical set details

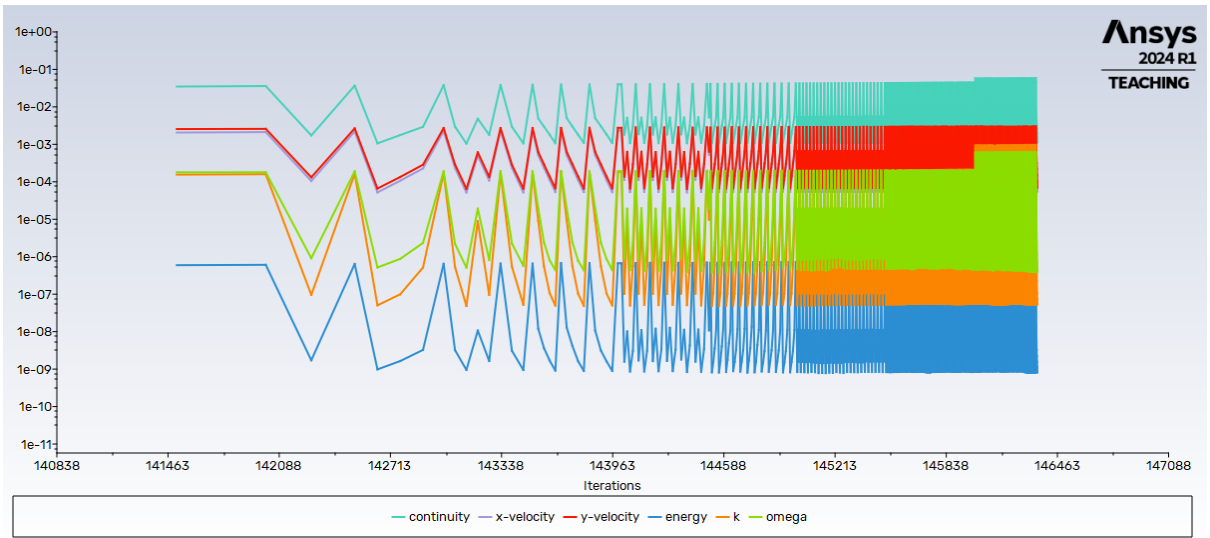
Numerical Parameter	Value/Model Used
Solver	Pressure-Based Transient
Gravity	-9.81 m/s ² , Vertical, (along the Y -axis in this model)
Models	Energy Equation: ON

	Turbulence : K- ω SST
Cell zone conditions / Material properties	Fluid: Air, incompressible Solid : Corten steel
Boundary conditions	Heat source: no slip, wall with temperature 450 deg C Container walls: adiabatic, corten steel Air temperature = 27 deg C
Solution method (Pressure-Velocity Coupling)	Semi-implicit pressure linked equation (SIMPLE for transient)
Spatial Discretization	Least square cell based Pressure: Second Order Upwind Momentum: Second Order Upwind Energy: Second-Order Upwind Turbulence (k, ω): Second-Order Upwind
Residuals	Continuity = $1e^{-5}$, x- velocity = $1e^{-5}$ y- velocity = $1e^{-5}$ Energy = $1e^{-6}$ K = $1e^{-5}$ Epsilon = $1e^{-5}$
Iterations per time step	10
Total number of time steps	15000
Time Step (Δt)	0.002 s (ensuring Courant number < 2)
Total Simulation Time	30 sec

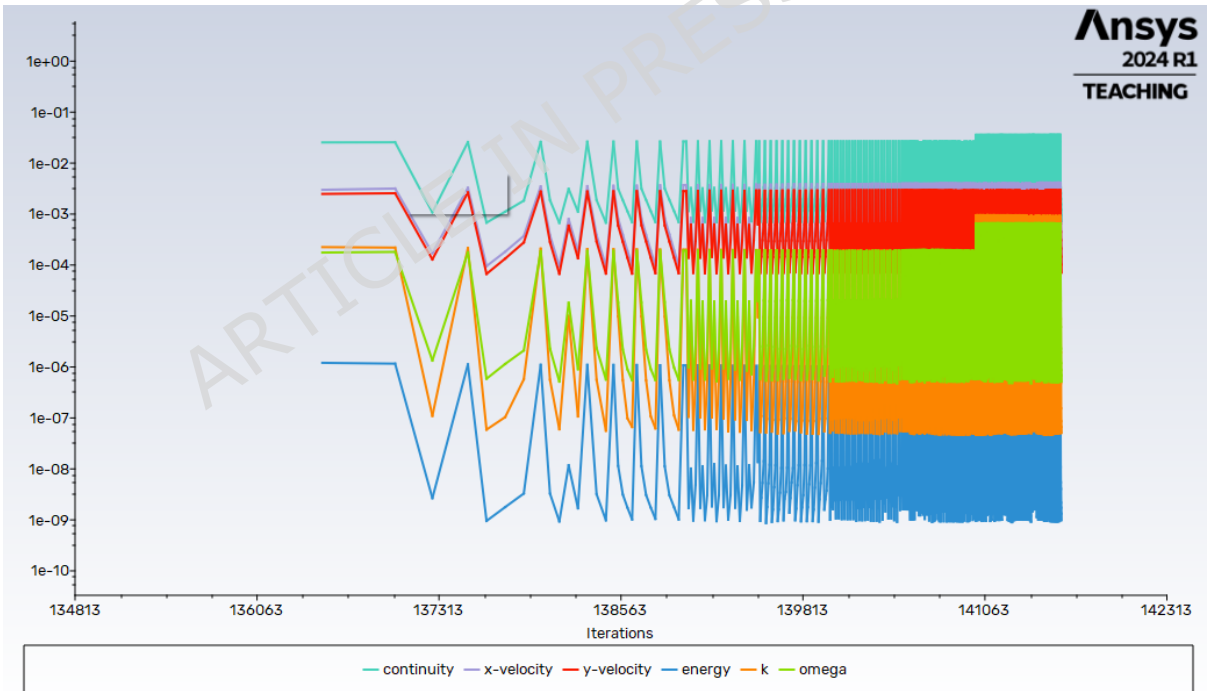
2.7 Monitoring Parameters

The convergence criteria for the present simulation were set using residuals to ensure solution accuracy. The residuals for continuity, x-velocity, y-velocity, turbulent kinetic energy (k), and turbulent dissipation rate (ϵ) were set to 1×10^{-5} , while the energy residual was set to 1×10^{-6} . The simulation was performed for a total physical time of 30 sec, with a time step size of 0.002 sec. Each time step was solved using 10 iterations, which ensured that all residuals were reduced below the specified thresholds, indicating a stable

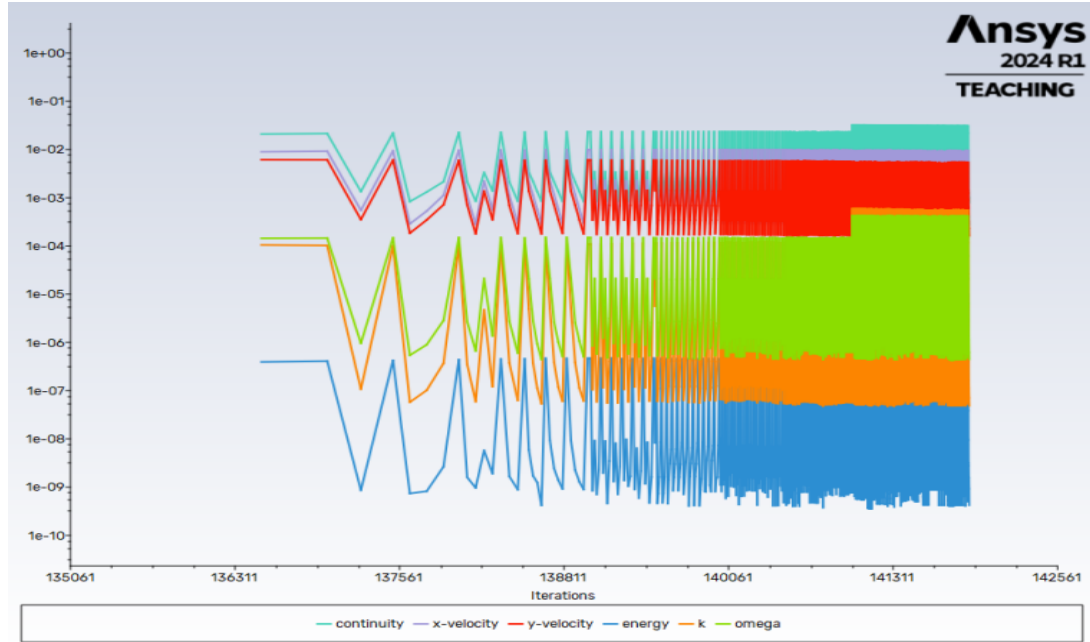
and converged solution. The residual graphs for the different orientation of the heat source has mentioned in the Figure 5.



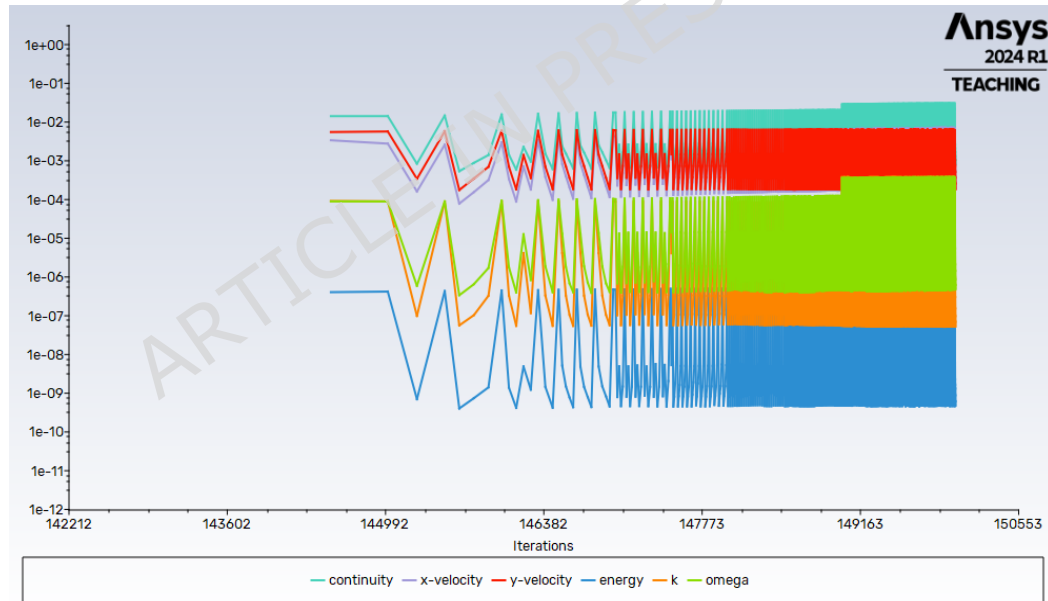
(a) Residual graph for case 1 , D1 ($x=0.65\text{m}$, $y=0.65\text{m}$)



(b) Residual graph for case 2 , D2 ($x=1.95\text{m}$, $y=0.65\text{m}$)



(c) Residual graph for case 3, D3 ($x=1.95\text{m}$, $y=1.95\text{m}$)



(d) Residual graph for case 4, D4 ($x=0.65\text{m}$, $y=1.95\text{m}$)

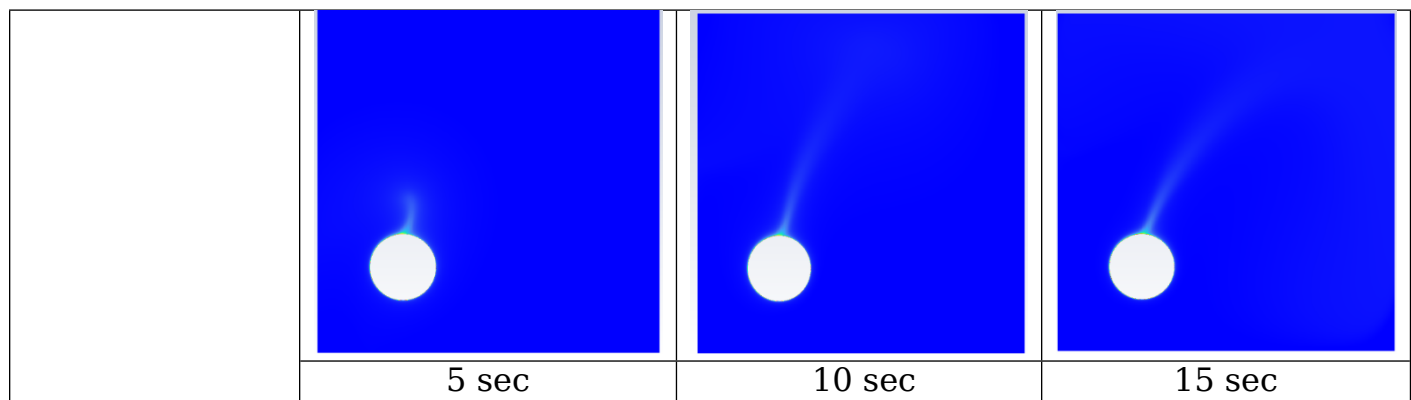
Figure 5: Residual graphs for different cases

3. Results and Discussion

3.1 Case-wise Heat Propagation Analysis

3.1.1 Case-1 Heat source placed location D1 - (0.65 m, 0.65m):

The transient temperature contours from the two-dimensional simulation illustrate the evolution of a buoyancy-driven thermal plume generated by a corner-mounted circular heat source (D_1 : 0.65 m, 0.65 m) maintained at 450 °C within a 2.6 m $\times$ 2.6 m enclosure. Within the first few seconds, temperature rise is highly localized, and a narrow vertical plume forms directly above the heat source, indicating natural convection as the dominant heat transfer mechanism. Between 10–15 s, the plume elongates and thickens due to entrainment of surrounding cooler air, with initial lateral heat spreading becoming evident. By 20 s, interactions with enclosure boundaries induce recirculation and thermal stratification, resulting in wider, non-uniform temperature distribution. At later times (25–30 s), significant heat accumulation occurs, and temperature gradients reduce as the system approaches transient thermal equilibrium. These results highlight the strong influence of confinement on heat propagation and emphasize, from a marine and industrial fire-safety perspective, the critical importance of early thermal detection above potential fire sources and timely activation of water-mist or Hyper Mist systems, as delayed suppression allows widespread lateral and vertical heat spread, reducing control effectiveness. Optimized placement of thermal sensors and mist nozzles along expected plume paths and upper regions is therefore essential for rapid fire response. Key findings include: early formation of a buoyancy-driven plume confirming vertical heat transport dominance, progressive plume elongation and thickening due to air entrainment, increasing lateral spreading with plume-boundary interactions, significant confinement effects after 20 s, development of recirculation and thermal stratification, substantial heat accumulation at later times, and reduced temperature gradients signaling approach to thermal equilibrium. The study provides practical guidance for sensor and suppression system design, demonstrating that fire source location governs heat propagation and that timely, strategically positioned mitigation measures substantially improve fire control efficiency in confined marine and industrial environments.



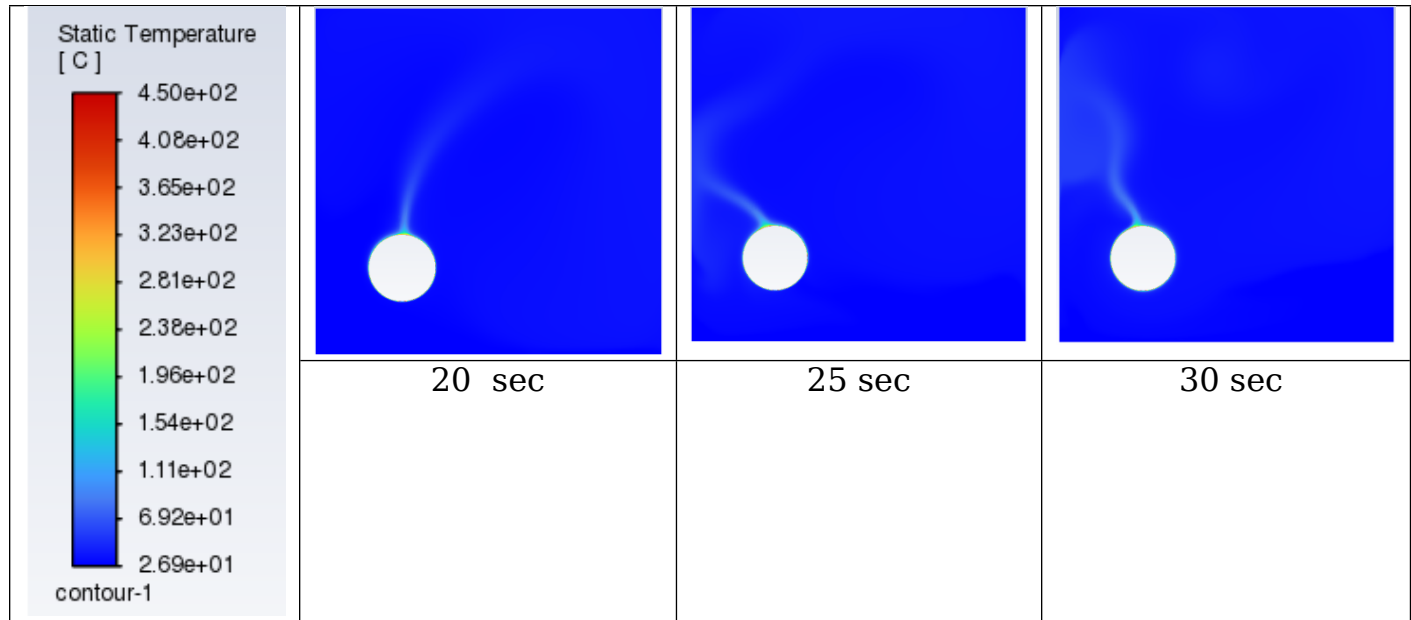


Figure: 6 Temperature contours for D1 - (0.65 m, 0.65m)

3.1.2 Case-2 Heat source placed location D2 - (1.95 m, 0.65m)

The transient temperature contours illustrate the evolution of heat propagation from a corner-mounted circular heat source maintained at 450 °C within a 2.6 m × 2.6 m enclosure. Within the first 5 s, a narrow vertical thermal plume forms directly above the source, indicating the onset of buoyancy-driven natural convection, while the remainder of the enclosure remains near ambient temperature. By 10–15 s, the plume elongates and begins to curve due to interactions with enclosure boundaries, reflecting asymmetric flow development induced by confinement. Around 20 s, hot gases spread laterally along the upper regions, forming recirculation zones that redistribute heat away from the source. At 25–30 s, large-scale circulation leads to widespread temperature elevation, with reduced local gradients near the source indicating dominance of convective transport over conduction and increasing influence of enclosure effects on overall thermal behavior. These results confirm that buoyancy-induced flow governs early heat transport, while confinement shapes later-stage temperature distribution. From a fire-safety perspective, the study underscores that delayed intervention allows extensive heat spread and reduces suppression effectiveness. Early detection directly above the source, coupled with prompt activation of mist-based fire suppression systems, is therefore critical. Optimal placement of thermal sensors and Hyper Mist nozzles along anticipated plume paths and in upper regions of the enclosure ensures rapid, effective fire control in confined marine and industrial spaces.

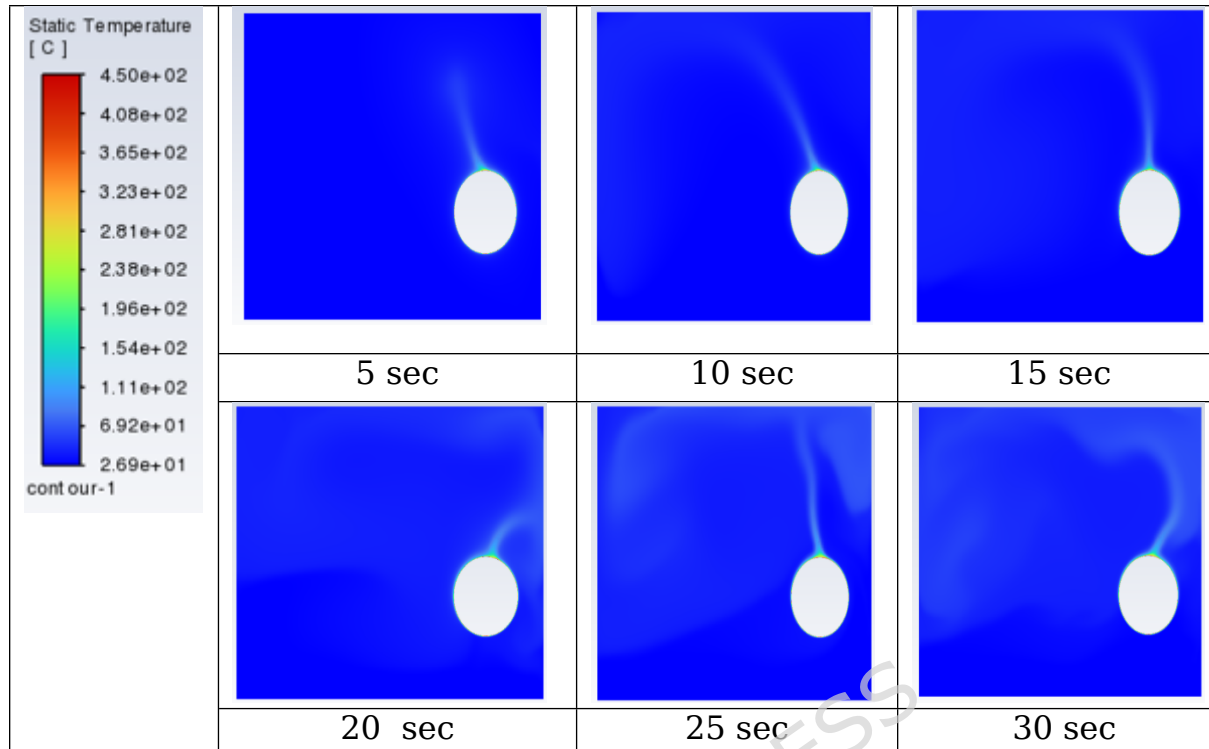


Figure: 7 Temperature contours for D2 - (1.95 m, 0.65m)

3.1.3 Case-3 Heat source placed location D3 - (1.95m, 1.95m)

The transient temperature contours illustrate the heat transfer behavior when the circular heat source is positioned near the upper region of the 2.6 m × 2.6 m enclosure, showing altered plume dynamics due to proximity to the ceiling. Within 5 s, the thermal plume forms above the source but quickly impinges on the top boundary, limiting vertical rise and initiating early lateral spreading along the ceiling. By 10–15 s, hot gases travel horizontally beneath the ceiling, creating a ceiling-hugging thermal layer rather than a freely rising plume, indicating that buoyancy-driven flow is rapidly constrained by enclosure geometry. Around 20 s, recirculation zones develop as heated air moves laterally and descends alongside walls, promoting internal mixing. At 25–30 s, the upper enclosure exhibits sustained elevated temperatures while the lower region remains cooler, demonstrating rapid thermal stratification. These results highlight that ceiling proximity accelerates enclosure heating, suppresses vertical plume development, and alters heat propagation patterns. From a fire-safety perspective, early detection is most effective with ceiling-mounted thermal sensors, and mist or Hyper Mist nozzles positioned solely at mid-height may underperform initially. Effective suppression design should therefore include ceiling-level discharge to intercept hot gas layers, emphasizing that fire source elevation critically influences thermal behavior and mitigation strategies in confined marine and industrial spaces.

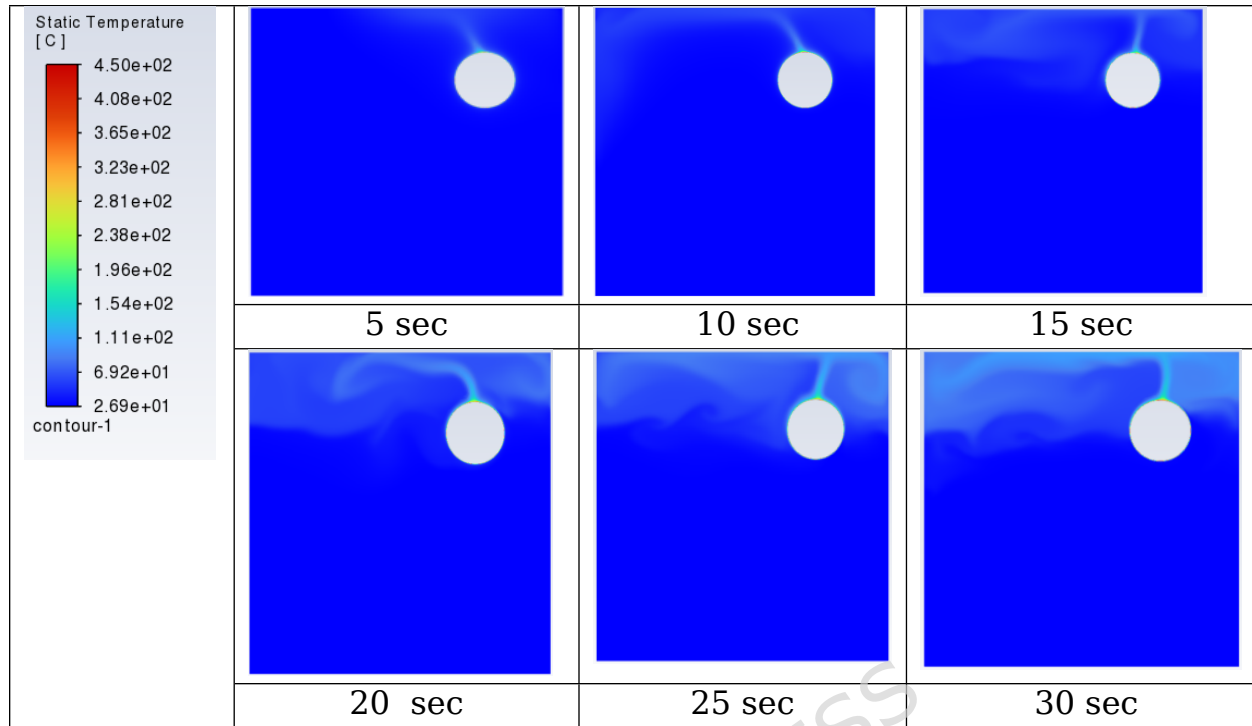


Figure: 8 Temperature contours for D3 - (1.95m, 1.95m)

3.1.4 Case-4 Heat source placed location D4 - (0.65m , 1.95 m):

The transient temperature contours illustrate the thermal response of the enclosure when the circular heat source is positioned near the upper corner. Within 5 s, a short, narrow thermal plume forms but immediately interacts with nearby boundaries, restricting vertical growth. By 10–15 s, the plume spreads laterally along the ceiling and adjacent wall, forming an inclined hot gas layer that propagates away from the source, highlighting the strong influence of geometric confinement on buoyancy-driven flow. Around 20 s, complex recirculation patterns develop, with warm air accumulating in the upper corner while cooler air remains in the lower region. At 25–30 s, the hot layer thickens and extends along the ceiling, and temperature gradients reduce due to increased mixing. These results demonstrate that corner placement accelerates localized heat buildup, alters plume symmetry, and produces pronounced thermal stratification. From a fire-safety perspective, early detection requires corner-mounted thermal sensors, while standard centrally positioned mist nozzles may be ineffective; Hyper Mist systems should incorporate targeted corner and ceiling-level nozzle placement. Overall, the findings emphasize that the fire source location critically governs heat propagation, localized hot-spot formation, and optimal suppression strategies in confined enclosures.

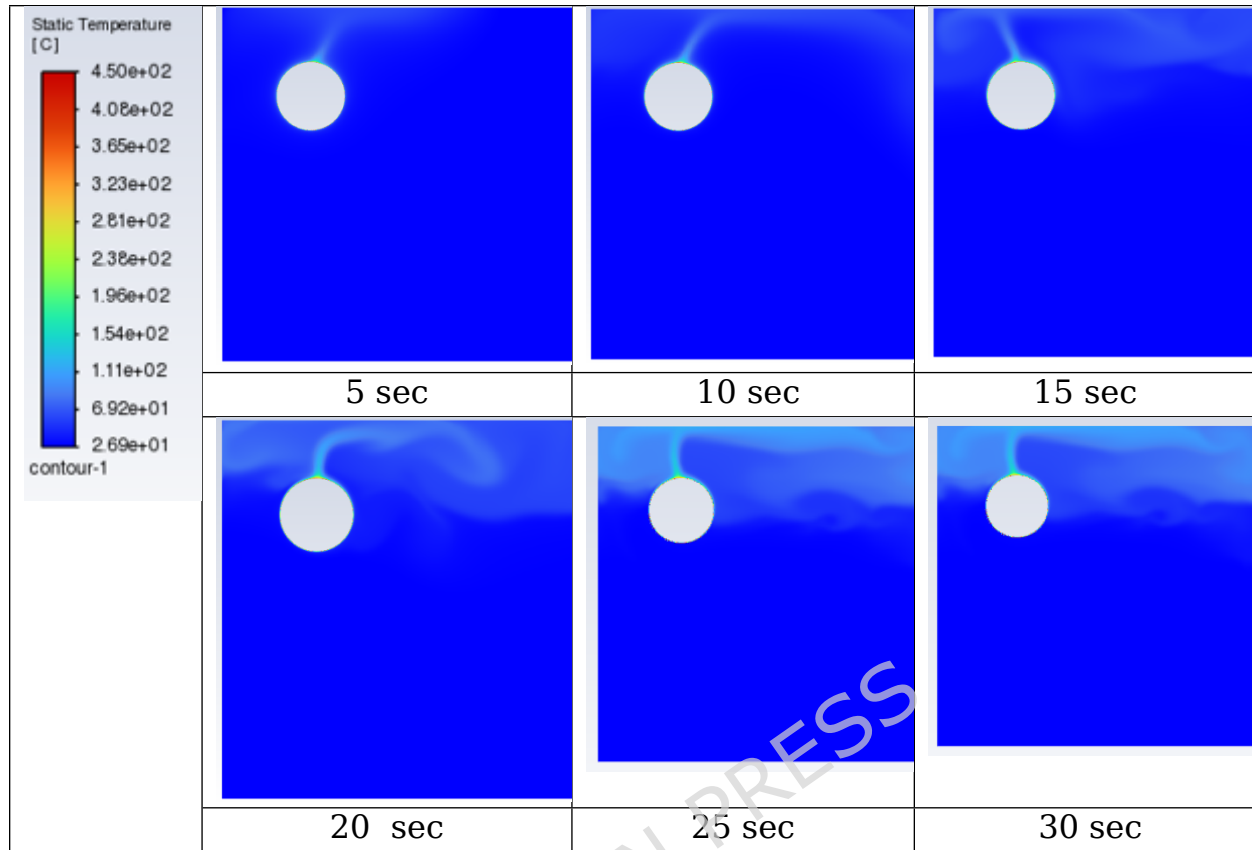


Figure: 9 Temperature contours for D4 - (0.65m , 1.95 m)

3.2 Pressure contour Analysis

Diagonal positions	Static pressure contours	Diagonal positions	Static pressure contours
D1 (x=0.65m, y=0.65m)		D3 (x=1.95m, y=1.95m)	

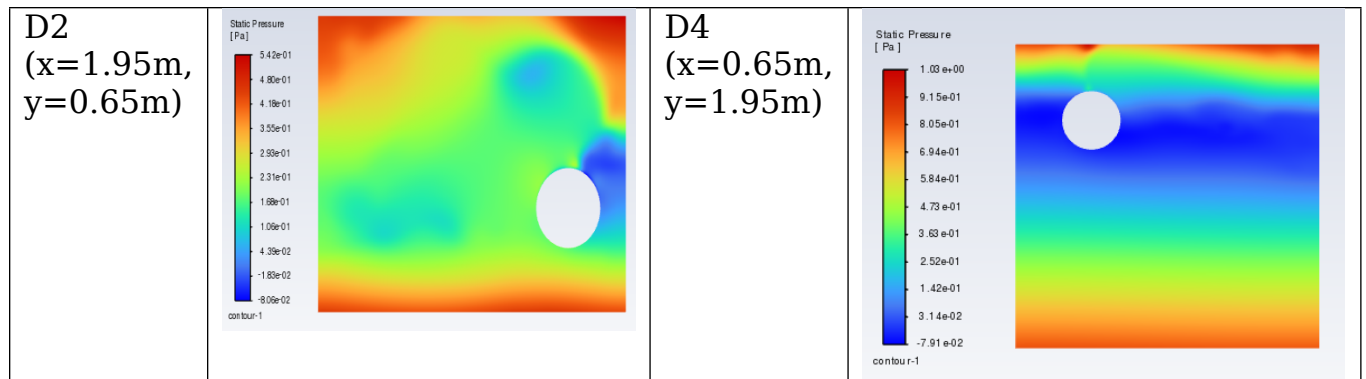
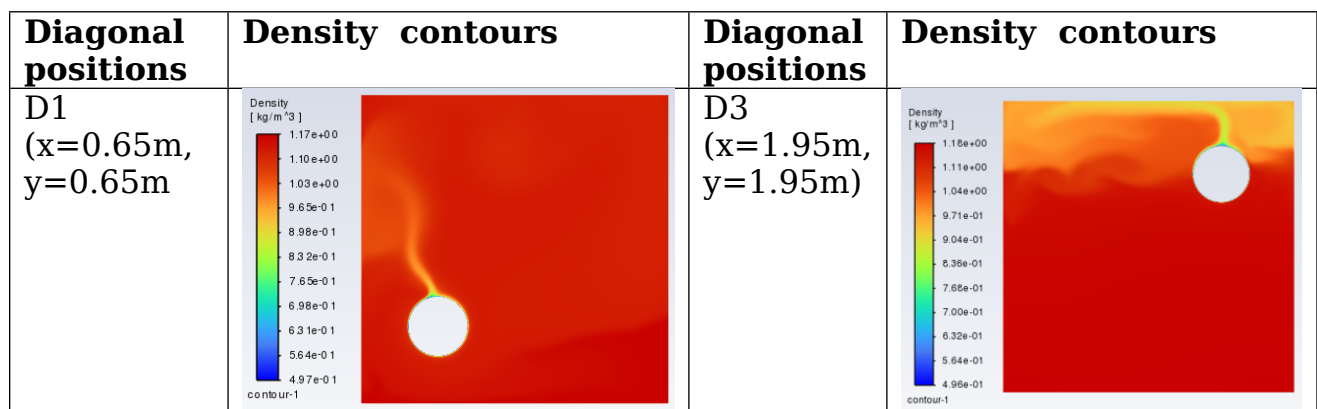


Figure: 9 Static pressure contours for different orientations of heat source in container @ 30 sec

The static pressure contours represent relative (gauge) pressure distributions obtained from the CFD solver and are used to indicate local buoyancy-driven flow patterns within the enclosure. For D1 and D2, localized low-pressure regions develop near the heat source due to upward acceleration of heated air and associated plume formation, while D3 and D4 exhibit more stratified pressure distributions because of ceiling interaction and confinement effects. The observed pressure variations, typically of the order of 10^{-1} to 10^0 Pa, primarily reflect relative flow redistribution and buoyancy-induced circulation, rather than physically significant pressure changes. Therefore, the pressure contours are interpreted qualitatively to understand plume evolution, recirculation behavior, and enclosure effects, and should not be considered direct indicators of fire severity, pressure hazards, or suppression performance. These results provide complementary information regarding flow structure alongside temperature, velocity, and density distributions.

3.3 Density Contour Analysis



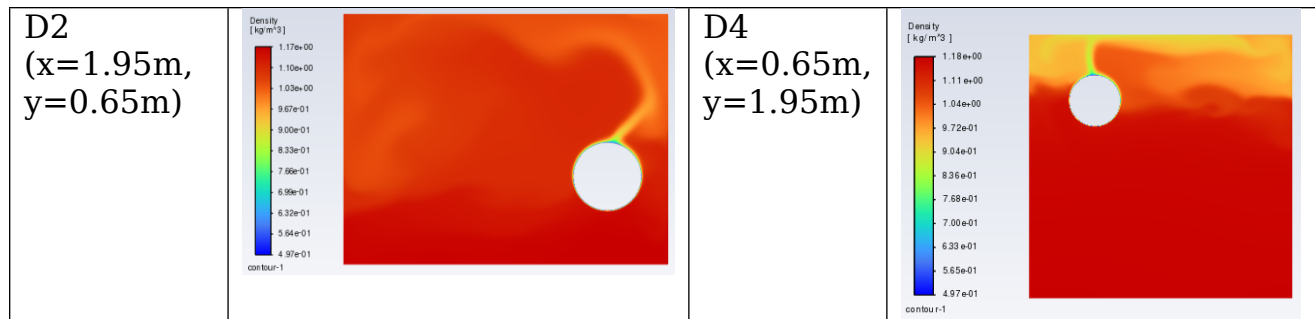


Figure: 10 Density contours for different orientations of heat source in container @ 30 sec

The density contours illustrate the influence of heat source location on buoyancy-driven density variations within the 2.6 m × 2.6 m enclosure, with elevated temperatures corresponding to reduced air density that drives natural convection and plume formation. Across all configurations, densities range from approximately 0.49 kg·m⁻³ in the hottest plume regions to 1.17 kg·m⁻³ in cooler ambient zones. For D1, a strong vertical low-density plume forms above the source (0.49–0.55 kg·m⁻³), with surrounding air near 1.10–1.17 kg·m⁻³, promoting efficient buoyant transport with minimal early wall interaction. In D2, wall proximity deflects the rising plume, producing an asymmetric low-density region (0.55–0.65 kg·m⁻³) and enhanced lateral mixing. Upper-corner sources D3 and D4 experience early plume impingement on the ceiling, forming stratified low-density layers (0.60–0.72 kg·m⁻³) near the top while lower regions remain denser (1.10–1.17 kg·m⁻³), with D4 exhibiting slightly smoother gradients due to enhanced circulation. These results highlight that lower-source placements generate strong vertical plumes, whereas upper-source positions induce early stratification and reduced plume penetration. Density gradients are steepest in unrestricted plumes, while proximity to walls and ceiling significantly alters distribution and trajectory. Low-density regions correspond to preferred hot-gas paths and rapid buoyant acceleration, influencing smoke transport, early fire detection, and mist penetration. Overall, the study confirms that heat source orientation critically governs density stratification, plume behavior, and effective suppression strategy design in confined marine and industrial enclosures.

3.4 Velocity contour analysis

Diagonal positions	Velocity contours	Diagonal positions	Velocity contours
--------------------	-------------------	--------------------	-------------------

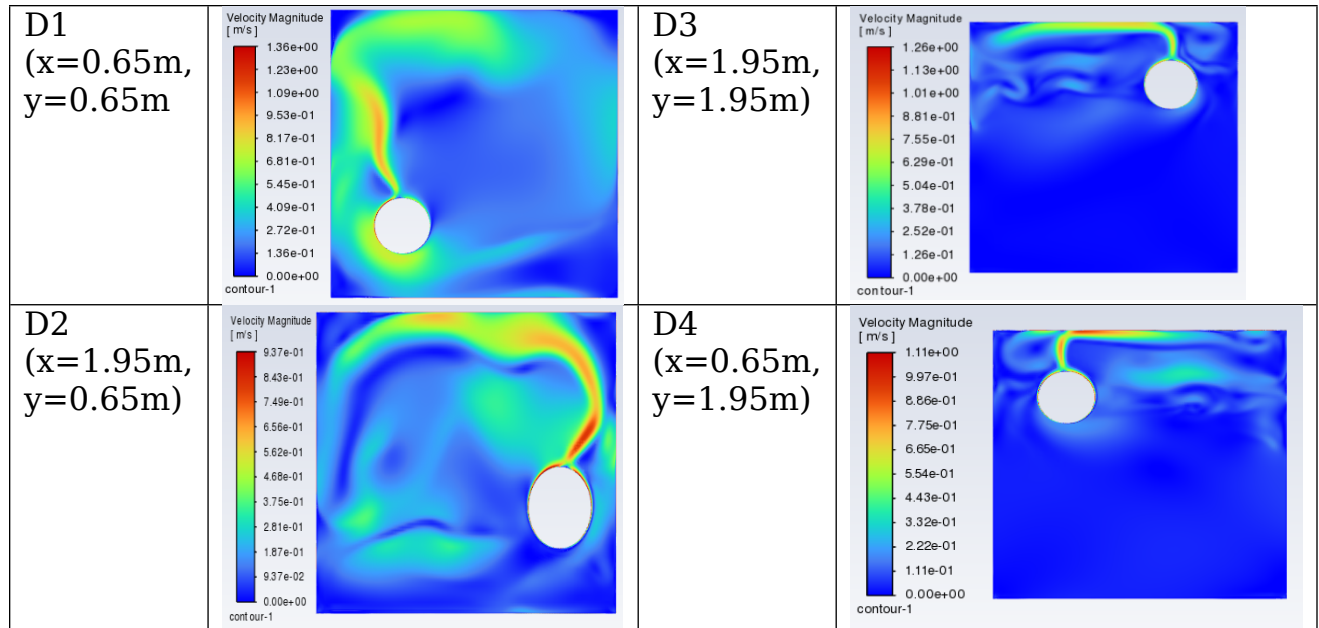


Figure: 11 Velocity contours for different orientations of heat source in container @ 30 sec

The velocity magnitude contours demonstrate the development of buoyancy-driven flow and reveal the strong dependence of flow structure on heat source location within the $2.6 \text{ m} \times 2.6 \text{ m}$ enclosure. Across all configurations, velocities range from quiescent regions near $0 \text{ m}\cdot\text{s}^{-1}$ to peak values around $1.36 \text{ m}\cdot\text{s}^{-1}$ in regions of intense plume acceleration or ceiling-attached jets. For D1, a strong vertical plume forms above the source with maximum velocities of approximately $1.36 \text{ m}\cdot\text{s}^{-1}$ and secondary recirculation along enclosure boundaries, indicating efficient upward transport of hot gases. In D2, wall proximity deflects the plume, producing an asymmetric flow with peak velocities around $0.94 \text{ m}\cdot\text{s}^{-1}$ and extensive lateral recirculation. Upper-corner sources D3 and D4 generate ceiling-attached horizontal jets with maximum velocities of $1.26 \text{ m}\cdot\text{s}^{-1}$ and $1.11 \text{ m}\cdot\text{s}^{-1}$, respectively, while lower regions remain relatively stagnant, and large-scale recirculation develops along side walls. These results indicate that lower-source placements favor strong vertical plumes, whereas upper-source positions produce horizontal ceiling jets, with wall and corner proximity enhancing asymmetry and confinement effects. Velocity gradients correlate with temperature and density distributions, highlighting preferred paths for heat and smoke transport. From a fire-safety perspective, high-velocity regions influence mist droplet penetration and residence time, emphasizing the importance of aligning mist nozzle orientation with dominant flow paths to ensure effective suppression, particularly in corner and upper-source configurations.

3.5 Turbulent Kinetic Energy contour analysis

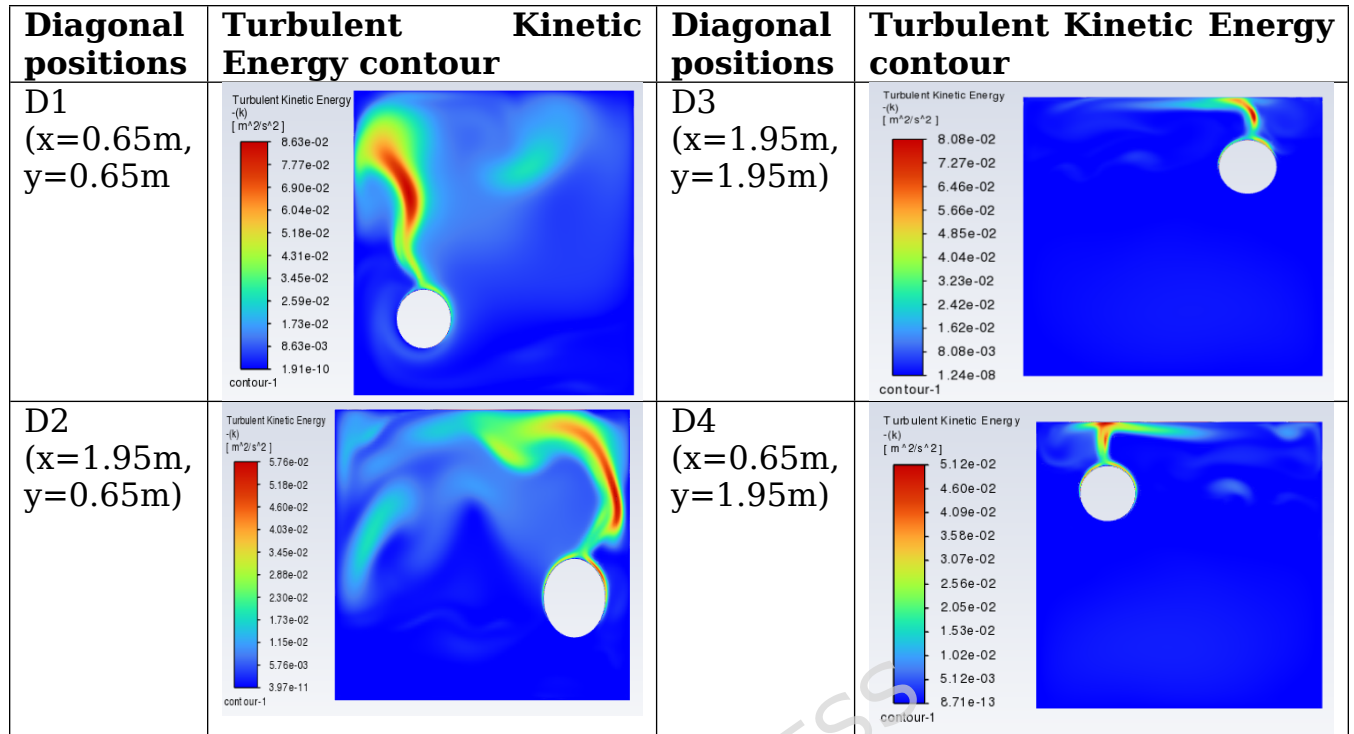


Figure: 12 Turbulent Kinetic Energy contour for different orientations of heat source in container @ 30 sec

The turbulent kinetic energy (TKE) contours reveal the intensity and spatial distribution of turbulence generated by buoyancy-driven flow within the 2.6 m × 2.6 m enclosure, with values ranging from approximately $10^{-12} \text{ m}^2 \cdot \text{s}^{-2}$ in quiescent regions to $8.5 \times 10^{-2} \text{ m}^2 \cdot \text{s}^{-2}$ in plume and shear-layer zones. For D1, an unobstructed vertical plume produces the highest localized TKE along the plume core and shear layers, while the surrounding enclosure remains nearly laminar. In D2, wall proximity and asymmetric recirculation generate moderately high TKE ($\approx 5.8 \times 10^{-2} \text{ m}^2 \cdot \text{s}^{-2}$) over a broader area, highlighting turbulence induced by plume-wall interaction. Upper-corner sources D3 and D4 produce turbulence confined mainly to ceiling-jet regions, with maximum TKE around $8.0 \times 10^{-2} \text{ m}^2 \cdot \text{s}^{-2}$ and $5.1 \times 10^{-2} \text{ m}^2 \cdot \text{s}^{-2}$, respectively, while the lower enclosure remains largely laminar. These results indicate that lower-source placements generate stronger and vertically extended turbulence, whereas upper-source positions restrict turbulence to horizontal ceiling jets. Wall and boundary proximity enhance TKE through increased velocity gradients, promoting mixing and accelerating heat and smoke dispersion. From a fire-safety perspective, high TKE regions may reduce water-mist penetration due to droplet breakup and entrainment, making accurate modeling of turbulence critical for predicting plume spread, stratification, and optimizing suppression system performance in confined marine and industrial enclosures.

3.6 Mass imbalance contour analysis

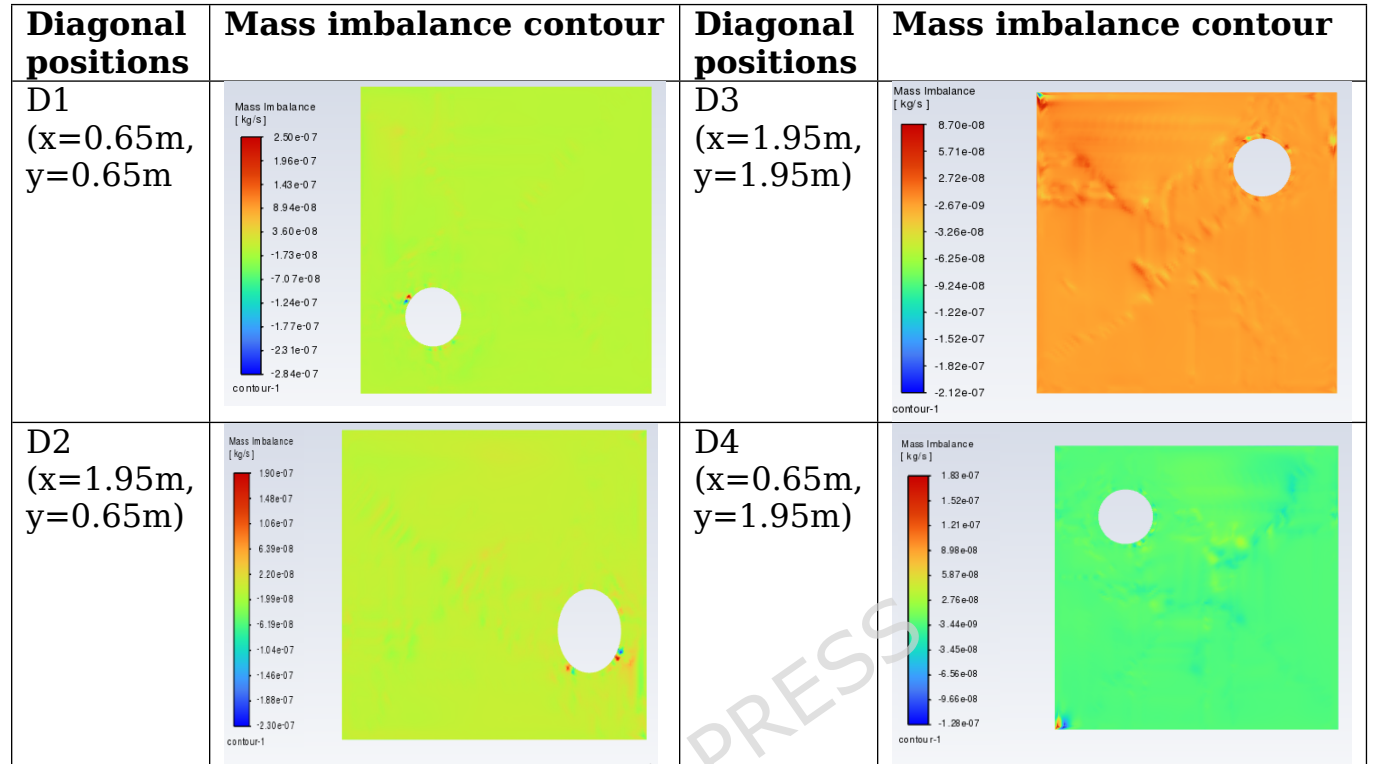


Figure: 13 Mass imbalance contour for different orientations of heat source in container @ 30 sec

The mass imbalance contours for configurations D1-D4 show that local continuity residuals remain low and spatially confined, with values approximately within -2.8×10^{-7} to $+8.8 \times 10^{-8} \text{ kg}\cdot\text{s}^{-1}$. Minor localized deviations near the heat source and boundary regions are attributed to buoyancy effects, plume-wall interactions, and steep thermal gradients. The observed mass imbalance indicates satisfactory numerical convergence and continuity conservation across all cases. However, these results are interpreted only as numerical diagnostics and do not independently validate the physical accuracy of plume behavior, temperature distribution, or turbulence predictions.

3.7 Velocity vector analysis

The velocity vector plots provide a detailed visualization of buoyancy-driven flow, plume development, and recirculation patterns inside the $2.6 \text{ m} \times 2.6 \text{ m}$ enclosure, highlighting both the magnitude and direction of hot gas movement, entrainment of ambient air, and interactions with enclosure boundaries. For D1, a strong vertical plume forms with peak velocities around 1.3 m/s , accompanied by counter-rotating recirculation cells that enhance entrainment and sustain upward plume growth. In D2, proximity to the wall deflects the plume, generating asymmetric flow with peak velocities near

0.95 m/s and dominant clockwise recirculation, promoting lateral heat and smoke transport. Upper-corner sources D3 and D4 exhibit ceiling-jet-dominated behavior, with peak velocities of 1.29 m/s and 1.12 m/s, respectively, limited vertical plume rise, and strong horizontal transport across the enclosure, while downward return flows create large-scale recirculation that sustains thermal stratification. These results show that lower-source placements produce taller vertical plumes, whereas upper-source placements induce horizontal ceiling jets with enhanced lateral heat and smoke spread. Velocity vectors identify preferred transport paths, inform early thermal detection, and guide optimal Hyper Mist nozzle orientation and droplet penetration, confirming the critical role of flow direction and recirculation patterns in fire growth, suppression efficiency, and sensor placement in confined marine and industrial compartments.

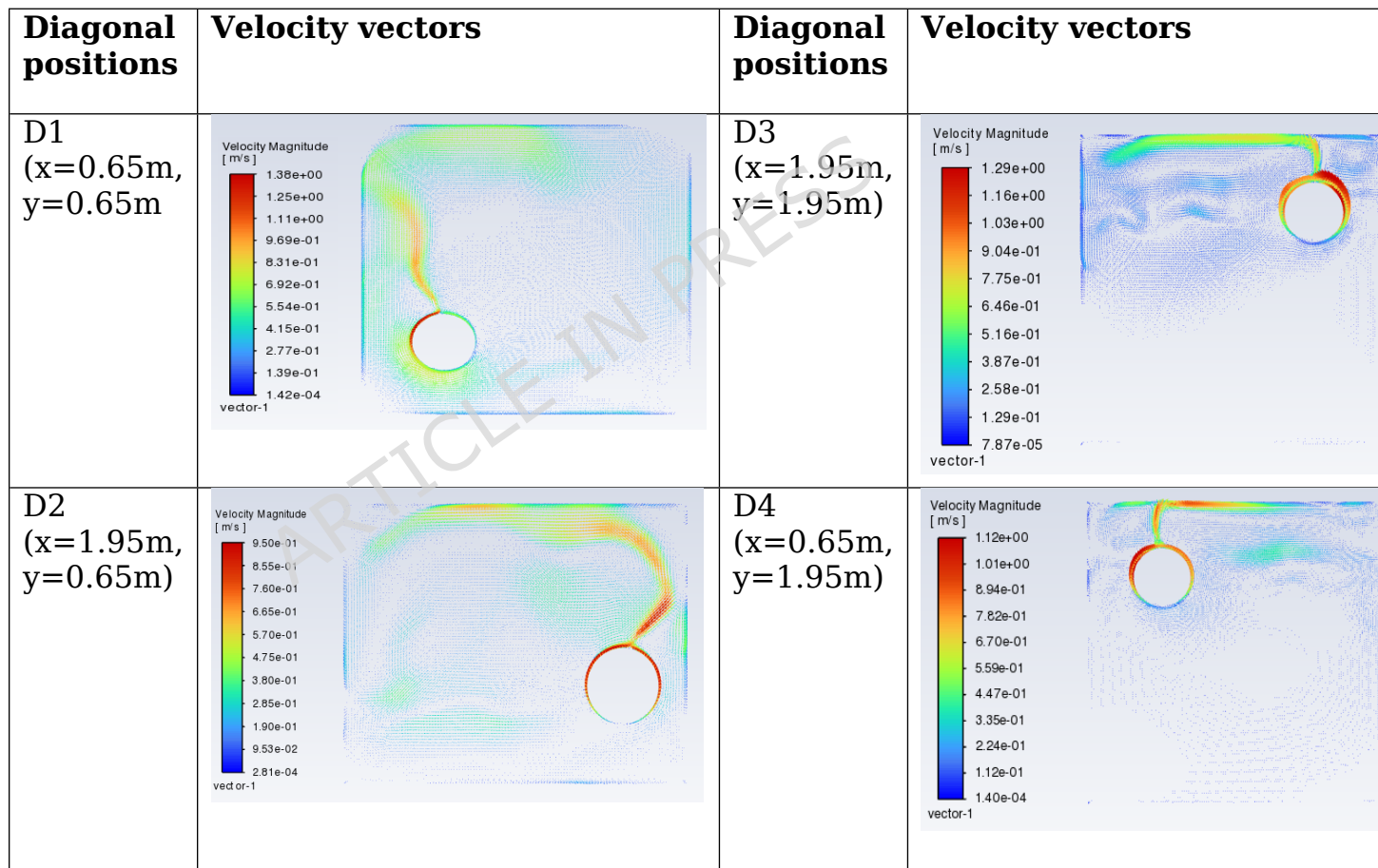
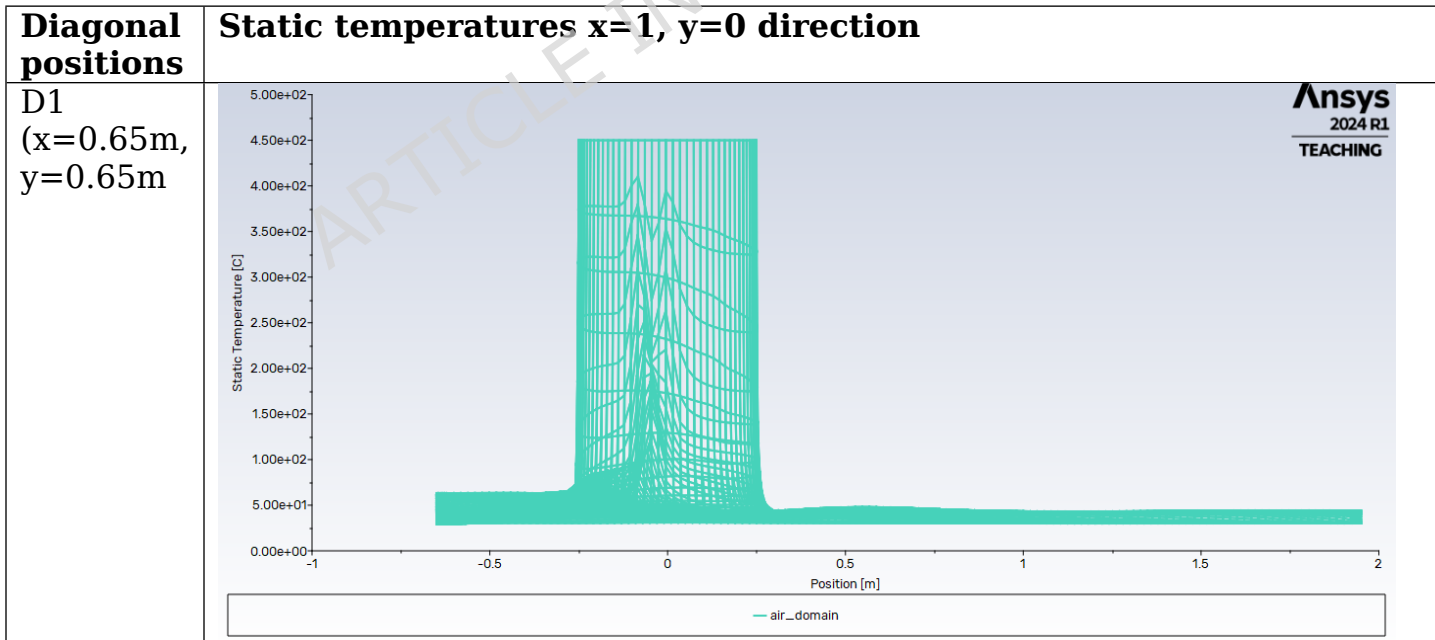


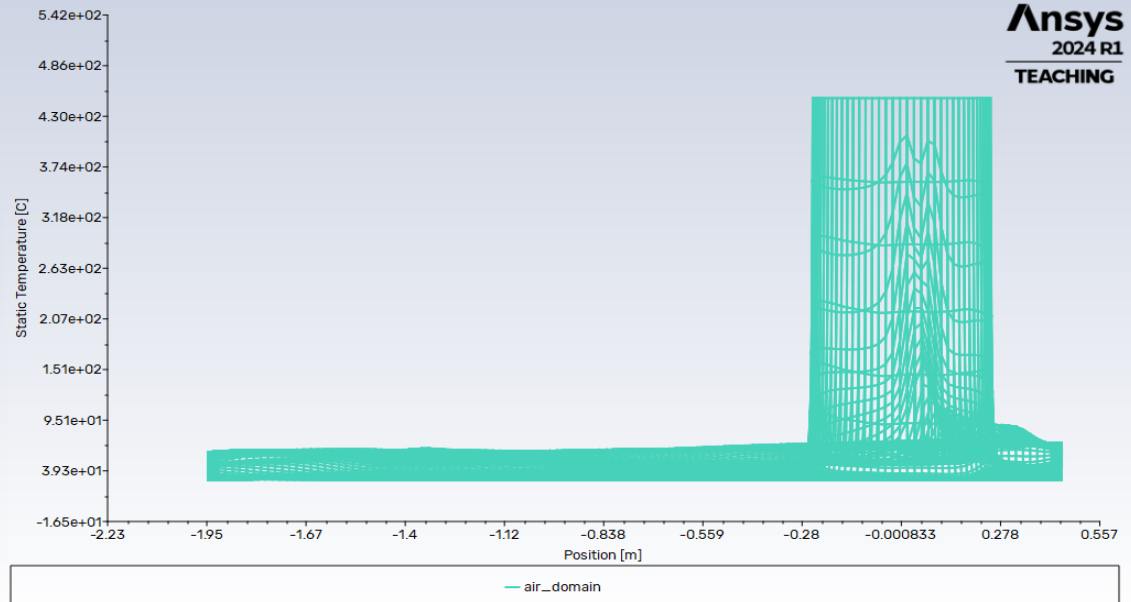
Figure: 14 Velocity vectors for different orientations of heat source in container @ 30 sec

3.8 Static temperature analysis for x=1, y=0 direction

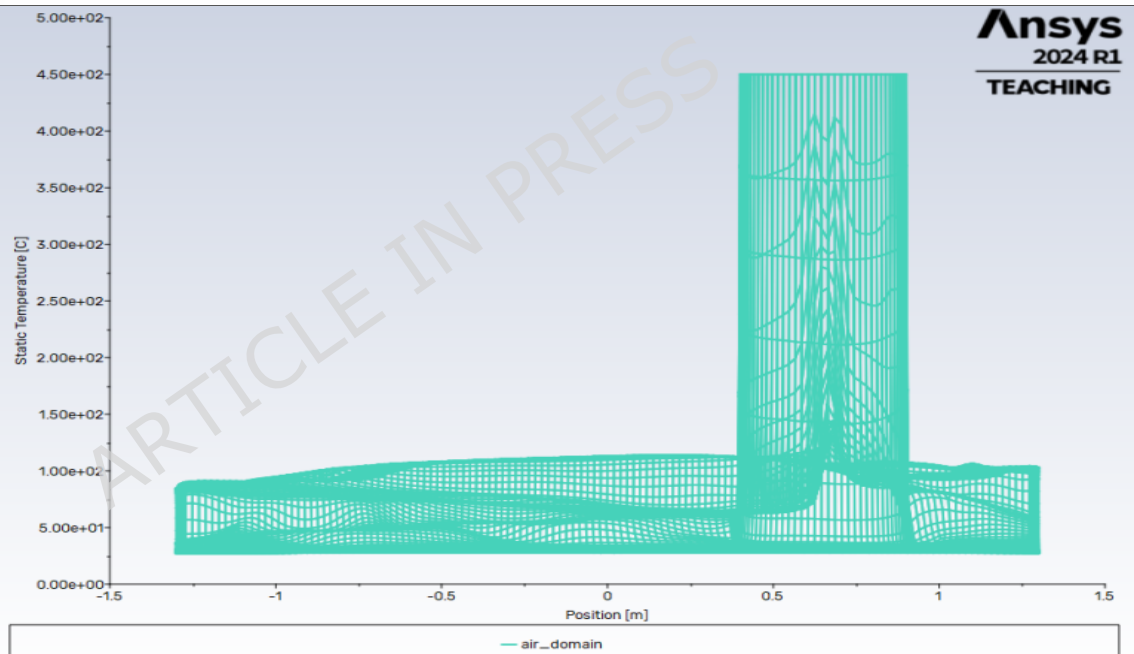
The static temperature profiles along $x = 1$ (vertical) and $y = 0$ (horizontal) provide quantitative insight into heat propagation within the $2.6 \text{ m} \times 2.6 \text{ m}$ enclosure, complementing contour and vector analyses. For D1, the vertical profile shows a sharp peak near the heat source followed by rapid decay, indicating a strong vertical plume with limited lateral spread, while the horizontal profile remains low and uniform away from the source. In D2, the vertical peak is broader and decays more gradually due to wall-induced mixing, and the horizontal profile shows elevated temperatures extending further across the enclosure, reflecting enhanced lateral transport. Upper-corner sources D3 and D4 exhibit truncated vertical peaks near the ceiling, confirming early plume impingement and ceiling-jet formation, while horizontal profiles indicate rapid lateral spreading, with D4 showing more symmetric distribution due to reduced side-wall confinement. These line profiles highlight plume height, buoyancy strength, lateral heat spread, and thermal stratification, providing objective criteria for sensor activation thresholds and nozzle placement. Lower-source configurations produce taller, sharply peaked plumes, whereas upper-source placements are dominated by ceiling-jet-controlled heat transfer, emphasizing the importance of ceiling-mounted sensors and lateral nozzle spacing for effective Hyper Mist suppression in confined marine and industrial compartments.



D2
(x=1.95m,
y=0.65m)



D3
(x=1.95m,
y=1.95m)



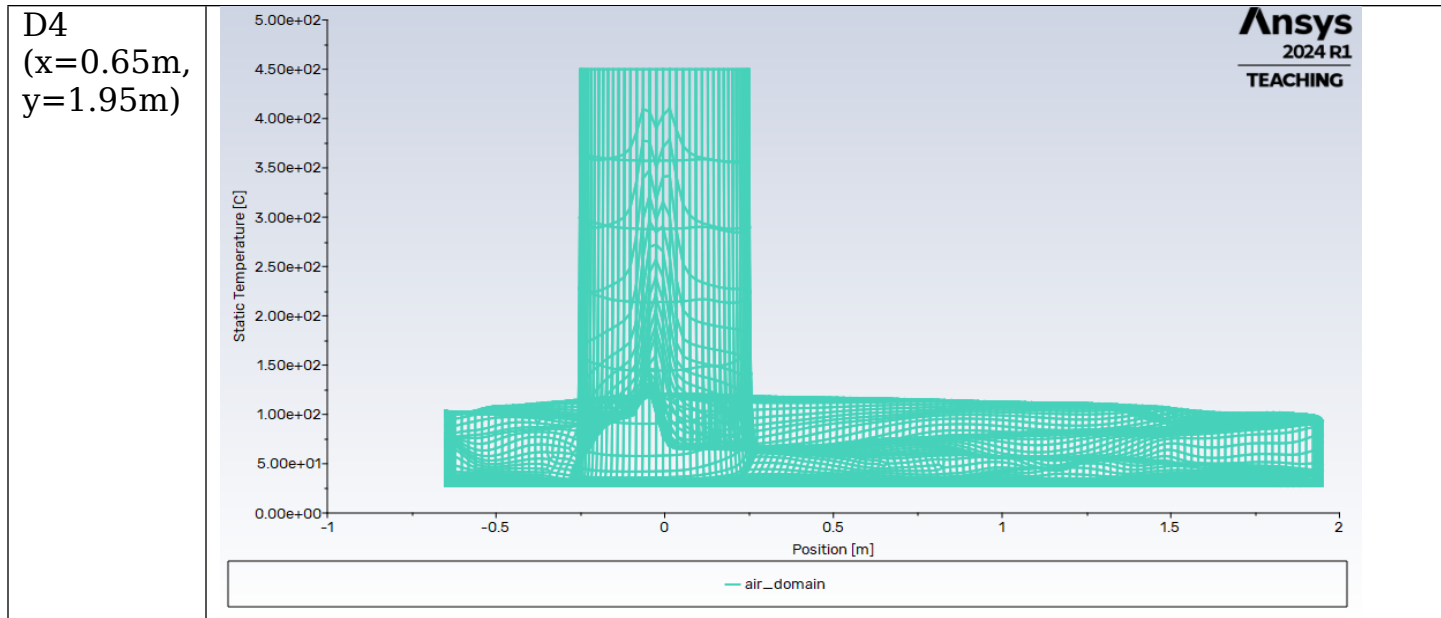


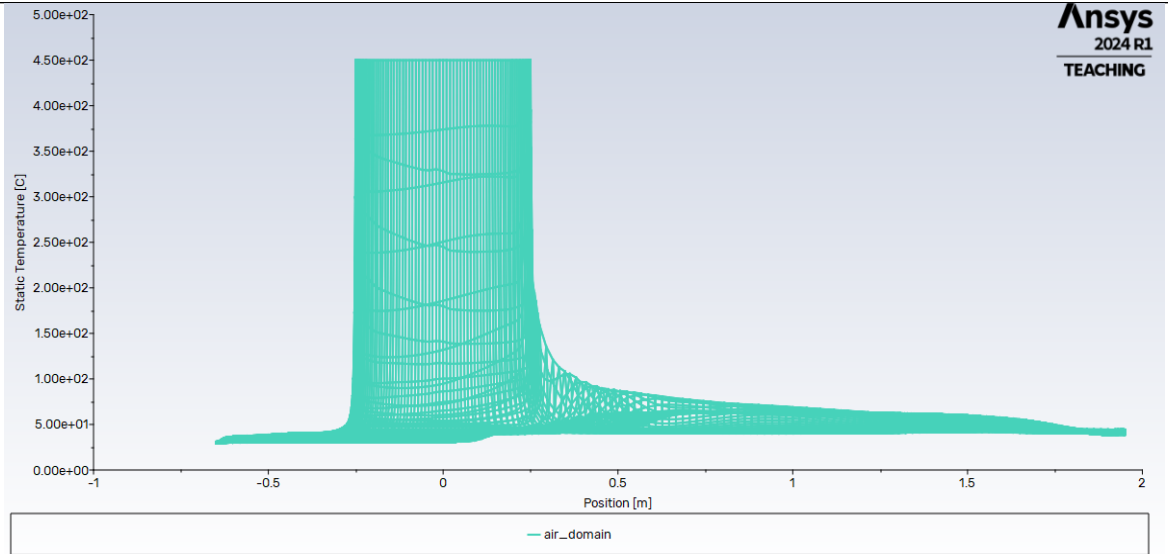
Figure: 15 Static temperatures $x=1$, $y=0$ direction for different orientations of heat source in container @ 30 sec

3.9 Static temperature analysis for $x=1$, $y=0$ direction

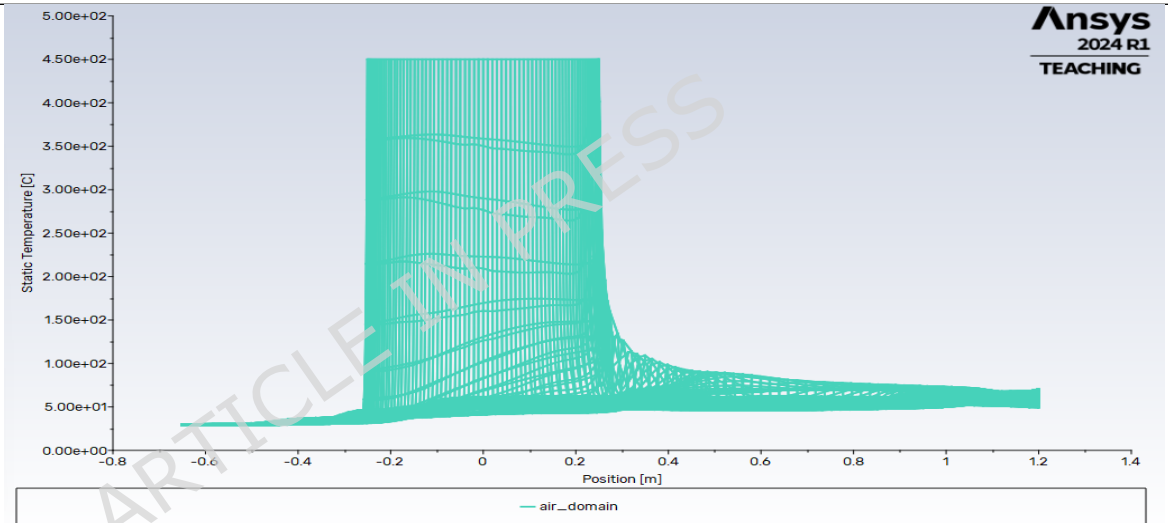
The static temperature profiles along $x = 0$ (vertical boundary) and $y = 1$ (horizontal mid-height) provide quantitative insight into boundary-layer heating, plume-wall interaction, and lateral heat propagation within the $2.6 \text{ m} \times 2.6 \text{ m}$ enclosure. For D1, the $x = 0$ profile shows a sharp rise near the plume with gradual decay toward the wall, indicating minimal early heating, while the $y = 1$ profile demonstrates moderate lateral spread dominated by buoyant entrainment. In D2, the opposite wall remains relatively cool along $x = 0$, whereas the $y = 1$ profile reflects enhanced lateral transport due to plume deflection and wall-induced recirculation. Upper-corner sources D3 and D4 show elevated $x = 0$ temperatures in the upper enclosure, highlighting ceiling-jet influence, while $y = 1$ profiles indicate steep gradients near the source and sustained horizontal heat spread, with D4 exhibiting more uniform lateral transport due to wall proximity. These line-based profiles reveal boundary heating trends, lateral transport efficiency, and thermal stratification, emphasizing that lower-source placements delay wall heating, upper-source positions rapidly affect ceiling and side boundaries, and high-gradient zones are critical for optimal sensor and Hyper Mist nozzle placement, supporting compartment-specific fire-safety design and validating transient thermal equilibrium predictions from full-field CFD analysis.

Diagonal positions	Static temperature $x=0$, $y= 1$ direction
---------------------------	---

D1
($x=0.65\text{m}$,
 $y=0.65\text{m}$)



D2
($x=1.95\text{m}$,
 $y=0.65\text{m}$)



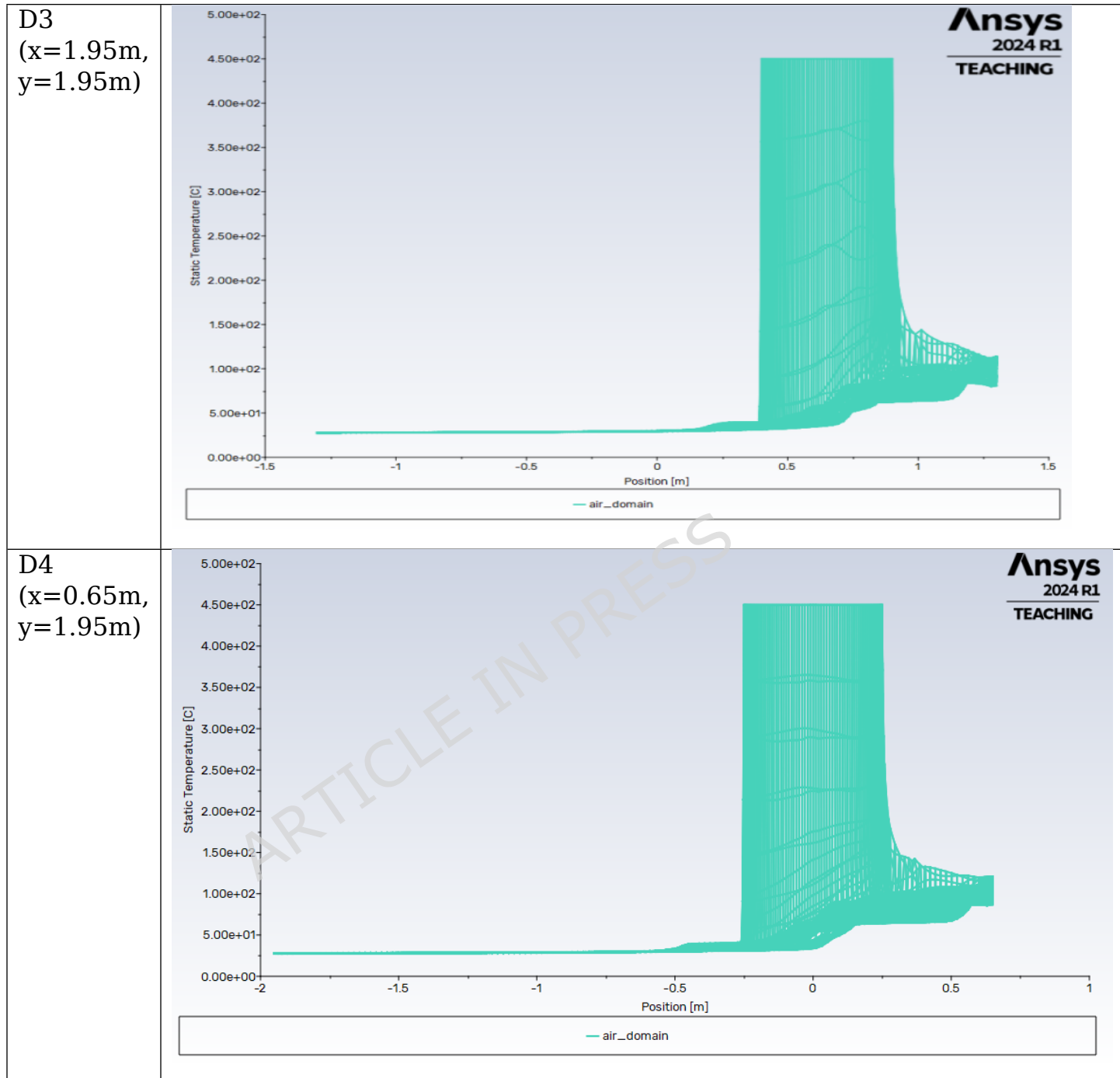


Figure: 16 Static temperatures x=1, y=0 direction for different orientations of heat source in container @ 30 sec

3.10 Key findings and significance

The comprehensive CFD analyses conducted in this study, spanning temperature contours, static profiles, pressure and density fields, velocity

magnitude and vector maps, turbulent kinetic energy, and mass imbalance diagnostics, reveal an interconnected picture of how heat source location governs buoyancy-driven flow dynamics in confined enclosures. One of the most consistent observations across all configurations is that lower heat source positions (D1 and D2) promote tall, vertically oriented thermal plumes with strong upward velocity, steep temperature and density gradients, and pronounced buoyant acceleration. These characteristics produce high-gradient zones directly above the source, which are ideal locations for early thermal and smoke detection sensors. In contrast, upper heat source positions (D3 and D4) rapidly redirect buoyant flow toward the ceiling, forming ceiling-jet dominated regimes and pronounced stratification with reduced vertical penetration but enhanced lateral spread. This behavior underscores that sensor design in such scenarios must prioritize ceiling-level detection rather than mid-height placement, as supported by compartment fire studies emphasizing the importance of detecting lateral spreading and stratification in early stages of fire growth.

The influence of boundary proximity and geometric confinement also emerged as a critical factor across multiple flow variables. Near-wall and corner placements (D2 and D4) generated asymmetric pressure and velocity fields, enhanced recirculation patterns, and broader regions of elevated turbulent kinetic energy. These features not only accelerate lateral heat and smoke transport but also modify the underlying buoyancy forces that drive plume evolution. The asymmetric pressure distributions observed, for example, stronger lateral gradients and significant ceiling pressure maxima in D4—suggest that most discharge systems must account for local flow distortions rather than assume uniform suppression performance across a compartment. Similar CFD investigations into sprinkler orientation and activation timing have shown that misalignment with dominant flow paths can significantly delay suppression responses, reinforcing the need to align mist nozzles with plume trajectories identified via CFD.

The static temperature profiles and line plots provided objective metrics for evaluating plume height, lateral spread, and boundary interaction. In particular, horizontal temperature profiles and boundary-aligned plots illuminated the rate at which thermal energy reaches walls and mid-height regions, which directly affects both sensor activation thresholds and mist interaction zones. These quantitative features extend beyond qualitative contour visualizations, offering design criteria for optimizing sensor trigger points and predicting the earliest likely time of detection. Past research on sensor placement in thermally stratified environments has highlighted that detectors placed too low or too centralized may fail to respond promptly in ceiling-jet environments, often underestimating the maximum plume rise when stratification is present.

From a fire-suppression perspective, the integration of velocity, density, and turbulent kinetic energy fields shows that regions of high velocity and turbulence correspond to preferred pathways for heat and smoke transport, but these same regions can also impede mist droplet penetration due to enhanced entrainment and droplet breakup. High TKE regions near plume shear layers or ceiling jets may reduce mist effectiveness, especially if nozzles are centrally placed and misaligned with the flow. Numerical studies on mist interaction with buoyant flames and suppression effectiveness consistently demonstrate that nozzle location, spray pattern, and discharge angle significantly influence cooling performance and extinguishment time, particularly when the source is near boundaries or corners.

The consistent mass imbalance results and residual convergence across all cases confirm that the observed physical trends are not numerical artefacts but reflect genuine flow behavior with engineering significance. Good mass conservation not only ensures fidelity of velocity and temperature predictions but also builds confidence in using these CFD outcomes for design recommendations. This validation is particularly important for confined marine and industrial spaces where safety systems must be engineered with a high degree of reliability.

This study highlights that heat source elevation influences plume development and potential detection regions, while boundary proximity and enclosure geometry alter temperature, pressure, and velocity distributions. Thermal stratification may affect potential sensor response regions, and turbulence characteristics influence mist transport behavior. The findings provide preliminary guidance regarding potential sensor and Hyper Mist nozzle locations based on predicted plume trajectories and thermal field evolution in confined compartments. However, quantitative optimization involving sensor activation thresholds, response time, field of view, false alarm risk, and suppression performance was beyond the scope of the present study and requires future investigation.

3.11 Thermal Sensor Placement Optimization

Based on the predicted plume dynamics, temperature gradients, velocity vectors, and turbulence characteristics, potential thermal sensor locations may be inferred from regions exhibiting rapid temperature increase and dominant plume trajectories. For lower heat source positions, regions above the source and along plume paths experience earlier thermal rise, whereas upper heat source configurations show stronger ceiling-level temperature accumulation due to plume impingement and ceiling-jet formation. Additional monitoring near plume-ceiling interaction regions may improve early thermal detection because of sustained heat accumulation. The proposed locations are intended as qualitative guidance based on CFD-predicted thermal behavior and do not represent quantitatively optimized sensor placement

considering activation thresholds, response time, or false alarm characteristics.

3.12 Hyper Mist Nozzle Placement and Activation Strategy

The predicted velocity fields, plume trajectories, and turbulent kinetic energy distributions provide qualitative indications of regions where mist-plume interaction may be enhanced. For lower heat source configurations, plume-aligned regions above the source may favor earlier interaction with rising hot gases, whereas upper source locations suggest stronger ceiling-level flow transport associated with ceiling jets. Regions with elevated turbulence may influence droplet dispersion and mixing behavior. These observations provide preliminary guidance regarding potential nozzle placement; however, quantitative evaluation of suppression efficiency, droplet dynamics, activation timing, and nozzle performance is required before establishing optimized Hyper Mist deployment strategies.

4. Conclusion

A comprehensive two-dimensional transient CFD analysis was carried out to investigate buoyancy-driven heat transfer and plume dynamics generated by a circular heat source maintained at 450 °C inside a 2.6 m × 2.6 m confined enclosure. The simulations successfully captured the formation and temporal evolution of thermal plumes governed primarily by natural convection. The results indicate that peak temperature rise remains localized near the heat source during the early stages (≈ 5 s), followed by plume elongation and lateral spreading due to entrainment and confinement effects. Maximum plume velocities in the range of 0.9–1.3 m/s were observed along the plume core and ceiling-jet regions, confirming strong buoyancy-induced acceleration.

Pressure contours revealed low-pressure zones within the plume core, with pressure variations of the order of 10^{-1} Pa, while density contours showed a clear reduction in air density to approximately 0.50–0.70 kg/m³ near high-temperature regions, compared to ambient values. Upper heat source configurations resulted in early plume impingement and formation of strong ceiling jets, significantly enhancing lateral heat transport. Turbulent kinetic energy peaked along plume shear layers and ceiling interaction zones, reaching values of the order of 10^{-2} m²/s², indicating intense mixing. Mass imbalance remained within $\pm 10^{-7}$ kg/s, confirming numerical stability and solution accuracy.

Line temperature plots along selected directions clearly quantified plume height, boundary heating rates, and thermal stratification patterns. Overall, the study demonstrates that heat source location strongly influences plume development, heat accumulation, and flow recirculation within confined

spaces. The proposed sensor placement recommendations are based on temperature distribution and plume behavior and should be considered as preliminary guidance rather than quantitatively optimized locations. The present study is based on an idealized two-dimensional enclosed domain without ventilation openings, internal obstructions, fuel loads, smoke generation, or three-dimensional flow effects. Therefore, the findings should be interpreted as indicating general thermal plume behavior and preliminary design trends, rather than direct prediction of fire dynamics in actual marine or industrial compartments. Future studies should incorporate realistic geometries, ventilation, multiphase effects, and three-dimensional simulations to improve practical applicability.

5. Future Scope

The present two-dimensional study can be extended to three-dimensional simulations to capture corner effects, cross-flows, and realistic compartment geometries encountered in marine and industrial environments. Future work may include coupling combustion and smoke species transport to simulate real fire scenarios rather than prescribed heat sources. Integration of water-mist spray dynamics and droplet evaporation models will enable direct evaluation of Hyper Mist suppression effectiveness. Experimental validation using scaled compartment fire tests is recommended to further strengthen model reliability. Additionally, AI-assisted sensor and nozzle optimization can be explored for adaptive fire detection and suppression systems in complex confined spaces.

Acknowledgement:

The authors wish to thank the management of Tolani Maritime Institute for providing all the required resources.

Funding: The authors present their appreciation to King Saud University for funding this research through the Ongoing Research Funding program (ORF-2025-164), King Saud University, Riyadh, Saudi Arabia. This article was co-funded by the European Union under the REFRESH – Research Excellence for Region Sustainability and High-tech Industries project number CZ.10.03.01/00/22_003/0000048 via the Operational Programme Just Transition and has been done in connection with project Students Grant Competition SP2024/087 Specific Research of Sustainable Manufacturing Technologies “financed by the Ministry of Education, Youth and Sports and Faculty of Mechanical Engineering VŠB-TUO. The article has been done in connection with the project Students Grant Competition SP2024/087”, Specific Research of Sustainable Manufacturing Technologies “financed by the Ministry of Education, Youth and Sports and Faculty of Mechanical Engineering VŠB-TUO”.

Data availability: Data will be made available upon request. Contact the Corresponding Author if someone wants to request the data from this study.

References

- [1] J. G. Quintiere, *Principles of Fire Behavior*. Boca Raton, FL, USA: CRC Press, 2016.
- [2] W. W. Jones, "Buoyant plume behavior in compartment fires," *Fire Safety Journal*, vol. 78, pp. 123-134, 2015.
- [3] C. S. Ahn, B. H. Bang, M. W. Kim, T. G. Kim, J. H. Kim, S.Y. Lee, and K. Y. Kim, "Numerical investigation of smoke dynamics in unconfined and confined environments," *International Journal of Heat and Mass Transfer*, vol. 127, pp. 571-582, Dec. 2018.
- [4] Y. Su, H. Chen, and X. Wang, "Numerical study of water-mist penetration in fire-driven plumes," *Applied Thermal Engineering*, vol. 176, pp. 115-129, 2020.
- [5] IMO MSC Circ. 1165, "Guidelines for Water Mist Systems," International Maritime Organization, 2019.
- [6] Z. Hu, X. He, F. Restuccia, Y. Huang, and G. Rein, "Numerical Study of Self-Heating Ignition of a Box of Lithium-Ion Batteries During Storage," *Fire Technology*, vol. 56, pp. 2603-2621, 2020.
- [7] M. Hosseini and A. Fakhri, "Thermal plume dynamics in cubical enclosures," *International Journal of Heat and Mass Transfer*, vol. 141, pp. 643-655, 2019.
- [8] E. Ghasemi, H. Aminfar, and M. Mohammadpourfard, "Performance evaluation of k- ϵ and k- ω turbulence models in natural convection heat transfer simulations," *Energy Conversion and Management*, vol. 185, pp. 218-230, Apr. 2019.
- [9] P. Freitag, T. Verstraete, and S. Scholl, "Stability analysis and Courant criteria in transient thermal CFD simulations," *Numerical Heat Transfer, Part B: Fundamentals*, vol. 69, no. 2, pp. 152-168, 2016.
- [10] Z. Zheng and Y. Wang, "Influence of heater location on heat spread in confined spaces," *Thermal Science*, vol. 26, pp. 411-423, 2022.
- [11] G. Kallios and D. Papadopoulos, "Sensor network optimization for indoor fire detection," *Fire Safety Journal*, vol. 118, pp. 103-118, 2021.

- [12] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*. Pearson, 2019.
- [13] P. Moin, "Advances in LES for turbulent combustion," *Annual Review of Fluid Mechanics*, vol. 52, pp. 257-289, 2020.
- [14] ANSYS Fluent Theory Guide, Release 2023-R1, Canonsburg, PA.
- [15] D. Pope, *Turbulent Flows*. Cambridge University Press, 2018.
- [16] M. Modest, *Radiative Heat Transfer*. Academic Press, 2021.
- [17] X. Yu, Y. Li, C. Wang, and Z. Zhang, "Transient CFD modeling of buoyant plume rise," *Energy*, vol. 237, pp. 121-134, 2021.
- [18] R. Kumar and S. Gupta, "CFD assessment of compartment heat propagation," *Heat Transfer Engineering*, vol. 44, no. 6, pp. 512-526, 2023.
- [19] Kwang-Sub Lee, J. M. Hyun, and H. Lee, "Air stratification in sealed enclosures," *International Journal of Thermal Sciences*, vol. 182, pp. 106-124, 2022.
- [20] J. Tian and L. Xu, "Grid sensitivity in natural convection CFD," *Computers & Fluids*, vol. 215, 104836, 2021.
- [21] E. J. Barrero and J. R. Forbes, "Courant-limited transient heat simulation," *Numerical Heat Transfer*, vol. 73, pp. 245-260, 2020.
- [22] S. Kalogirou, *Marine Fire Protection Systems*. Elsevier, 2022.
- [23] Jaewook Kim, Minsuk Choi, and Donghyun Shin, "Performance of mist suppression on engine-room fires," *Ocean Engineering*, vol. 242, 110062, 2022.
- [24] Raju George, Gaurav Agarwal, and Sunil Kumar R., "CFD modelling of marine machinery-space fire suppression," *Safety Science*, vol. 150, 105692, 2022.
- [25] L. Patel and N. Shah, "Mist-based suppression on board ships," *Journal of Marine Engineering*, vol. 49, pp. 41-53, 2021.
- [26] IMO MSC Fire Test Procedures Code, 2020 Edition.
- [27] T. Iqbal et al., "Thermal sensor response in engine-room fires," *Fire and Materials*, vol. 46, pp. 1101-1120, 2022.

- [28] N. Banerjee, "Hyper-mist system deployment framework," *Maritime Technology Journal*, vol. 67, no. 3, pp. 63–78, 2023.
- [29] A. Shinde et al., "Compartment fire CFD for nozzles optimization," *Thermal Engineering Review*, vol. 58, pp. 129–142, 2023.
- [30] R. L. Alarcon et al., "Suppression mapping via CFD," *Applied Fire Science*, vol. 9, pp. 211–226, 2023.
- [31] H. Tanaka et al., "Ceiling layer temperature during compartment fires," *Fire Technology*, vol. 57, no. 6, pp. 2657–2676, 2021.
- [32] R. Hu et al., "Natural convection stratification analysis," *Energies*, vol. 15, 11502, 2022.
- [33] S. Dogan and F. Koca, "Thermal plume structure in enclosures," *Energy and Buildings*, vol. 247, 111142, 2021.
- [34] F. Yildirim et al., "Temperature evolution study in closed volumes," *Energy Conversion and Management*, vol. 266, 115243, 2022.
- [35] L. Wang, "Heat rise rate influencing detection time," *Fire Safety Journal*, vol. 122, 103296, 2021.
- [36] A. H. Memon et al., "Plume acceleration in corner fires," *Combustion Science and Technology*, vol. 195, pp. 86–101, 2023.
- [37] Z. Chen et al., "Computational assessment of fire-heat spread," *Thermophysics and Heat Transfer*, vol. 37, no. 4, pp. 781–797, 2023.
- [38] J. Habib et al., "Ceiling jet formation analysis," *Fire Technology*, vol. 59, pp. 423–441, 2023.
- [39] T. B. Gatski, *Turbulence Modeling for CFD*. Springer, 2022.
- [40] W. Sun and L. Zhao, "Boundary layer near high-temperature plates," *Experimental Thermal Fluid Science*, vol. 138, 110500, 2022.
- [41] A. Sandhu et al., "Mist droplets & plume coupling," *International Journal of Multiphase Flow*, vol. 143, 103686, 2021.
- [42] J. Singh et al., "Temperature decay under droplet cooling," *Applied Thermal Engineering*, vol. 204, 118064, 2022.
- [43] Q. Lin et al., "Heat source offset effect in enclosures," *Energy*, vol. 234, 121058, 2021.

- [44] B. Kumar et al., "Influence of heater placement on ceiling dynamics," *Case Studies in Thermal Engineering*, vol. 35, 102138, 2022.
- [45] M. Ito et al., "Transient turbulent plume modelling," *Journal of Fluid Engineering*, vol. 145, 031203, 2023.
- [46] O. Park et al., "High-fidelity LES on fire-room models," *Combustion and Flame*, vol. 245, 112–129, 2022.
- [47] A. Oberkampf and T. Trucano, *Verification and Validation in Computational Science*. Cambridge University Press, 2020.
- [48] J. Mahapatra et al., "CFD validation for heat source simulations," *Heat and Fluid Flow Journal*, vol. 58, 104869, 2021.
- [49] K. Das et al., "Mesh metrics for plume stability," *Computers & Fluids*, vol. 228, 105064, 2022.
- [50] ASHRAE Handbook of Fundamentals, 2023.
- [51] H. Park and S. Lee, "Accuracy in conduction-convection coupling," *Thermal Science and Engineering Progress*, vol. 40, 101212, 2023.
- [52] ISO-15371 Marine Mist System Standard, 2023.
- [53] F. Ugale et al., "Thermal hotspot simulation evaluation," *Journal of Thermal Engineering*, vol. 9, pp. 320–336, 2023.
- [54] D. Moradi, "Marine fire CFD benchmark dataset," *Ocean Systems and Safety*, vol. 14, no. 2, pp. 142–159, 2024.
- [55] M. Chan et al., "Sensor activation time prediction via CFD," *Fire Detection Research*, vol. 12, pp. 51–69, 2024.
- [56] D. Kan and S. Feng, "Full-Scale Experimental Investigation on the Thermal Control of a Water Mist System in a Road Tunnel Under Single-Source and Double-Source Fire Scenarios," *Fire*, vol. 8, no. 8, p. 317, 2025.
- [57] V. Parsa et al., "Computational Fluid Dynamics of Compartment Fires: A Review of Methods and Applications," *Appl. Sci.*, vol. 15, no. 5, p. 2342, 2025.
- [58] Hakan Coşanay , Hakan F. Öztop , Muhammed Gür , Eda Bakır , "Analysis of turbulent wall jet impingement onto a moving heated body," *International Journal of Numerical Methods for Heat & Fluid Flow*, vol. 32, p. 2938–2963, 2022.

- [59] H. F. Öztop, E. Bakır, F. Selimefendigil, M. Gür & H. Coşanay, "Effects of inclined plate in a channel to control melting of PCM in a body inserted on the bottom wall," *Experimental Techniques* Volume 47, pages 123-137, 2023.
- [60] Hakan F. Oztop, Muhammed Gür; Fatih Selimefendigil; Hakan Coşanay, "Analysis of melting of phase change material inserted in a block via impinging turbulent slot jet," *International Journal of Numerical Methods for Heat & Fluid Flow*, vol. 33, p. 3467-3491. 2023.
- [61] Hakan F. Öztop, Muhammed Gür , Fatih Selimefendigil , Hakan Coşanay "Impacts of inclined plate on melting time for phase change material filled blocks: Experimental and numerical analysis," *Thermal Science and Engineering Progress*, vol. 48, p. 102427, 2024.
- [62] Muhammed Gur; Hakan Oztop; Nirmalendu Biswas; Fatih Selimefendigil, "Three-dimensional analysis of turbulent twin-swirling jets onto a heated rectangular prism in a channel," *International Journal of Numerical Methods for Heat & Fluid Flow*, vol. 35, p. 1137-1171, 2025.
- [63] H. F. Oztop, I. Dagtekin, A. Bahloul, and Y. Varol, "Numerical simulation of magnetohydrodynamic buoyancy-induced flow in a non-isothermally heated square enclosure," *Communications in Nonlinear Science and Numerical Simulation*, vol. 14, no. 3, pp. 770-778, Mar. 2009.
- [64] L. Kolsi, O. Mahian, H. F. Oztop, W. Aich, M. N. Borjini, N. Abu-Hamdeh, and H. Ben Aissia, "3D buoyancy-induced flow and entropy generation of nanofluid-filled open cavities having adiabatic diamond shaped obstacles," *Entropy*, vol. 18, no. 6, p. 232, Jun. 2016.
- [65] Eda Bakır Gür, Erdem Işık, Aynur Uçar, "Numerical analysis of PCM integrated in solar dryers for thermal efficiency", *Ain Shams Engineering Journal*, vol. 17, Issue 1, p. 103847, Jan 2026, <https://doi.org/10.1016/j.asej.2025.103847>.