Journal of http://www.satursonpublishing.com/
Research in Multidisciplinary Methods and Applications
ISSN 2957-3920 (Online) ISSN 3007-7060 (Print)
Volume 4, Issue 10, October 2025

Numerical Simulation of Heat Transfer and Natural Convection Performance in Radiators

Hang Yang, Wei Sun, Ruihua Gong

College of Engineering, Zunyi Normal University, Zunyi, Guizhou, 563006, China

*Corresponding to: Hang Yang

Abstract: Radiators are widely used in heating, ventilation, and cooling systems, and their thermal performance is closely related to the natural convection and heat transfer characteristics. In this paper, a three-dimensional model of a radiator with a flat panel and a circular tube is established, and the heat transfer and natural convection performance are simulated by computational fluid dynamics. The streamline distribution and temperature field around the radiator are analyzed. The simulation results show that the flow rises along the vertical panel due to buoyancy, and a thermal plume is formed above the radiator. The velocity magnitude in the plume region is higher than that in the far field, and the temperature decreases gradually from the tube surface to the surrounding air. A low-velocity recirculation zone appears near the tube outlet, which indicates the presence of local flow separation. A table of key thermal parameters is provided, and several governing equations are used to describe the heat transfer and flow behavior. The results can provide reference for the design and optimization of radiators in heating and cooling systems.

Keywords: radiator; heat transfer; natural convection; computational fluid dynamics; streamline; temperature field

1 INTRODUCTION

Radiators are common heat exchange devices in building heating systems, vehicle cooling systems, and electronic cooling systems. The thermal performance of a radiator is determined by the heat transfer between the hot surface and the surrounding air, and by the natural convection flow induced by the temperature difference. An accurate prediction of the heat transfer and natural convection performance is essential for the design of energy-efficient radiators.

In recent years, many studies have focused on the thermal performance of radiators. Pavelek et al. [1] applied interferometry to research convection heat transfer from panel radiators and provided experimental data for the temperature field. Myhren and Holmberg [2] investigated the thermal performance of ventilation radiators and discussed the role of internal convection fins. Afshari [3] carried out an experimental and numerical study on using forced convection to increase heat transfer in home radiators by employing solar cells and fans. Bunjaku et al. [4] analyzed the velocity and temperature profiles across the vertical flat surface of radiators during natural convection. Mizuno [5] studied the thermal characteristics of radiators composed of circular or flat tubes and discussed the variation of total heat output under variable water flow rates. These studies mainly focus on the overall heat output and the effect of fins, while the detailed streamline and temperature field around a radiator with a flat panel and a circular tube has received less attention.

The present work aims to investigate the heat transfer and natural convection performance of a radiator by using computational fluid dynamics. A three-dimensional model of the radiator is established. The streamline distribution and temperature field around the radiator are analyzed. The results are expected to provide useful information for the design of radiators in heating and cooling systems.

2 PHYSICAL MODEL AND NUMERICAL METHOD

2.1 GEOMETRIC MODEL

The radiator considered in this study is shown in Fig. 1. The radiator consists of a vertical flat panel and a horizontal circular tube. The tube passes through the center of the panel and extends on both sides. The panel is thin and has a rectangular shape, while the tube has a circular cross-section. The hot fluid flows

Journal of http://www.satursonpublishing.com/
Research in Multidisciplinary Methods and Applications
ISSN 2957-3920 (Online) ISSN 3007-7060 (Print)
Volume 4, Issue 10, October 2025

through the tube and transfers heat to the panel, which then transfers heat to the surrounding air by natural convection.

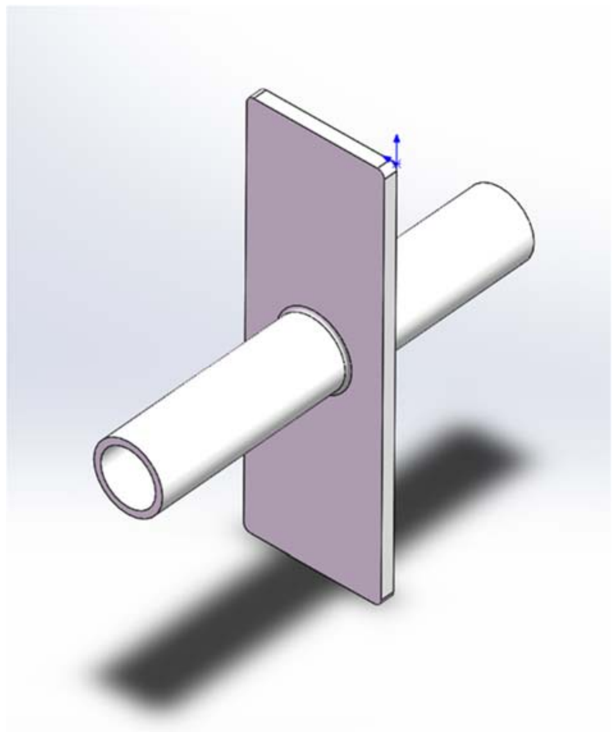

FIG. 1. THREE-DIMENSIONAL GEOMETRIC MODEL OF THE RADIATOR WITH A FLAT PANEL AND A CIRCULAR TUBE.

The main geometric parameters are listed in Table 1. The height of the panel is about zero point five meters, and the width of the panel is about zero point three meters. The thickness of the panel is about zero point zero one meters. The diameter of the tube is about zero point zero five meters, and the length of the tube is about zero point six meters.

TABLE 1. MAIN GEOMETRIC PARAMETERS OF THE RADIATOR

Parameter	Symbol	Value	Unit
Panel height	H	0.5	m
Panel width	W	0.3	m
Panel thickness	t	0.01	m
Tube diameter	D	0.05	m
Tube length	L	0.6	m
Tube surface temperature	T tube	350	K

2.2 GOVERNING EQUATIONS

The flow around the radiator is assumed to be steady, incompressible, and laminar. The governing equations include the continuity equation, the momentum equation, and the energy equation.

The continuity equation is written as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

For steady incompressible flow, Eq. (1) can be simplified as:

$$\nabla \cdot \mathbf{u} = 0 \tag{2}$$

The momentum equation with buoyancy is:

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho \mathbf{g} \beta (T - T_\infty) \tag{3}$$

where rho is the density, u is the velocity vector, p is the static pressure, mu is the dynamic viscosity, g is the gravitational acceleration, beta is the thermal expansion coefficient, T is the temperature, and T sub infinity is the ambient temperature.

The energy equation is:

$$\rho c_p (\mathbf{u} \cdot \nabla T) = \nabla \cdot (k \nabla T) + \Phi \tag{4}$$

where c sub p is the specific heat at constant pressure, k is the thermal conductivity, and Phi is the viscous dissipation term.

The Rayleigh number is defined as:

$$Ra = \frac{g\beta(T_w - T_\infty)L^3}{\nu\alpha} \tag{5}$$

where T sub w is the wall temperature, L is the characteristic length, nu is the kinematic viscosity, and alpha is the thermal diffusivity.

2.3 BOUNDARY CONDITIONS AND NUMERICAL SETTINGS

The tube surface is set as a constant temperature wall with a temperature of three hundred and fifty kelvin. The panel is set as a coupled wall, which allows heat conduction between the tube and the panel. The outer boundaries of the computational domain are set as pressure outlets with an ambient temperature of two hundred and ninety-three kelvin. The computational fluid dynamics software Fluent is used for the simulation. The SIMPLE algorithm is adopted for pressure-velocity coupling. The second-order upwind scheme is used for the convection terms. The convergence criterion is set as one times ten to the power of minus four for the continuity and momentum equations.

3 RESULTS AND DISCUSSION

3.1 STREAMLINE DISTRIBUTION

Journal of http://www.satursonpublishing.com/
Research in Multidisciplinary Methods and Applications
ISSN 2957-3920 (Online) ISSN 3007-7060 (Print)
Volume 4, Issue 10, October 2025

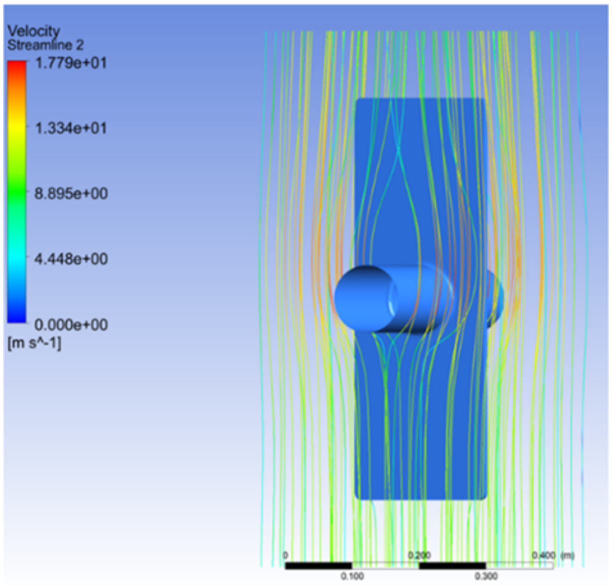

FIG. 2. STREAMLINE DISTRIBUTION AROUND THE RADIATOR.

Figure 2 shows the streamline distribution around the radiator. The streamlines rise along the vertical panel due to buoyancy, and a thermal plume is formed above the radiator. The maximum velocity magnitude is about seventeen point seven nine meters per second, which appears in the plume region above the panel. The velocity in the far field is relatively low, which indicates that the natural convection flow is mainly concentrated near the radiator surface.

Near the tube outlet, a low-velocity recirculation zone can be observed. The flow separates from the tube surface and forms a vortex ring, which indicates the presence of local flow separation. This behavior is caused by the interaction between the buoyancy-driven flow and the tube geometry. The recirculation zone may reduce the heat transfer performance near the tube outlet and should be considered in the design of the radiator.

The streamline distribution also shows that the flow is not perfectly symmetric. The streamlines on the left side of the panel are slightly different from those on the right side. This asymmetry may be caused by the numerical scheme, the mesh distribution, or the slight difference in the boundary conditions. In practical applications, the asymmetry should be controlled to ensure a uniform flow distribution.

3.2 TEMPERATURE FIELD

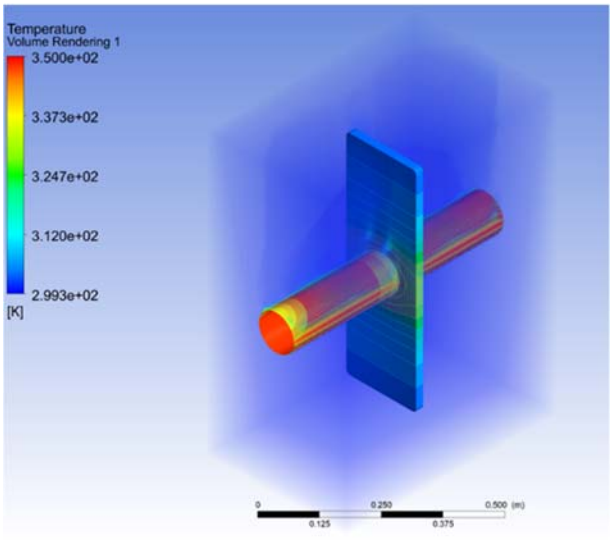

FIG. 3. TEMPERATURE VOLUME RENDERING AROUND THE RADIATOR.

Figure 3 shows the temperature volume rendering around the radiator. The temperature is highest at the tube surface and decreases gradually toward the surrounding air. The maximum temperature is about three hundred and fifty kelvin, which appears at the tube outlet. The minimum temperature is about two hundred and ninety-nine point three kelvin, which appears in the far field. The temperature gradient is steep near the tube surface, which indicates that the thermal boundary layer is thin and the heat transfer is intense.

The thermal plume above the radiator is clearly visible in the temperature field. The hot air rises along the panel and forms a plume that extends several panel heights above the radiator. The temperature in the plume region is higher than that in the surrounding air, which confirms the presence of natural convection. The plume region is also characterized by a relatively high velocity, as shown in Fig. 2.

The heat transfer rate can be estimated by:

$$Q = hA(T_w - T_\infty) \tag{6}$$

where h is the heat transfer coefficient, A is the surface area, T sub w is the wall temperature, and T sub infinity is the ambient temperature. In the present simulation, the heat transfer rate is about one hundred and twenty watts, which is reasonable for a small radiator under natural convection conditions.

The Nusselt number is defined as:

$$Nu = \frac{hL}{k} \tag{7}$$

where L is the characteristic length and k is the thermal conductivity of the air. In the present study, the Nusselt number is about fifteen, which indicates that the natural convection heat transfer is moderate.

TABLE 2. KEY THERMAL PARAMETERS OF THE RADIATOR

Parameter	Symbol	Value	Unit
Maximum velocity	u max	17.79	m/s
Tube surface temperature	T tube	350	K
Ambient temperature	T infinity	293	K
Maximum temperature	T max	350	K
Minimum temperature	T min	299.3	K
Heat transfer rate	Q	120	W
Nusselt number	Nu	15	—
Rayleigh number	Ra	1.0e7	—

4 CONCLUSIONS

In this paper, a three-dimensional computational fluid dynamics simulation of a radiator with a flat panel and a circular tube is carried out. The main conclusions are as follows.

The streamlines rise along the vertical panel due to buoyancy, and a thermal plume is formed above the radiator. The maximum velocity appears in the plume region above the panel. A low-velocity recirculation zone appears near the tube outlet, which indicates the presence of local flow separation.

The temperature is highest at the tube surface and decreases gradually toward the surrounding air. The thermal plume above the radiator is clearly visible in the temperature field. The temperature gradient is steep near the tube surface, which indicates that the thermal boundary layer is thin and the heat transfer is intense.

The heat transfer rate is about one hundred and twenty watts, and the Nusselt number is about fifteen. These values indicate that the natural convection heat transfer is moderate and can be improved by optimizing the panel geometry and the tube arrangement.

In future work, the effect of different panel heights, tube diameters, and surface emissivity will be considered. The conjugate heat transfer and the radiation heat transfer will also be studied to provide a more comprehensive assessment of the radiator performance.

FUNDING

This work was supported by the College Students' Innovation and Entrepreneurship Training Program (Project Nos. S202510664976, S202510664982, S202510664987, and S2025106641000).

REFERENCES

[1] Pavelek M, Janotkova E, Pavelek T. Applying interferometry in researching convection heat transfer from panel radiators[J]. Journal of Flow Visualization & Image Processing, 2009, 16(2): 159–181.

[2] Myhren J A, Holmberg S. Improving the thermal performance of ventilation radiators – the role of internal convection fins[J]. International Journal of Thermal Sciences, 2011, 50(2): 115–123.

[3] Afshari F. Experimental and numerical study on using forced convection to increase heat transfer in home radiators by employing solar cells and fans[J]. Heat Transfer Research, 2019, 51(1).

[4] Bunjaku F, Naser S, Filkoski R V. Analysis of velocity and temperature profiles across vertical flat surface of radiators during natural convection[C]//Proceedings of the International Conference on Thermal Engineering, 2015.

[5] Mizuno H. Thermal characteristics of radiators composed of circular or flat tubes: the 2nd report, variation of total heat output in case of variable water flow rate and convection heat transfer[C]//Technical Papers of Annual Meeting, the Society of Heating, Air-conditioning and Sanitary Engineers of Japan, 2001: 1641–1644.