Cirebon Annual Multidisciplinary International Conference (CAMIC 2024)

Temperature Distribution Fire Furnaces: Experiment & Numerical Approach

1st Irfan Santosa
Department of mechanical engineering, faculty of engineering and computer science
Universitas Pancasakti
Tegal, Indonesia
ci_ulya@yahoo.co.id

2nd Ahmad Farid
Department of mechanical engineering, faculty of engineering and computer science
Universitas Pancasakti
Tegal, Indonesia
ci_ulya@yahoo.co.id

3rd Okky Hendra Hermawan
Department of mechanical engineering, faculty of engineering and computer science
Universitas Pancasakti
Tegal, Indonesia
ci_ulya@yahoo.co.id

1st Agus Wibowo
Department of mechanical engineering, faculty of engineering and computer science
Universitas Pancasakti
Tegal, Indonesia
ci_ulya@yahoo.co.id

***Abstract*—A fire furnace is a combustion tool, that has a function to produce sufficient heat for combustion purposes. The even distribution of temperature can accelerate the combustion process. This research aims to investigate the temperature distribution in a fire furnace, both experimentally and numerically, to analyze temperature distribution during combustion. The air spray speed in the stove will be varied at 1.36 m/s, 1.45 m/s, and 1.53 m/s. The research method involves experimental observation and numerical simulation using Computational Fluid Dynamics (CFD). The results showed an air jet speed of 1.36 m/s, the experimental and numerical temperature values are 619.2 °C and 621.5 °C with a percentage error of 0.37%. For an air jet speed of 1.45 m/s, the temperatures obtained in the experiment and simulation are 637.3 °C and 642.5 °C with an error percentage of 0.81%. Finally, at an air jet speed of 1.53 m/s, the experimental and numerical temperatures are 659 °C and 670.9 °C with an error percentage of 1.80%. The research concludes that both experimental and numerical methods provide a consistent and accurate temperature distribution.**

Keywords— ***temperature distribution; fire furnace; experiment; numeric***

I. INTRODUCTION

A fire furnace is a heating device designed to generate and distribute heat for various purposes. Its primary function is to produce and deliver heat to a designated space or system. Furnaces are commonly used in residential, commercial, and industrial settings [1], [2]. While widely used for heating and industrial applications, fire furnaces have several weaknesses or limitations depending on their design, operation, and maintenance. Some of the problems with fire furnaces are uneven temperature distribution, caused by inconsistent heating, poor furnace design, improper fuel-air mixing, or inadequate insulation. Their impact reduces product quality and efficiency in the combustion process. Therefore, many studies have been carried out that pay attention to heat transfer in furnaces using numerical and experimental methods [3],[4].

Liu et al. [5] experimentally and numerically studied 2-D and 3-D temperature distribution in coal-burning furnaces using the least squares QR (LSQR) decomposition method. That research demonstrates that numerical reconstruction accurately models temperature distribution in both laboratory-scale and large-scale furnaces, while

experimental reconstruction reasonably reflects the main features of actual temperature distribution. Karabas [6] developed a three-dimensional fire resistance test furnace and used the ANSYS Fluent program to analyze its computational fluid dynamics, concluding that the internal temperature distribution significantly influences the average temperature of the furnace volume. Shanmukharadhya et al. [7], performed numerical and experimental studies on industrial biomass-fueled furnaces with tangential overfire air systems. The flue gas path profile along the furnace's height and depth demonstrated that the airflow system's impact on bagasse combustion could be accurately predicted and validated against measured results. Rezazadeh [8] conducted experimental and numerical research on heat transfer in steel industry furnaces using ANSYS Fluent. That research reveals that the realizable k-ε model provides more accurate load center line temperature predictions with experimental data.

Liao et al. [9] proposed the characterization of thermal deviations in the furnace body and the tubes using Fluent software. Numerical results show that with the increase of thermal load, the flame structure in the furnace gradually becomes concentrated, the average flue gas temperature increases and the outlet temperature of the furnace chamber increases by 19%, with the thermal efficiency decreasing by 5.11%.

The research highlights the importance of numerical studies as a complement to experimental research, particularly for fire furnaces, to analyze temperature, pressure, and velocity distributions during combustion. This study aims to design and construct a fire furnace using oil as fuel and experimentally test it by varying the air spray speed. Unlike previous research, this fire furnace features a compact design. It incorporates an adjustable air spray speed at the bottom of the combustion chamber, eliminating the need for a compressor to supply air.

II. METHOD

This research applies two types of methods: experimental and numerical. The experimental process involved making a furnace with the following dimensions: 61 cm long, 60 cm wide, and 34.5 cm high, as seen in Figure 1.

At the bottom of the furnace, a fan is installed, which can be adjusted using a potentiometer knob. This fan assists the combustion process by blowing air into the combustion chamber. It is equipped with three-speed options: 1.36 m/s, 1.45 m/s, and 1.53 m/s. Air is blown into the combustion chamber for 5 minutes. The fuel used in the furnace is waste oil, which is placed within the combustion chamber.

To analyze the temperature distribution during the combustion process, a thermocouple is used as a measuring instrument. Additionally, a numerical comparison is conducted using the Computational Fluid Dynamics (CFD) method to predict the temperature distribution in the furnace. The problem-solving approach is based on fundamental equations, including the continuity equation, the momentum equation, and the energy equation [10]. To illustrate the research methodology, the process is depicted in Figure 2.

a. Equation of conservation of mass:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \text{ , ... } (1)$$

b. Conservation of momentum equation:

$$\rho\left[u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right]\right.$$

$$\rho\left[u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right]\right.$$

....(2)

c. Energy conservation equation:

$$\frac{\partial}{\partial t}[\rho\,(e + \frac{V^2}{2})] + \nabla\,[\rho\,(e + \frac{V^2}{2})\vec{V}], \text{ ....(3)}$$

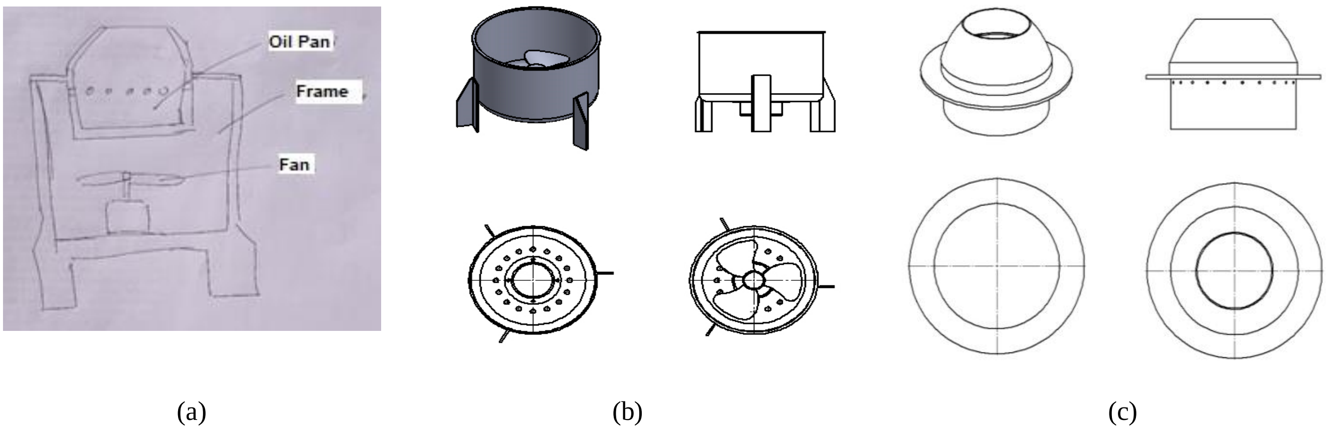

Figure 1. Fire furnace design (a) sketch, (b) firebox frame, fan mount, and (c) furnace combustion

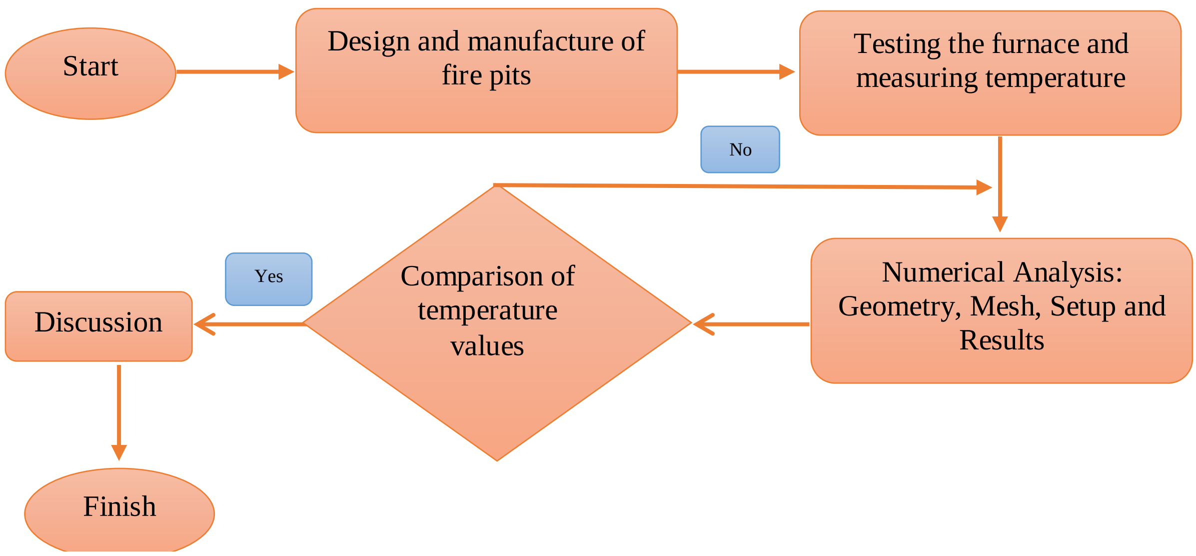

Figure II. Flow chart research

III. RESULTS AND DISCUSSION

Geometry

For numerical analysis, the furnace is represented in terms of 3D geometry, as shown in Figure 3. After the design of the furnace geometry has been determined, the next step is to determine the boundaries of the geometry field inside the furnace, as shown in Figure 4. The parameters defining the boundary conditions, along with their values, are presented in Table 1. The next step in the modeling process is to discretize the computational domain by dividing it into smaller discrete sub-domains, (mesh) in the plane of the furnace. In this study, a mesh with a tetrahedral structure consisting of 484,460 elements was used. A detailed visualization of this mesh is shown in Figure 5.

Setup

The setup process in Computational Fluid Dynamics (CFD) analysis is used to formulate the assumptions underlying the numerical modeling of the furnace. These assumptions are important to simplify the problem and enable efficient simulations. In this study, a steady-state model was chosen as the basis for modeling, with justification based on temperature data obtained from experiments. In solving this case, the SIMPLE method was chosen, which utilizes pressure variables to correct the estimated values. To achieve a higher level of accuracy in the calculation, the discretization process needs to be performed using a second-order scheme. The setup process is shown in Table 2.

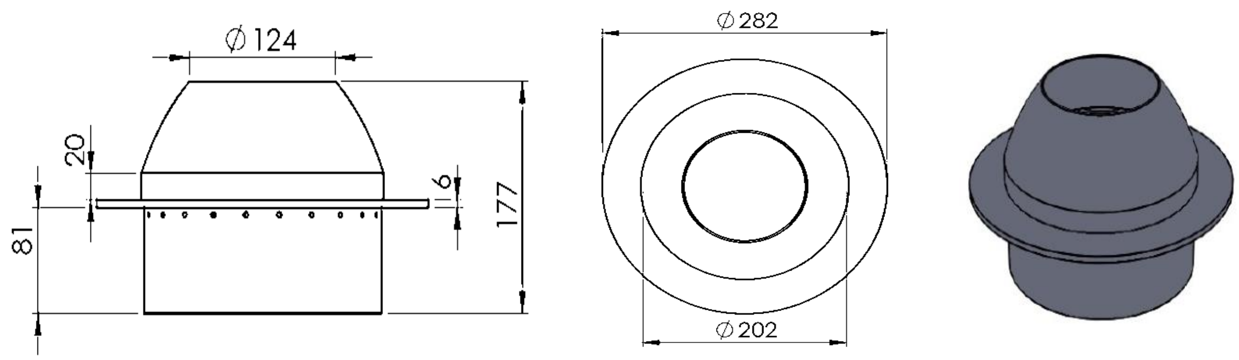

Figure III. Fire furnace geometry

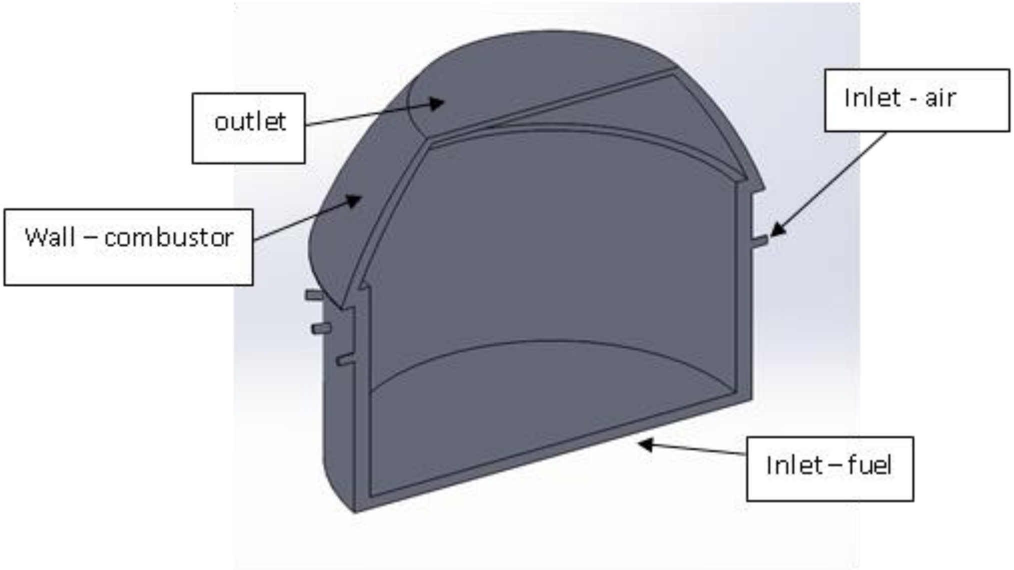

Figure IV. Boundary conditions of the furnace combustion chamber

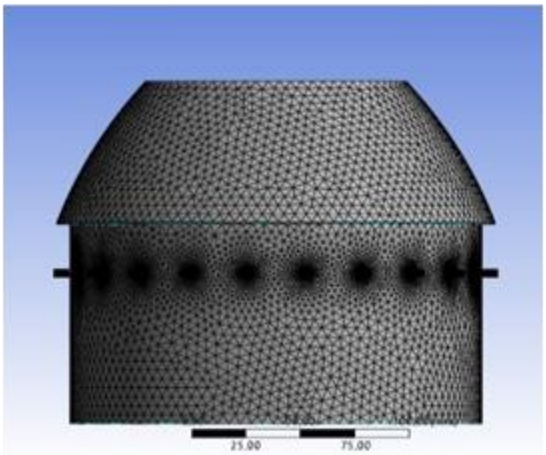

Figure V. Mesh on the fire furnace

TABLE I. INITIALIZE BOUNDARY CONDITIONS

Name	**Boundary condition**	**Value**
Air Inlet	Velocity	1,36 m/s; 1,45 m/s ; 1,53 m/s
Fuel Inlet	Fuel oil air	870 kg/m^3
Outlet	Pressure Outlet	1 atm
Combustor Casing	Wall	adiabatic

TABLE II. SIMULATION PARAMETER

Setup Parameter	**Setting**
Tipe *Solver*	Pressure based
Turbulensi Model	K-epsilon
Wall treatment	Standard wall function
Model species	Non-premixed combustion
Spatial discretization	Second order

Figure VI. Assess the rotational speed of the fan usimg a tachometer

Comparison of experimental results with simulations

In this furnace experiment, oil is used as fuel in the furnace, and the temperature in the combustion chamber is monitored using a thermocouple temperature gauge. Furthermore, the fan rotation speed was varied three times. Quantitative measurements of the rotation speed were made using a tachometer instrument, as shown in Figure 6. In the experiments examining the effect of fan rotational speed variation on furnace temperature, three speeds of 1.36 m/s, 1.45 m/s, and 1.53 m/s were used. After burning for 5 minutes, the average temperatures inside the furnace chamber were 619.2 °C, 637.3 °C, and 659.0 °C, respectively. A visualization of the flame formed inside the furnace during the execution of the experiment can be seen in Figure 7.

The observations in Figure 7 show that the higher the fan rotation speed, the more even and higher the flame produced in the furnace. This also has an impact on the temperature distribution, which becomes more uniform and has a higher value compared to the low fan rotation speed condition. The results of the numerical study reinforce these findings, by providing data on the temperature distribution inside the furnace at varying fan rotational speeds. A speed of 1.36 m/s produces a temperature of 621.5 °C, a speed of 1.45 m/s produces 642.5 °C, and a speed of 1.53 m/s produces 670.9 °C. The correspondence between the results of numerical analysis and experimental data can be seen in Figure 8. The comparison between the experimental and simulated temperature distributions was analyzed to determine the error rate. The percentage errors are summarized in Table 3. The values obtained from the simulation of temperature, pressure, and velocity distributions in the furnace can be seen in Figures 9, 10, and 11.

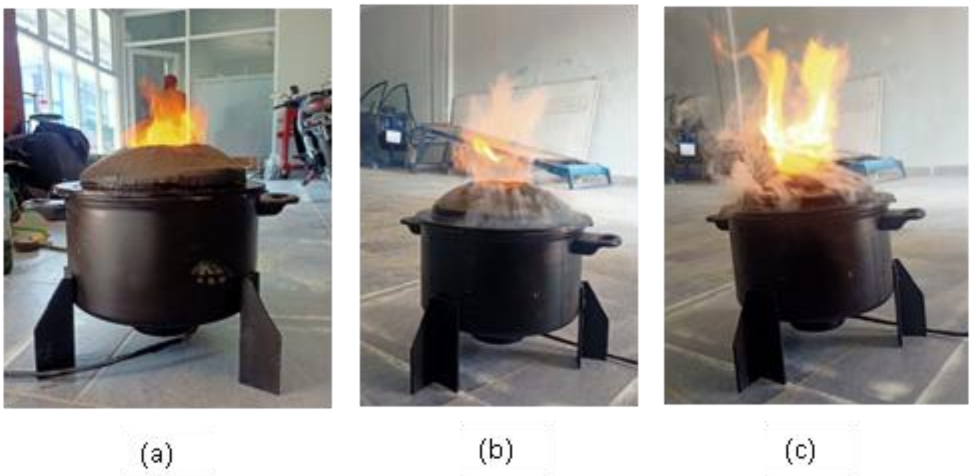

Figure VII. The following figure displays the shape of the flame formed inside the furnace at several different air spray speed in the stove will be varied at 1.36 m/s, 1.45 m/s, and 1.53 m/s

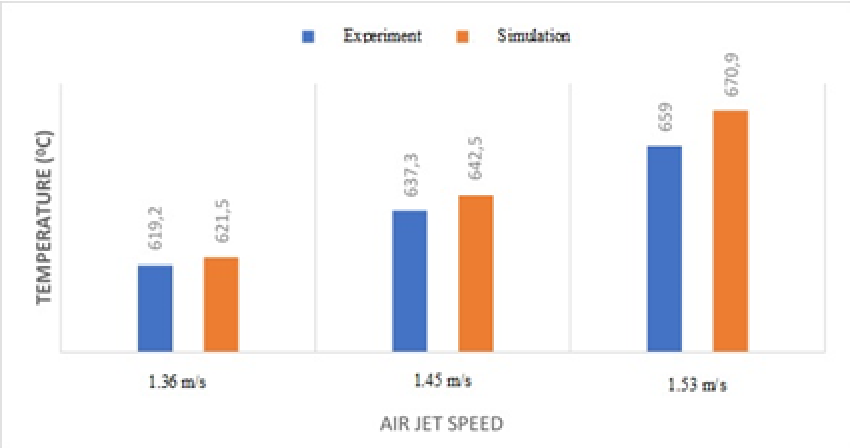

Figure VIII. Temperature comparison results were obtained from each different air spray speed.

TABLE III. PERCENTAGE ERROR OF EXPERIMENTAL AND NUMERICAL TEMPERATURE DISTRIBUTION

Fan Rotation Speed (m/s)	Testing	Time	Experiment temperatures (°C)	Numerical temperatures (°C)	Error percentage (%)
1.36	1	5 Minute	619,4	621.5	0,37 %
	2		617,3		
	3		621,1		
	Average		**619,2**		
1.45	1		638,4	642.5	0,81 %
	2		635,7		
	3		637,8		
	Average		**637,3**		
1.53	1		658,6	670.9	1,80 %
	2		657,5		
	3		661,0		
	Average		**659,0**		

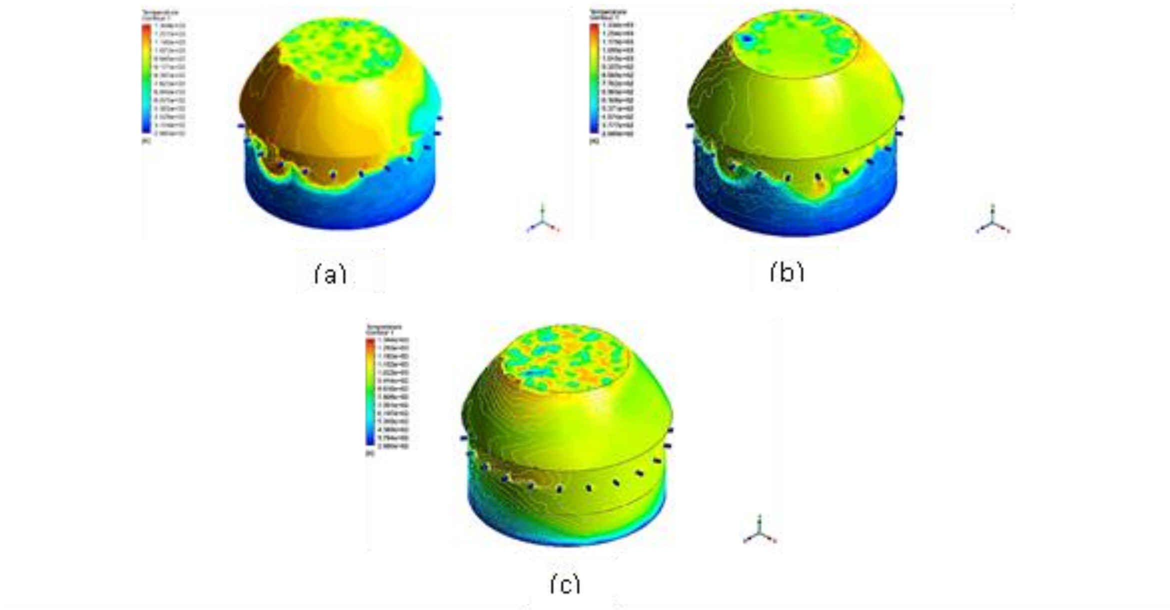

Figure IX. Change temperature distribution at various air spray speeds (a) 1.36 m/s, (b) 1.45 m/s, and (c)

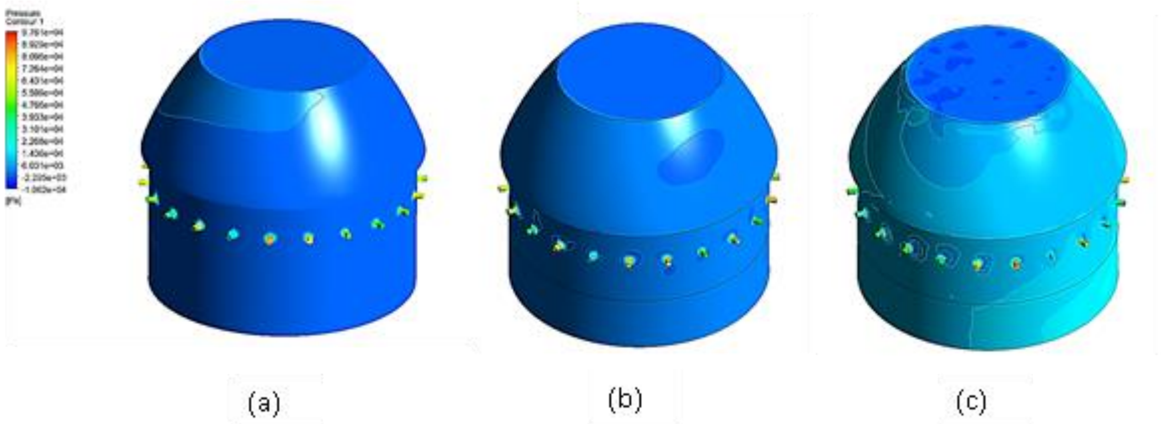

Figure X. Changes in pressure distribution due to variations in air spray speed (a) 1.36 m/s, (b) 1.45 m/s, and (c) 1.53 m/s

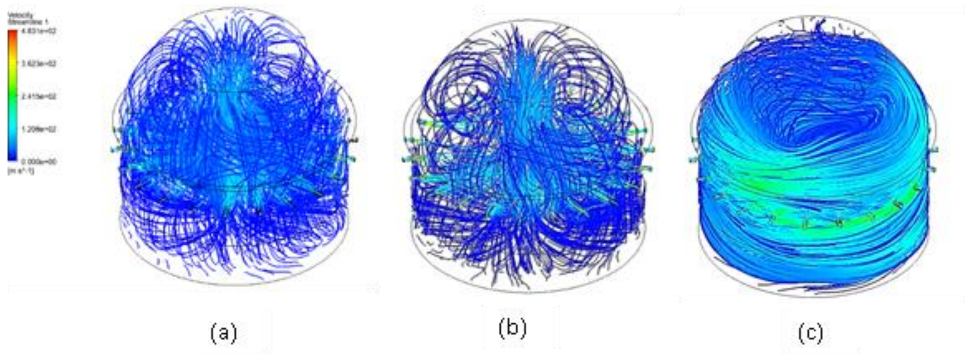

Figure XI. Analysis of fluid velocity distribution inside the flame furnace at various air spray speeds (a) 1.36 m/s, (b) 1.45 m/s, and (c) 1.53 m/s

IV. CONCLUSIONS

From the results of research using experimental and numerical methods on fire furnaces, the following conclusions were obtained:

a. The furnace plays a crucial role in the combustion process.

b. The design of the furnace and the rotation speed of the fan located at its bottom influence the distribution of temperature, pressure, and velocity inside the furnace.

c. Experimental testing at air spray speed of 1.36 m/s recorded temperatures of 619.2 °C, at 1.45 m/s of 637.3 °C, and at 1.53 m/s of 659.0 °C. In comparison, numerical research modelling the temperature distribution inside the furnace have results of 621.5 °C for a speed of 1.36 m/s, 642.5 °C for 1.45 m/s, and 670.9 °C for 1.53 m/s.

d. The percentage error for the temperature value was obtained by comparing the experimental results and numerical simulations. This error is calculated at each fan's rotational speed: 0.37% at 1.36 m/s, 0.81% at 1.45 m/s, and 1.80% at 1.53 m/s.

e. Based on the research results, numerical modeling using Computational Fluid Dynamics (CFD) can be utilized to analyze the distribution of temperature, pressure, and velocity of fluid flow in a furnace.

f. This research can be the benchmark for developing furnaces in the glass and metal melting industries, ensuring efficiency through effective heating and even temperature distribution.

A suggestion for future research is to continue by modifying the furnace geometry, such as testing different fuel types or airflow dynamics.

REFERENCES

Paper used at least 15 relevant references (80% from up-to-date primary sources derived from reputable international journal papers, accredited national journal papers). Reference style using IEEE style.

[1] [1] A. Pratama, Y. W. Atmojo, and G. W. Ramadhan, "Rancang Bangun Kompor (Burner) Berbahan Bakar Oli Bekas," vol. 19, no. September, pp. 95–103, 2020.

[2] [2] I. S. Mulyana, F. T. Industri, and U. Gunadarma, "Analisis aliran udara pada pipa kompor burner," vol. 3, no. 2, pp. 66–76, 2024.

[3] [3] A. A. Jalil, "Design of a Burner Stove Fueled by Used Oil and Used Cooking Oil [Perancangan Kompor Burner Berbahan Bakar Oli Bekas Dan Minyak Jelantah]," pp. 1–13.

[4] [4] M. Demuth, C. Gaber, H. Gerhardter, and C. Hochenauer, "Energy Conversion and Management : X CFD simulation aided glass quality and energy efficiency analysis of an oxy-fuel glass melting furnace with electric boosting," vol. 15, no. June, 2022, doi: 10.1016/j.ecmx.2022.100252.

[5] [5] D. Liu, J. Yan, F. Wang, Q. Huang, Y. Chi, and K. Cen, "Experimental reconstructions of flame temperature distributions in laboratory-scale and large-scale pulverized-coal fired furnaces by inverse radiation analysis," Fuel, vol. 93, pp. 397–403, 2012, doi: 10.1016/j.fuel.2011.09.004.

[6] [6] O. Karabaş, Ö. Kaplan, K. S. Yiğit, and M. Gür, "Numerical Investigation of Temperature Distribution in a Fire Resistance Test Furnace," vol. 2016, no. November, 2016.

[7] [7] K. S. Shanmukharadhya, "Numerical and Experimental Investigations for the Temperature Profiles of a Biomass," pp. 15885–15891, 2017, doi: 10.15680/IJIRSET.2016.0608057.

[8] [8] N. Rezazadeh, H. Hosseinzadeh, and B. Wu, "The study of heat transfers in heat treatment furnaces in steel industry," IOP Conf. Ser. Earth Environ. Sci., vol. 163, no. 1, pp. 0–10, 2018, doi: 10.1088/1755-1315/163/1/012108.

[9] [9] F. Liao, Z. Yan, C. Li, Y. Tao, and S. Zhu, "Thermal deviation mechanisms for coupled heat transfer between the combustion side and the furnace tube side in the tubular heating furnace," Appl. Therm. Eng., vol. 257, no. PC, p. 124436, 2024, doi: 10.1016/j.applthermaleng.2024.124436.

[10] [10] A. V. Field et al., "Navier-Stokes Equations," pp. 1–20.

.