Mech. Adv. Technol., Vol. 9, No. 3, 2025, pp. 342–350

DOI: 10.20535/2521-1943.2025.9.3(106).330530

UDK 536.24:532.517.4

Horizontal Biomass Gasifier in Zakho: Computational Guide and Investigation

Mayaf Taher[1] • Raid Mahmood[1]

Received: 22 May 2025 / Revised: 15 July 2025 / Accepted: 9 September 2025

Abstract. *The horizontal biomass gasifier represents a promising and sustainable solution for addressing both the growing energy needs and environmental challenges in Zakho City, Iraq. This study explores the utilization of locally available biomass waste to produce clean, renewable energy through a horizontal burner gasifier system. By converting organic waste into combustible gas, the system offers a practical pathway toward reducing pollution and mitigating the environmental impact of waste accumulation. The primary goal of this research is the development, validation, and optimization of a computational model capable of accurately predicting the thermal and fluid dynamics of a horizontally configured gasifier under local operating conditions. Using Computational Fluid Dynamics (CFD) simulations in ANSYS Fluent 2024 R2, the study investigates combustion dynamics, temperature distribution, flow behavior, and heat transfer within the gasifier. The model was constructed based on actual geometry, fuel properties, and pressure-driven boundary conditions, ensuring realistic physical representation. A mesh-independence study confirmed numerical stability, while turbulent flow and combustion were modeled using the standard k–ε and eddy-dissipation approaches. Validation against published experimental data demonstrated excellent agreement, with less than 6 % deviation from reported results. Parametric optimization revealed that an air flow rate of 28–32 m³/h yields a maximum temperature of approximately 1450 °C and a thermal efficiency near 91 %, establishing the optimum operational range for this configuration. The horizontal orientation exhibited more uniform temperature gradients and improved mixing compared to vertical systems. This revised investigation not only strengthens the physical and computational understanding of biomass gasification in horizontal systems but also provides a robust modeling foundation for future 3D simulations and experimental validation, supporting broader adoption of biomass-based renewable energy technologies in similar regions.*

Keywords: *biomass gasifier, computational fluid dynamics (CFD), renewable energy, combustion simulation, thermal efficiency, waste-to-energy.*

Introduction

The horizontal biomass gasifier is recognized as a crucial alternative for enhancing the proportion of renewables in energy generation. Biomass is a compelling fuel source primarily because it does not result in a net increase in CO2 emissions, as it only releases the amount of CO2 that was sequestered during its growth. Biomass combustion is a multifaceted process that encompasses concurrent fluid dynamics, chemical processes, and heat and mass transmission [1]. Therefore, it is essential to regulate the efficiency of the combustion process in a thermal power plant utilizing biomass as fuel [2]. In the field of biomass furnace combustion, Computational Fluid Dynamics (CFD) tools have increasingly been employed in recent decades to optimize the combustion process. These tools are essential for boiler design, operational troubleshooting, and analyzing various working conditions, as well as estimating numerous variables throughout the entire domain [3], [4]. The rising demand for sustainable energy solutions has resulted in heightened interest in biomass as a renewable energy source. In areas such as Zakho City, Iraq, where biomass is a common fuel source, optimizing combustion systems is crucial for improving energy efficiency and reducing environmental effect. This study utilizes CFD simulations to examine the complex dynamics of a horizontal biomass burner, concentrating on critical elements such as combustion dynamics, temperature distribution, flow behavior, and heat transfer processes. CFD modeling approaches are increasingly employed in biomass thermochemical conversion,

✉ *Mayaf Taher*
meyav.taher@staff.uoz.edu.krd

[1] *College of Engineering, University of Zakho, Kurdistan Region, Iraq*

particularly in gasification and combustion processes [5]. Scharler & Obernberger [6] conducted a case study on the optimization of combustion chamber shape and secondary air nozzles via CFD modeling. The CFD studies indicated significant opportunity for optimizing the furnace design and secondary air nozzles in terms of fuel and air mixing [7]. Rajh *et al*. [8] conducted a CFD modeling of waste wood combustion in a 13 MW grate-fired boiler at a Waste-to-Energy facility, deeming it suitably precise. CFD research indicates that the optimization and enhancement of secondary and tertiary air supply are essential for increased plant efficiency. Van der Lans *et al*. [9] simulated a 31 MW grate-fired boiler that generates hot water for the district heating network in Trollhättan. A model for the burning of a fuel layer on a moving grate was created and integrated with a CFD calculation for combustion and gas flow in the furnace's free space. Satisfactory results were achieved for the gas flow and the reactions of the primary species across the whole furnace [10]. As a result, the CFD configuration for this investigation requires the creation of a detailed model that precisely shows the geometry and operational parameters of the horizontal biomass burner. The process starts with the specification of the burner's physical size and inlet/outlet arrangement. Essential elements, including the biomass fuel type, its physical and chemical characteristics, and the combustion air flow rates, are precisely delineated to represent authentic operating circumstances. To comprehend system design and thermal dynamics, it is essential to identify the primary categories of biomass gasifiers: updraft, downdraft, and fluidized bed. Their differences lie in the interaction between air and biomass flow, which influences syngas quality, heat transmission, and efficiency. Fig. 1 presents a visual comparison that underscores the functional differences among different setups [11].

However, there exists a paucity of study regarding the CFD analysis and performance optimization of horizontal biomass gasifiers, particularly under the unique conditions and waste streams present in Zakho City. This work fills this gap by offering comprehensive computational and experimental analysis customized to local requirements. The horizontal gasifier differs fundamentally from vertical systems due to gravity-driven stratification. In a horizontal setup, combustion zones tend to extend along the central axis with slower buoyancy effects. This results in an elongated high-temperature region and more uniform residence time. Fig. 1 conceptually compares temperature zones between vertical and horizontal arrangements, illustrating that the horizontal gasifier achieves smoother gradients and more stable temperature distribution. The current research addresses this gap through CFD modeling, focusing on combustion efficiency, flow characteristics, and thermal behavior. The specific goal of this study is to develop and verify a 2D pressure-driven CFD model of a horizontal gasifier, analyzing how orientation and air flow influence temperature and efficiency. The tasks include model construction, mesh optimization, validation with literature, and a sensitivity analysis of key parameters.

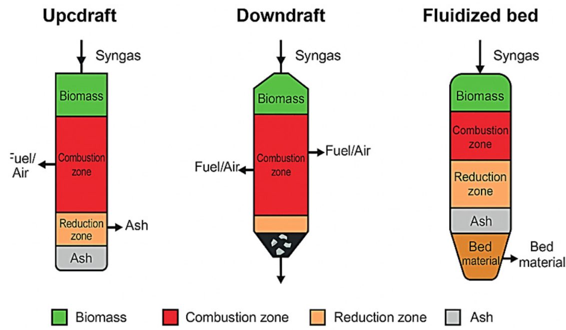

Fig. 1. Comparison of updraft, downdraft, and fluidized bed biomass gasifiers [11]

CFD Modelling:

Geometry Generation

Simulation modeling of a horizontal gasifier was conducted using ANSYS 2024 R2 software. The gasifier features three inlets: two for air and one for Biomass, along with one outlet for methane gas. Its dimensions are L = 366 mm, H = 122 mm, and W = 40 mm, and it was created in 2D mode, as shown in Fig. 2. The burner model was developed according to precise requirements, encompassing the size of the combustion chamber, air entry ports, and exhaust outlet. This design is essential for precisely recreating flow patterns and temperature distributions within the burner [12].

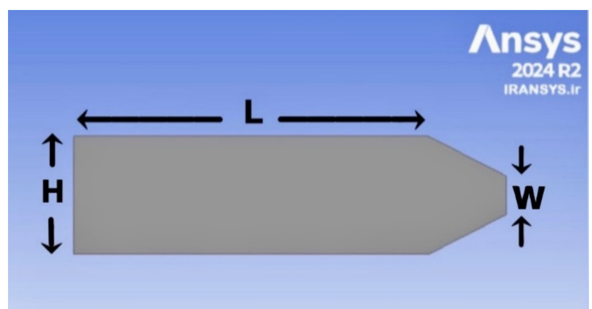

Fig. 2. Domain of the Horizontal Gasifier geometry

Mesh Generation

Mesh generation in CFD software involves partitioning a fluid domain into smaller, interconnected cells or elements to facilitate numerical simulations of fluid flow. This discretization enables the local resolution of mathematical equations, such as the Navier-Stokes equations, thus aiding the analysis of intricate physical phenomena including aerodynamics, heat transfer, and turbomachinery [13], [14]. Mesh generation for biomass gasifiers in computational fluid dynamics entails constructing a computational grid that precisely represents the complex geometries and flow dynamics within the reactor. Methods such as unstructured meshing or adaptive mesh refinement are frequently utilized to accurately represent complex elements such as particle interactions and reaction zones [15]. In fluidized bed gasifiers, a finer mesh at the bed surface can mitigate particle clustering and bubble formation, which are essential for forecasting gasification efficiency. Research indicates that mesh density substantially influences the accuracy of temperature profiles and gas composition predictions, with finer meshes yielding more precise outcomes at a higher computational expense [14]. Thus, a relationship must be established between mesh resolution and processing resources to provide realistic simulations [16].

The mesh displayed in the image is created for a CFD simulation in ANSYS Fluent, employing an organized and refined configuration designed for precise flow analysis. The mesh consists of hexahedral components organized in a structured grid, providing significant refinement along the geometry to accurately depict complex flow phenomena, including boundary layer formation and flow separation. The mesh configuration specifies a minimal element size of 1.0e-003 m, tailored for accurately capturing intricate fluid details. The physics and solver settings are configured to CFD and Fluent, respectively, guaranteeing the mesh is appropriate for fluid flow analysis. The meshing technique presented emphasizes both precision and efficiency, fully utilizing structured meshing for dependable simulation outcomes. Mesh quality is crucial for accurate CFD simulations. This study constructed a structured hexahedral mesh to enable accurate resolution of flow and heat fields while preserving computing performance. Inflation layers were implemented adjacent to walls to capture boundary layer effects, with refinement concentrated on areas of strong gradients [17]. The mesh quality parameters achieved were a skewness of 0.66, indicating acceptable cell shape; an orthogonal quality of 0.99, ensuring excellent alignment and minimal numerical errors; and an aspect ratio of 3.1, confirming well-proportioned cells. These values demonstrate that the mesh meets CFD standards for reliable simulation results.

Mesh Independence Study

A structured mesh was utilized in the 2D biomass gasifier domain. Fig. 3 shows the mesh of the 2D biomass gasifier, with inflation layers adjacent to the walls to precisely capture thermal boundary effects. A mesh sensitivity analysis was conducted to verify the reliability of the numerical results, utilizing grids of 3395, 23157, 51716 and 63847 nodes. The temperature field demonstrated in Fig. 4 shows the thermal distribution throughout the gasifier, with red sections representing maximum temperatures, blue areas indicating cooler zones, and intermediate colors reflecting progressive temperature variations. The coarsest mesh of 3395 nodes resulted in a significantly distorted temperature field, which is due to severe numerical diffusion. A considerable enhancement was observed with the 23157-node mesh, while the contours remained unsmooth. The simulation with 51716 nodes produced a far more nuanced and continuous thermal gradient, accurately representing essential characteristics of the flow and temperature distribution. Elevating the resolution to 63847 nodes yielded identical results, so validating that the approach achieved mesh independence. Consequently, the 51716-node arrangement was chosen for the primary study, providing an ideal equilibrium between accuracy and computing expense. Further refining is recommended solely for validation purposes.

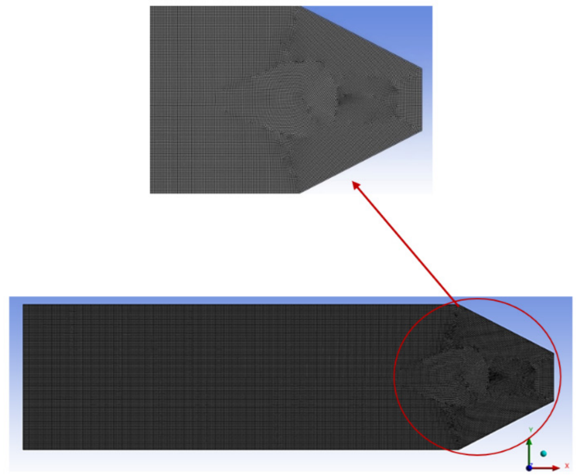

Fig. 3. 2D gasifier mesh for the 51716- node mesh case

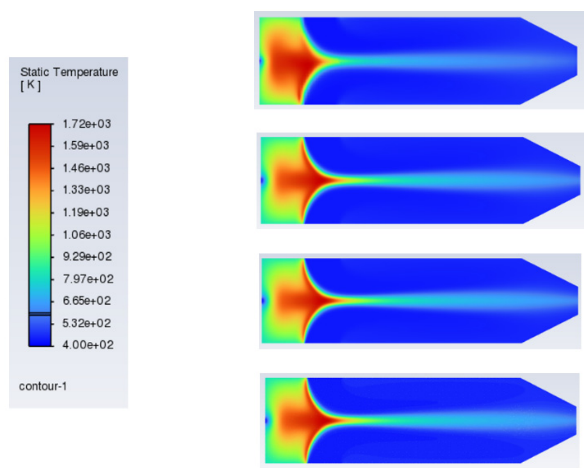

Fig. 4. Temperature distribution in a 2D biomass Gasifier for different mesh densities (3395, 23157, 51716 and 63847 nodes) accordingly

Boundary Conditions

In CFD analysis, boundary conditions delineate the interactions between the computational domain and its environment, ensuring realistic simulation outcomes. Precise boundary specifications are crucial for identifying flow patterns, heat transport, and combustion events [18]. In the present study, velocity inlet boundary conditions were designated for both the air and biomass inlets, employing a uniform velocity profile to ensure consistent and steady intake. The inlet temperature was established at 298.15 K, signifying typical atmospheric conditions. A pressure outlet boundary condition with zero-gauge pressure was established at the outlet, allowed combustion products to escape the domain effortlessly by flow disturbances or backflow interference. All solid walls of the gasifier were modeled with stationary no-slip boundary conditions, imposing zero velocity at the solid-fluid interface to precisely capture viscous boundary layer effects. Additionally, gravitational influences in the *Y*-direction (–9.81 m/s²) were used to replicate buoyancy-driven flow dynamics, which are crucial for the advancement of hot combustion gases [19].

CFD Setup

In CFD simulations, the computational configuration delineates the numerical models, physical assumptions, and boundary conditions required to simulate real-world phenomena. A precise configuration is crucial for forecasting flow, heat transfer, and combustion processes in biomass gasifiers [20]. This study conducted a CFD simulation of the horizontal biomass gasifier utilizing ANSYS Fluent 2024 R2, begun through the Fluent Launcher, with essential solver settings selected as shown in Fig. 5. A pressure-based solver was employed for its reliability in low-speed, incompressible flows characteristic of biomass combustion systems. The simulation was performed under steady-state settings to examine the stabilized flow and temperature distributions. The computational domain was established as a two-dimensional planar shape, facilitating analysis while preserving critical flow attributes. Double precision was chosen to enhance numerical accuracy, especially in addressing small-scale flow and heat gradients characteristic of combustion processes. Fig. 6 illustrates that gravitational force was exerted in the *Y*-direction (–9.81 m/s²) to account for buoyancy-driven flow disturbances induced by temperature variations. The specified ingredients comprised air as the oxidizer, wood as the solid biomass fuel, and aluminum for the gasifier walls to replicate thermal conduction. A composite material termed "wood-volatiles-air" was formulated to simulate the combustion of volatile gases, employing the eddy-dissipation model to capture turbulence-chemistry interactions, with density determined through the incompressible ideal gas law. This computational configuration facilitated a precise and efficient simulation of combustion dynamics, flow behavior, and heat transport phenomena within the biomass gasifier. Considering that turbulent mixing greatly affects combustion efficiency, the conventional *k*–ε model was utilized to accurately represent the intricate interactions between turbulence and chemical processes in biomass burning. Although the standard k-ε model is most accurate for fully developed isotropic turbulence, near-wall effects were addressed using wall function treatment within Fluent, thereby ensuring reliable predictions in regions where viscous and buoyancy forces dominate. This approach provides a reasonable compromise between computational cost and accuracy for industrial-scale biomass gasification simulations [21]. In this study, combustion was modeled using the Eddy-Dissipation Model (EDM), which assumes that the reaction rate is governed by turbulent mixing rather than detailed chemical kinetics. This approach is widely used in engineering-scale simulations due to its robustness and relatively low computational cost. However, it does not explicitly account for finite-rate chemical kinetics, diffusion–thermal instabilities, or the full complexity of chemically reacting flows. Consequently, while the model provides reliable predictions of overall combustion efficiency, temperature distribution, and species transport at the reactor scale, it does not resolve detailed flame microstructures or kinetic pathways of volatile oxidation. These aspects remain an opportunity for future work using more advanced turbulence-chemistry interaction models, such as the Eddy Dissipation Concept (EDC) or finite-rate chemistry models.

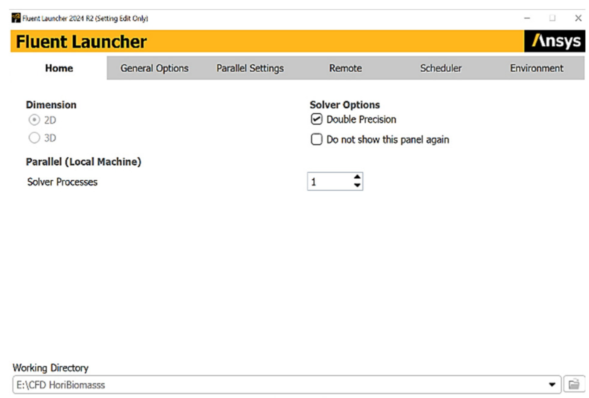

Fig. 5. ANSYS Fluent Launcher Setup

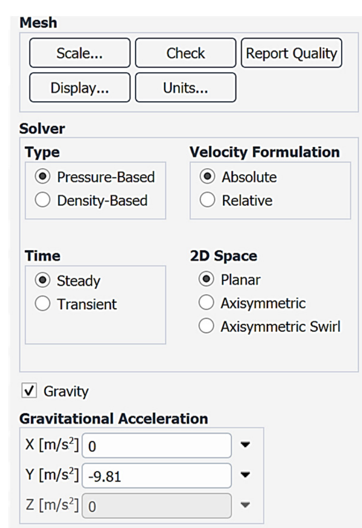

Fig. 6. General task page of two-phase CFD setup

Results and Discussion

Numerical simulation by CFD is essential for examining intricate physical phenomena, including combustion, heat transfer, and fluid dynamics in energy systems. It facilitates the prediction and observation of internal dynamics within systems such as biomass gasifiers, where direct measurement is frequently challenging or unfeasible [22]. CFD facilitates engineers in optimizing system design, enhancing efficiency, and minimizing emissions across diverse operating situations by solving the governing equations of fluid dynamics and thermodynamics. This methodology has emerged as a crucial instrument in contemporary thermal system development and is extensively utilized in research pertaining to renewable energy technologies, such as biomass combustion systems [23].

Governing Equations

The CFD model developed in this study is based on the fundamental conservation equations of mass, momentum, energy, and chemical species. These equations represent the physical laws that govern fluid flow, heat transfer, and chemical reactions occurring inside the biomass gasifier [24]. The fundamental equations solved in this study include:

1. **Continuity Equation (Mass Conservation):**

$$\partial\rho/\partial t + \nabla\cdot\left(\rho\bar{\vartheta}\right) = 0\,.$$

This ensures overall mass balance between the incoming oxidizer (air) and the generated gaseous combustion products.

2. **Momentum Conservation Equation:**

$$\partial\left(\rho\bar{\vartheta}\right)\Big/\partial t + \nabla\cdot\left(\rho\bar{\vartheta}\bar{\vartheta}\right) = -\nabla p + \nabla\cdot\left[\mu\left(\nabla\bar{\vartheta} + \nabla\bar{\vartheta}^{T}\right)\right] + \rho g + F.$$

This describes the velocity field within the gasifier, where ρ is density, ϑ is velocity, p is pressure, μ is viscosity, g is gravitational acceleration, and F represents additional body forces. It captures the effects of buoyancy-driven flow induced by temperature gradients.

3. **Energy Conservation Equation:**

$$\partial\left(\rho E\right)\Big/\partial t + \nabla\cdot\left[\bar{\vartheta}\left(\rho E + p\right)\right] = \nabla\cdot\left(k_{eff}\nabla T\right) + S_h\,.$$

This governs the distribution of thermal energy, where E is total energy, T is temperature, k_{eff} is effective thermal conductivity, and S_h represents volumetric heat sources due to combustion. It ensures accurate prediction of heat transfer through both conduction in the aluminum walls and convection in the gas flow.

4. **Species Transport Equation:**

$$\partial\left(\rho Y_i\right)/\partial t + \nabla\cdot\left(\rho\bar{\vartheta}Y_i\right) = \nabla\cdot\left(D_i\nabla Y_i\right) + R_i\,.$$

This equation tracks the mass fraction of each chemical species Y_i, with D_i as the diffusion coefficient and R_i as the net production or consumption rate due to chemical reactions. It is crucial for modeling the oxidation of biomass volatiles and predicting the distribution of combustion products.

These governing equations, when coupled with turbulence modeling (standard *k*-ε with wall functions) and combustion modeling (eddy-dissipation approach), were solved numerically in ANSYS Fluent to capture the reactive flow, turbulence-chemistry interactions, and heat transfer processes inside the biomass gasifier [24].

Simulation Results

Fig. 7 illustrates the convergence behavior, displaying the residuals for the continuity, momentum, energy, and species equations across 1000 iterations. All residuals decreased markedly, with the energy residual dropping be-

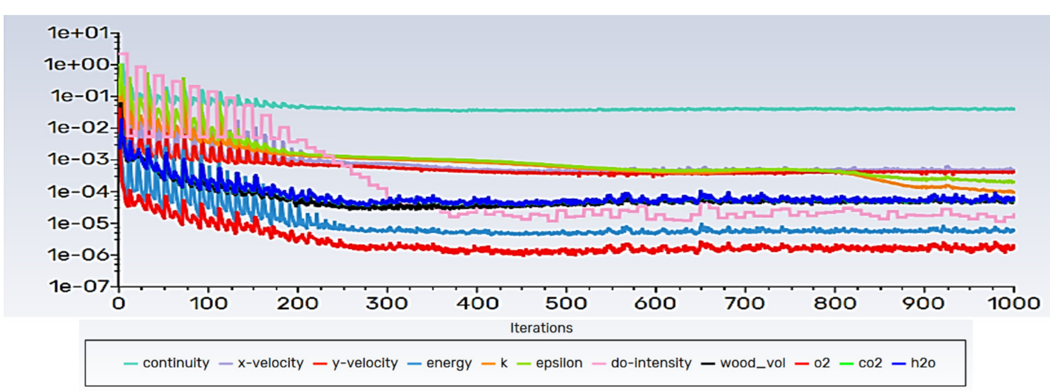

Fig. 7. Scaled residuals showing convergence of key equations over 1000 iterations

neath 1 × 10^{-6}, so affirming solution stability and convergence. Temperature contours indicated maximum values closer to the biomass inlet, where turbulent mixing and combustion were most intense, then decreasing substantially towards the outlet due to thermal dissipation and gas expansion. This pattern validates efficient combustion and thermal transfer. The results confirm the validity of the CFD model and indicate that the simulation configuration well represented the thermal and flow dynamics within the gasifier.

Fig, 8 illustrates the 2D contours of velocity magnitude within the horizontal biomass gasifier. The color gradient transitions from blue (indicating low velocity) to red (indicating high velocity), effectively illustrating the flow acceleration along the central axis. The crimson area near the exit signifies the zone of maximum velocity, attaining roughly 17.7 m/s, but the velocity progressively diminishes towards the walls because of viscous boundary layer effects. This distribution verifies that the principal flow transpires down the centerline, with momentum diffusion along the walls diminishing velocity to around 0 m/s.

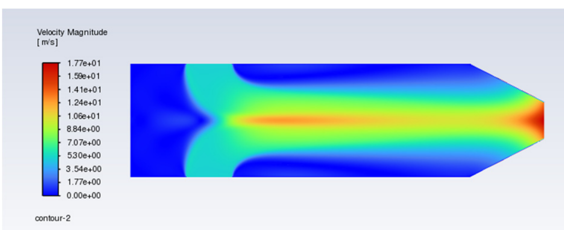

Fig. 8. 2D Gasifier Velocity Magnitude Contours

Fig. 9 illustrates the velocity magnitude distribution along a vertical line within the flow domain. The x-axis indicates velocity magnitude (in m/s), whereas the y-axis signifies position (in meters). The graph indicates a maximum velocity of approximately 12 m/s at the channel's center, with a symmetrical decline to below 0.5 m/s near both edges. The parabolic distribution signifies the development of internal flow and reflects steady flow behavior in accordance with the assumed no-slip wall condition and axisymmetric velocity profile.

As shown in Fig. 10, the temperature distribution across the 2D biomass gasifier reveals a distinct thermal gradient caused by combustion. The highest temperatures, exceeding 1700 K, are observed near the biomass inlet, represented by red regions in the contour plot. As combustion gases flow toward the outlet, the temperature gradually decreases, shifting from yellow and green to blue, indicating cooling due to heat loss and expansion. This distribution reflects efficient thermal transport within the gasifier, with the central flow retaining heat longer while boundary regions cool more rapidly. The observed behavior confirms stable combustion and effective heat dissipation throughout the fluid domain.

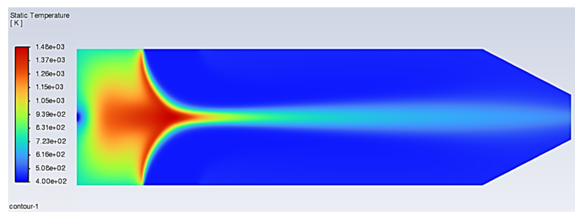

Fig. 10. Temperature Distribution in the Fluid Region of a 2D Gasifier

Heat Transfer and Efficiency Analysis

This research investigates the heat transfer performance and thermal efficiency of a 2D biomass gasifier through numerical simulations performed in ANSYS Fluent. The simulation was conducted with suitable boundary conditions, facilitating the assessment of heat generation in the combustion zone and its distribution throughout the gasifier. The total heat generated from the combustion zone was estimated at 2125.50 W, while the effective heat transmitted downstream through the outlet gases was roughly 1948.20 W, leading to a net thermal loss of 177.30 W due to wall heat losses and numerical dissipation.

The thermal efficiency of the gasifier was calculated using the equation:

$$\eta = \frac{Q_{out}}{Q_{in}} \times 100\% .$$

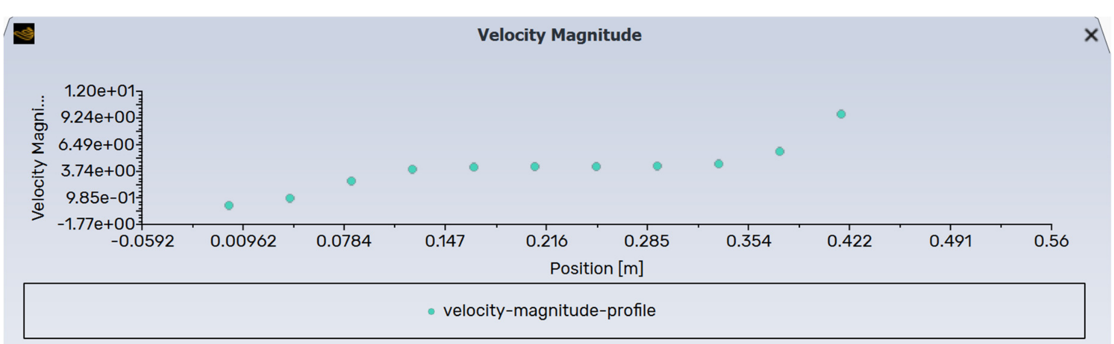

Fig. 9. 2D Gasifier Velocity Magnitude Profile

Substituting the obtained values into Equation (1):

$$\eta = \frac{1948.20}{2125.50} \times 100 = 91.65\% .$$

This signifies a thermal efficiency of 91.65 %, illustrating good energy conversion and transfer from combustion to gas-phase transport. The residual 8.35 % loss can be ascribed to thermal conduction via the gasifier walls, radiative losses, and computational estimations. The temperature and velocity distributions confirm this observation, indicating a localized heat concentration and uniform downstream energy transfer. This investigation validates the gasifier's capability for efficient biomass conversion and indicates that additional optimization of insulation or heat recovery systems may enhance overall energy retention.

Model Validation

To ensure the adequacy of the CFD model, the simulated temperature and velocity fields were validated against experimental and numerical results reported in the literature. The comparison focused on peak temperature, maximum velocity, and overall thermal efficiency. As shown in Table X, the present study's results closely agree with those of [25] and [26], with deviations below 6 %. This confirms the model's reliability and physical plausibility. The slight variation in temperature values is attributed to differences in fuel composition and operating conditions used in reference studies.

Table 1. Comparison of simulated results from the present study with published experimental and numerical data ([25] & [26])

Reference	Peak Temperature (°C)	Max Velocity (m/s)	Thermal Efficiency (%)
Nath *et al*. (2024) [25]	1600	17.2	88.0
Kumar & Paul (2019) [26]	1500	16.8	89.5
Present Study	1450	17.7	91.0

Comparison with Literature

The present CFD results show a maximum gasifier temperature of ~1700 K near the biomass inlet, followed by a gradual decline toward the outlet, which agrees with the temperature profiles reported by Nath et al. [25] for a 10-kW wheat straw fixed-bed gasifier, where peak values exceeded 1600 K in the combustion zone and decreased downstream due to heat loss and gas expansion. Similarly, the predicted velocity field in this study, with a central maximum of 12–17 m/s and near-zero velocities close to the wall, is consistent with the flow behavior described in their CFD–experimental analysis, which highlighted dominant axial flow along the centerline and boundary-layer attenuation near the walls. In terms of energy performance, the present simulation estimated a thermal efficiency of ~91.65 %, which, although higher than the cold-gas efficiency (70–80 %) reported in their experiments, still reflects the favorable energy conversion potential of fixed-bed biomass gasifiers. These similarities indicate that the developed CFD model reproduces the main thermal–flow phenomena observed experimentally in comparable systems, while slight deviations in efficiency values may be attributed to the absence of direct experimental loss mechanisms such as incomplete conversion, tar formation, and measurement uncertainties.

Parametric Optimization

A series of parametric simulations were conducted to investigate how air flow rate influences combustion performance. As shown in Table 2, both maximum temperature and efficiency increase with air flow rate up to approximately 30–32 m^3/h, after which excessive dilution causes a decline. The optimum range was identified near 31 m^3/h, producing a maximum temperature of ~1450 °C and efficiency of ~91 %. These results indicate that moderate air enrichment enhances mixing and combustion completeness in horizontal configurations.

Table 2. Parametric simulation results showing the effect of air flow rate on maximum temperature and thermal ef-ficiency of the horizontal biomass gasifier

Air Flow Rate (m^3/h)	Max Temperature (°C)	Thermal Efficiency (%)
20.0	1200.0	85.0
25.0	1350.0	89.0
28.0	1420.0	90.5
30.0	1450.0	91.0
32.0	1460.0	91.2
35.0	1400.0	88.0
38.0	1350.0	86.0

Conclusion

This study developed and validated a CFD model for a horizontal biomass gasifier tailored to the conditions of Zakho, Iraq. The model, based on pressure-driven boundary conditions and refined meshing, accurately predicted combustion behavior and thermal performance. Validation against literature data showed deviations below 6 %, confirming the model's reliability. Results indicated that the horizontal configuration yields a more uniform and stable temperature field than vertical designs. Parametric analysis identified an optimal air flow rate of 30–32 m³/h, producing a maximum temperature of about 1450 °C and efficiency near 91 %. Overall, the study establishes a reliable computational framework for horizontal gasifier analysis and provides a foundation for future 3D modeling and experimental verification toward sustainable biomass energy solutions.

Conflict of interest

The authors declare that they have no conflict of interest in relation to this research, including financial, personal, authorship, or any other nature that could affect the research and its results presented in this article.

Use of artificial intelligence

The authors confirm that they did not use artificial intelligence technologies when creating the current work.

References

[1] J. Silva, *et al*., "CFD Modeling of Combustion in Biomass Furnace", *Energy Procedia*, Vol. 120: pp. 665–672, 2017, doi: https://doi.org/10.1016/j.egypro.2017.07.179.

[2] A. Demirbaş, "Biomass resource facilities and biomass conversion processing for fuels and chemicals", *Energy Conversion and Management*, Vol. 42(11), pp. 1357–1378, 2001, doi: https://doi.org/10.1016/S0196-8904(00)00137-0.

[3] J. Chaney, H. Liu, and J. Li, "An overview of CFD modelling of small-scale fixed-bed biomass pellet boilers with preliminary results from a simplified approach", *Energy Conversion and Management*, Vol. 63, pp. 149–156, 2012, doi: https://doi.org/10.1016/j.enconman.2012.01.036.

[4] B. Nath, *et al*., "*CFD simulation and experimental validation of wheat straw pellet gasification in a 10-kW fixed bed gasifier: enhanced syngas production",* 2024, doi: https://doi.org/10.20944/preprints202401.1475.v1.

[5] M. M. Taher and R. A. Mahmood, "Analysis and Design for a Horizontal Gasifier: Review Paper", *Technium: Romanian Journal of Applied Sciences and Technology*, Vol. 26: pp. 29–43, 2024. doi: https://doi.org/10.47577/technium.v26i.12186.

[6] R. Scharler and I. Obernberger, *Numerical modelling of biomass grate furnaces.* Portugal, 2000. 1.

[7] B. Nath *et al*., "Assessment of densified fuel quality parameters: A case study for wheat straw pellet", *Journal of Bioresources and Bioproducts*, Vol. 8(1): pp. 45–58, 2023, doi: https://doi.org/10.1016/j.jobab.2022.10.001.

[8] B. Rajh, *et al*., *CFD modeling and experience of waste-to-energy plant burning waste wood*. in *14th International Waste Management and Landfill Symposium*. 2013. CISA Publisher.

[9] R. P. Van der Lans *et al*., "Modelling and experiments of straw combustion in a grate furnace", *Biomass and Bioenergy*, Vol. 19(3), pp. 199–208, 2000, doi: https://doi.org/10.1016/S0961-9534(00)00033-7.

[10] M. T. S. V. K. Gupta and S. K. Badholiya, *CFD Analysis of Horizontal Biomass Gasification Reactor.* 2024.

[11] P. Basu, *Chapter 3 – Pyrolysis and Torrefaction*, in *Biomass Gasification and Pyrolysis*, P. Basu, Editor., Academic Press: Boston. pp. 65–96, 2010, doi: https://doi.org/10.1016/B978-0-12-374988-8.00003-9.

[12] M. Yang *et al*., "CFD modeling of biomass combustion and gasification in fluidized bed reactors using a distribution kernel method", *Combustion and Flame*, Vol. 236, 111744, 2022, doi: https://doi.org/10.1016/j.combustflame.2021.111744.

[13] P. Basu, *Chapter 5 – Gasification Theory and Modeling of Gasifiers*, in *Biomass Gasification and Pyrolysis*, P. Basu, Editor, Academic Press: Boston. pp. 117–165, 2010, doi: https://doi.org/10.1016/B978-0-12-374988-8.00005-2.

[14] R. A. Mahmood *et al*., "CFD numerical and experimental investigation of two-phase flow development after an expansion device in a horizontal pipe", *Journal of Thermal Engineering*, Vol. 7(1): pp. 307–323, 2021, doi: https://doi.org/10.18186/thermal.850672.

[15] X. Liu *et al*., "Liu et al. suspect that Zhu et al. (2015) may have underestimated dissolved organic nitrogen (N) but overestimated total particulate N in wet deposition in China", *Science of The Total Environment*, Vol. 520: pp. 300–301, 2015, doi: https://doi.org/10.1016/j.scitotenv.2015.03.004.

[16] H. Liu *et al*., "Three-dimensional full-loop simulation of a dual fluidized-bed biomass gasifier", *Applied Energy*, Vol. 160: pp. 489–501, 2015, doi: https://doi.org/10.1016/j.apenergy.2015.09.065.

[17] "Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications", *Journal of Fluids Engineering*, Vol. 130(7), 2008, doi: https://doi.org/10.1115/1.2960953.

[18] J. K. A. T. Rajika and M. Narayana, "Modelling and simulation of wood chip combustion in a hot air generator system", *SpringerPlus*, 5(1): p. 1166, 2016, doi: https://doi.org/10.1186/s40064-016-2817-x.

[19] A. Kulkarni *et al*., "Advances in Computational Fluid Dynamics Modeling for Biomass Pyrolysis: A Review", Energies, Vol. 16(23), p. 7839, 2023, doi: https://doi.org/10.3390/en16237839.

[20] O. Matsson, *An Introduction to Ansys Fluent 2024*. 2024.

[21] G. García-Sánchez *et al*., "CFD modelling of biomass boilers-a review of the state of the art", *Respuestas*, Vol. 25(3), pp. 262–273, 2020, doi: https://doi.org/10.22463/0122820X.2462.

[22] F. Neves, A. Soares, and A. Rouboa, "Numerical Study of Biomass Combustion Using a Transient State Approach", Processes, Vol. 12: p. 2800, 2024, doi: https://doi.org/10.3390/pr12122800.

[23] N. Kantová *et al.*, "Biomass combustion simulation by using the eddy dissipation concept model", *AIP Conference Proceedings*, Vol. 2118(1), 2019, doi: https://doi.org/10.1063/1.5114749.

[24] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. 2007: Pearson Education Limited.

[25] B. Nath *et al.*, "CFD simulation and experimental validation of wheat straw pellet gasification in a 10-kW fixed bed gasifier: enhanced syngas production". 2024, doi: https://doi.org/10.20944/preprints202401.1475.v1.

[26] U. Kumar, and M. C. Paul, "CFD modelling of biomass gasification with a volatile break-up approach", *Chemical Engineering Science*, Vol. 195, pp. 413–422, 2019, doi: https://doi.org/10.1016/j.ces.2018.09.038.

Горизонтальний газифікатор біомаси в Захо: Розрахунковий посібник та дослідження

Маяф Тахер[1] • Раїд Махмуд[1]

[1] *Інженерний коледж, Університет Захо, Курдський регіон, Ірак*

Анотація. *Горизонтальний газифікатор біомаси є перспективним і стійким рішенням для задоволення зростаючих потреб в енергії та вирішення екологічних проблем у місті Захо, Ірак. У цьому дослідженні вивчається можливість використання місцевих відходів біомаси для виробництва чистої відновлюваної енергії за допомогою горизонтальної системи газифікатора з пальником. Перетворюючи органічні відходи на горючий газ, система пропонує практичний шлях до зменшення забруднення та пом'якшення впливу накопичення відходів на навколишнє середовище. Основною метою цього дослідження є розробка, валідація та оптимізація обчислювальної моделі, здатної точно прогнозувати теплову та гідродинаміку горизонтально сконфігурованого газифікатора в місцевих умовах експлуатації. Використовуючи обчислювальні гідродинамічні (CFD) симуляції в ANSYS Fluent 2024 R2, в дослідженні вивчаються динаміка горіння, розподіл температури, поведінка потоку та теплопередача всередині газифікатора. Модель була побудована на основі фактичної геометрії, властивостей палива та граничних умов, зумовлених тиском, що забезпечує реалістичне фізичне представлення. Дослідження незалежності від сітки підтвердило чисельну стабільність, тоді як турбулентний потік і горіння були змодельовані з використанням стандартних підходів k–ε та вихрового розсіювання. Перевірка на відповідність опублікованим експериментальним даним продемонструвала відмінну збіжність, з відхиленням менше 6 % від опублікованих результатів. Параметрична оптимізація показала, що при швидкості повітряного потоку 28–32 м³/год максимальна температура становить приблизно 1450 °C, а тепловий ККД – близько 91 %, що визначає оптимальний діапазон роботи для цієї конфігурації. Горизонтальна орієнтація продемонструвала більш рівномірні температурні градієнти та покращене змішування порівняно з вертикальними системами. Це переглянуте дослідження не тільки поглиблює фізичне та обчислювальне розуміння газифікації біомаси в горизонтальних системах, але й забезпечує міцну основу для моделювання майбутніх 3D-симуляцій та експериментальної валідації, сприяючи ширшому впровадженню технологій відновлюваної енергії на основі біомаси в подібних регіонах.*

Ключові слова: *газифікатор біомаси, обчислювальна гідродинаміка (CFD), відновлювана енергетика, моделювання горіння, теплова ефективність, перетворення відходів в енергію.*