



Parametric Investigation of Coal Combustion in a 150 MWe Tangentially Fired Boiler Using STAR CCM+

Umakant Samantaray¹, Pruthiviraj Nimalipuri^{1*}, Vivek Vitankar², Harish Chandra Das¹, Malay Kumar Pradhan³

¹ Department of Mechanical Engineering, National Institute of Technology Meghalaya, Shillong 793003, India

² FluidDimensions, Pune 411045, India

³ Directorate of Factories and Boilers, Govt of Odisha, Bhubaneswar 751001, India

Corresponding Author Email: pruthivirajnimalipuri@gmail.com

Copyright: ©2026 The authors. This article is published by IETA and is licensed under the CC BY 4.0 license (<http://creativecommons.org/licenses/by/4.0/>).

<https://doi.org/10.18280/ijht.440116>

ABSTRACT

Received: 2 October 2025

Revised: 10 December 2025

Accepted: 19 December 2025

Available online: 28 February 2026

Keywords:

Computational Fluid Dynamics, coal combustion, Eulerian-Lagrangian, optimization, STAR CCM+, tangentially fired boiler

Coal continues to be the primary non-renewable energy resource in India, and Pulverized Coal-Fired Boilers (PCFB) contribute nearly 40% of global electricity generation. Increasing power demand necessitates enhancements in boiler efficiency, reliability, and emissions performance. These improvements can be achieved through systematic modification of operational parameters, such as coal particle size distribution, airflow rates, and fuel properties, as well as hardware configurations like burner tilt and orientation. In this research, a comprehensive and robust Computational Fluid Dynamics (CFD) methodology is developed using Simcenter STAR-CCM+ to analyse combustion behaviour in an industrial-scale tangentially fired boiler (TFB). The modelling framework employs an Eulerian–Lagrangian approach to resolve the transport and interaction of pulverized coal particles with the surrounding gas flow. The Eddy-Breakup model simulates the burning of coal, and the Discrete Ordinates Method (DOM) captures the radiative heat transfer. A grid-independent study and existing experimental data are used to extensively validate the CFD method. The validated methodology is applied to a 150 MWe industrial TFB to assess its predictive capabilities in real-life scenarios. Key performance indicators, such as temperature distribution, gas composition (CO, CO₂), and NO_x emissions, match up well with plant measurements. This shows that the model is accurate and reliable. The results highlight the methodology’s capability to identify flame behaviour, diagnose operational issues, and assess the impact of parametric and design modifications on boiler performance. Overall, the developed CFD framework provides a powerful, adaptable, and validated digital tool for comprehensive optimization, combustion tuning, and troubleshooting in PCFB systems across various industrial configurations.

1. INTRODUCTION

Coal holds considerable importance in meeting the increasing demand for electricity across various regions worldwide, mainly due to its abundant reserves surpassing those of other fossil fuels. However, achieving efficient and environmentally sustainable coal utilization remains a considerable challenge in combustion processes [1]. In recent times, there has been a heightened focus on enhancing the performance of large-scale utility boilers. The objective is to prolong their operational lifespan, enhance thermal efficiency, and shorten the release of pollutants, notably nitrogen oxide (NO_x) emissions. Pulverized coal combustion is the commonly adopted power generation combustion system [2]. Due to the diverse factors and parameters influencing coal as a natural resource, its properties and composition vary significantly. This heterogeneity results in differing combustion behaviors and pollutant emissions for each type of coal. Nowadays, the challenge of using various coal types in the same boiler

presents operational complexities. The significant variability in coal properties demands innovative approaches to decision-making and operational strategies. Furthermore, stricter environmental regulations necessitate emissions reduction while maintaining profitability. However, due to the varied combustion behaviours of different coals, utility companies conduct firing tests in boilers before procurement, which can be expensive and potentially damaging to the boiler if unsuitable coal is used. Hence, the ability to predict the behaviour, performance, and pollutant emissions of coal-fired utility boilers is crucial. A notable advancement was made to leverage the higher volumetric heat release rates of pulverized coal and enhance system efficiencies through the implementation of superheaters, economizers, and combustion air preheaters. Absorbing most of the heat generated by coal combustion, utilising superheaters, re-heaters, economizers, and air pre-heaters helped improve the overall system efficiency. The efficient operation of combustion chambers in boilers relies on understanding oxidation reactions and heat

transfer between combustion products, chamber walls, and heat exchangers. Detailed analysis of these mechanisms is crucial. While numerous combustion modelling methods exist, only a handful can comprehensively address the entire combustion process.

Over the last decade, Computational Fluid Dynamics (CFD) simulations have emerged as a potent tool for controlling and analyzing coal-fired utility boilers. They have been instrumental in optimizing operational conditions for higher efficiency and lower emissions [3-12]. Moreover, CFD has played a significant role in troubleshooting boiler issues within coal-fired power plants. The design of new boilers, particularly their combustion systems and how different coal types affect combustion performance, has driven both academic and industrial communities to utilize CFD extensively [9, 13-20]. CFD serves as an engineering design tool, offering insights into boilers, such as flow patterns, temperature variations, and chemical distribution, that are challenging to acquire experimentally. While CFD has successfully tackled combustion problems in various scenarios like air-coal, oxy-coal, air-biomass, and air-coal-biomass blends [5, 20-28], much of this work has been on a pilot scale, relying on experimental setups [19].

The current research presents the development of an innovative CFD methodology tailored for analyzing an industry-scale tangentially fired boiler with a capacity of 150 MWe utilizing Simcenter STAR-CCM+. The adopted CFD methodology undergoes rigorous validation by comparing the computed results with empirical data obtained from experimental outcomes conducted on an IFRF furnace [29]. This research underscores the significance of employing advanced computational techniques in assessing the performance and behavior of large-scale industrial boilers, which are pivotal components in energy production and various industrial processes. By leveraging Simcenter STAR-CCM+, a state-of-the-art CFD software, the study aims to enhance the understanding of the complex fluid dynamics and combustion processes occurring within the tangentially fired boiler. The validation process involves a meticulous comparison between the simulated results generated by the novel CFD methodology and the empirical findings obtained from experimental studies conducted on an IFRF furnace. Through this comparative analysis, the accuracy and reliability of the developed CFD methodology are evaluated, thereby establishing its suitability for analyzing similar industrial-scale boilers.

2. COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF 150 MWe TANGENTIALLY FIRED BOILER

The computational investigation of the large-scale 150 MWe Tangentially Fired Boiler is performed using Siemens Simcenter STAR CCM+. The simulations are carried out by using the real-time boundary conditions of the industry. The coal combustion analysis is carried out by using the Eulerian-Lagrangian multiphase model with the Eddy Break-up combustion model and the Realizable $k-\epsilon$ turbulence model. The heat transfer inside the boiler is modeled using the Discrete Ordinates (DO) radiation model, and the NO_x formation is modeled using the extended Zeldovich mechanism.

2.1 Geometry creation

The 3D CAD model is created based on the general boiler

assembly (GA) drawing obtained from Aditya Aluminum, Sambalpur, Odisha. The boiler geometry ($11 \text{ m} \times 9 \text{ m} \times 34 \text{ m}$) is created in the Simcenter Star-CCM+ CAD interface, as shown in Figure 1 and Figure 2.

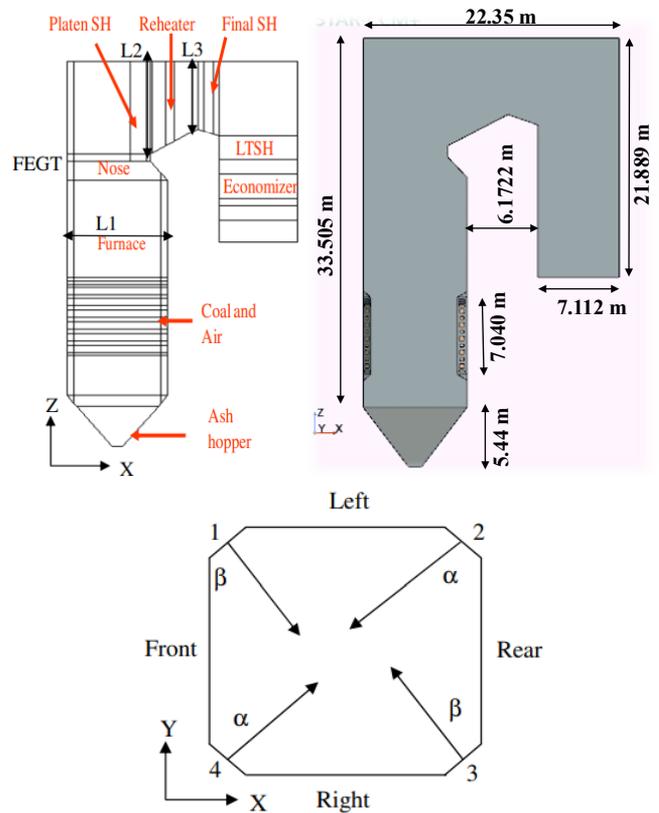


Figure 1. Schematic drawing of 150 MWe tangentially fired coal boiler with burners

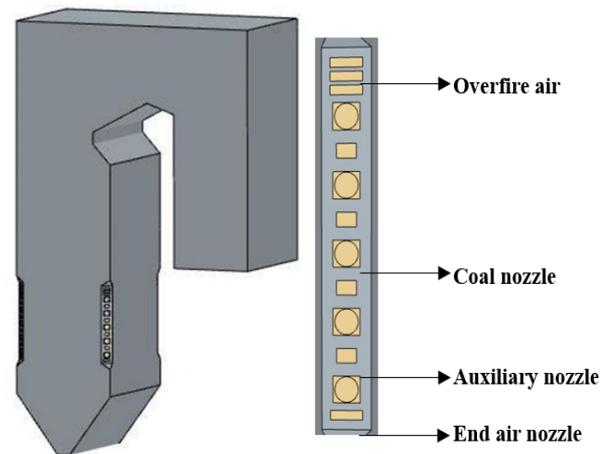


Figure 2. 3D CAD model of the boiler and the burner arrangement

It consists of a Furnace, Crossover pass, and Rear pass. The furnace consists of the platen, re-heater, final super-heater, LTSH, and lower and upper economizer. A reheater increases the temperature of steam that has partially expanded in the high-pressure (HP) turbine before sending it to the intermediate-pressure (IP) turbine. It is positioned after the HP turbine and before the IP turbine. It improves the overall thermal efficiency of the power plant and reduces the moisture content in the steam. Platen Superheater heats steam from the boiler drum to a high temperature before it enters the final superheater. Typically

found in the radiant zone of the furnace, where it receives direct radiation from the flame. It plays a crucial role in the initial stages of steam superheating. It enhances boiler efficiency by utilizing radiant heat. A Low-Temperature Superheater (LTSH) heats steam to a moderate temperature before it enters the final superheater. It is placed in the convective zone of the boiler. It gradually increases the steam temperature to reduce thermal stress on components. It serves as an intermediate stage in the steam superheating process. The final Superheater heats the steam to the final desired superheat temperature before it enters the HP turbine. It is placed in the hottest part of the flue gas path, usually near the furnace exit or in the convective pass. It maximizes the thermal efficiency of the power cycle and ensures that steam entering the turbine is at the optimal temperature and pressure. The economizer preheats the feedwater entering the boiler using the residual heat in the flue gases. It is installed in the flue gas path, typically after the superheater and reheater sections. It enhances overall boiler efficiency by recovering waste heat and reduces fuel consumption and operational costs. There are four corners in the furnace operating burners. Primary air and coal are injected through six auxiliary air nozzles from the fuel-air burners through which secondary air is passed. Fuel-air burners and auxiliary burners are arranged alternately, as shown in Figure 8. The air and coal are injected from the four corners of the furnace at angles of 31° and 43° with the walls.

2.2 Mesh independence study

Discretization plays a significant role in CFD solutions. It is essential to generate a high-quality mesh for an accurate solution. The convergence rate of the solution and the time required for the CPU depend on the mesh. For a suitable solution, the mesh quality depends on the density of the grid, the skewness of the cell, and the length/volume ratio of adjacent cells. The quality of the mesh should be high enough to capture all the combustion characteristics in the furnace. The boiler has meshed with polyhedral cells, as shown in Figure 3 and Figure 4.

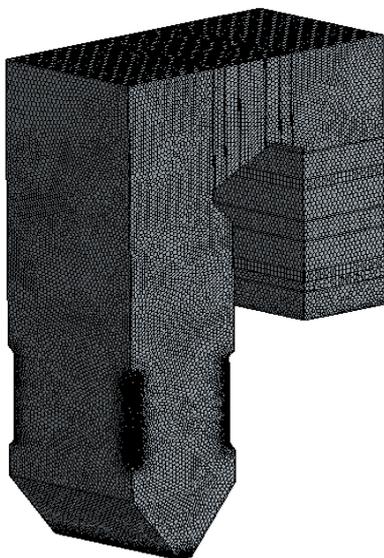


Figure 3. Discretization of the boiler

Polyhedral cells are a popularly used mesh due to their decreased cell count and increased gradient approximation because of more neighbouring cells. Initially, the mesh was generated with 1.36 million cells. Further, the mesh was

generated with 2.04 million cells and 3.06 million cells to check the dependency of the mesh on the solution. The mesh with 3.06 million cells is considered a grid-independent mesh because of its good accuracy in the solution. Mesh is refined at the critical regions of the furnace, and prism layers are introduced near the furnace walls to properly capture the physics near the walls. The temperature variation along the height of the furnace for three different mesh sizes is shown in Figure 5.

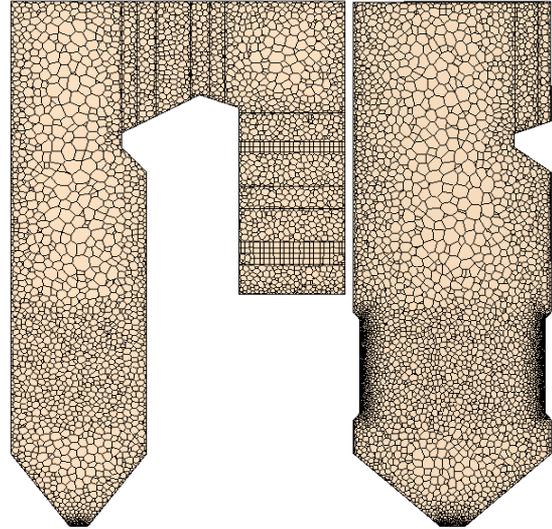


Figure 4. Mesh representation on the central and diagonal plane of the boiler

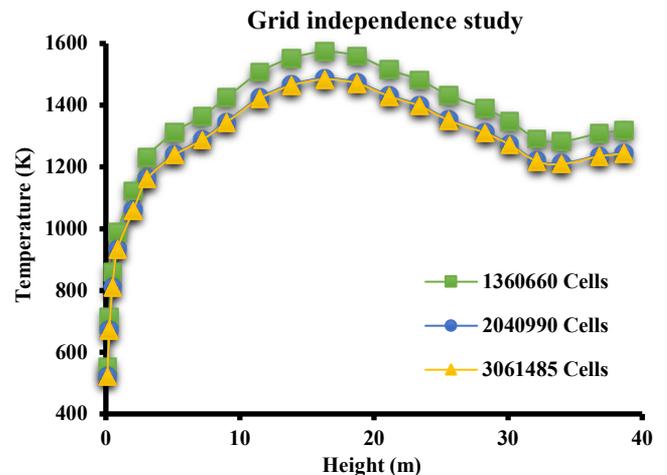


Figure 5. Temperature variation along the height of the furnace for three different meshes

2.3 Coal analysis and boundary conditions

Coal characterization is commonly carried out using two standard methods: Proximate analysis and ultimate analysis. These analyses provide essential insights into the composition and combustion behaviour of coal, which are critical for designing and optimizing thermal systems. Proximate analysis focuses on evaluating how coal responds when subjected to heating. It quantifies the mass percentages of four key constituents, such as Moisture, Volatile matter, Fixed carbon, and Ash.

In contrast, the ultimate analysis provides a detailed breakdown of the elemental composition of coal, typically

including carbon (C), hydrogen (H), nitrogen (N), sulfur (S), and oxygen (O). This method offers a deeper understanding of the coal's chemical makeup, which is essential for calculating theoretical air requirements and emissions. The proximate and ultimate analysis properties of coal are presented in Table 1, which serves as a comprehensive reference for providing the coal properties in the CFD simulation.

Table 1. Proximate and ultimate analysis of coal

Parametric Analysis		Ultimate Analysis	
Fixed carbon	26.32	Carbon	37.84
Volatile matter	22.1	Hydrogen	2.8
Moisture	11.51	Sulphur	0.44
Ash	40.07	Nitrogen	1.83
		Oxygen	5.51

The coal used in this study contains a high ash fraction of 40.07%, significantly exceeding typical Indian thermal coal (30–35%), which strongly influences combustion behaviour. CFD predictions indicate reduced flame temperature and delayed ignition due to ash dilution of the combustible matter, resulting in a longer and weaker flame. Ash accumulation on char particles restricts oxygen diffusion, leading to incomplete burnout and higher predicted unburnt carbon at the furnace exit. The reduced radiative emissivity of ash lowers wall heat flux, causing an increase in furnace exit gas temperature. Consequently, elevated CO levels and a pronounced CO–NO_x trade-off are observed, highlighting the need for combustion optimization when firing high-ash coal.

Furthermore, the boundary conditions used for the computational analysis of coal combustion are outlined in Table 2. These boundary conditions define the operational parameters and constraints applied during the analysis and are critical for ensuring realistic and accurate CFD predictions.

Table 2. Boundary conditions [Industry operating parameters]

Parameter	Value
Primary airflow rate	196 TPH
Temperature of primary air	298 °C
Secondary air flow rate	398 TPH
Temperature of secondary air	300 °C
Airflow rate (bottom ash hopper)	30 TPH
Temperature of air (bottom ash hopper)	250 °C
Coal flow rate	80 TPH
Temperature of coal	298 °C
Gross calorific value (GCV)	3534 Kcal/kg
Coal particle size	70 μm

2.4 Validation

The adopted CFD methodology is utilized for the analysis of the IFRF furnace [29]. Figure 6 represents the 3D CAD model and the discretization of the furnace. The furnace is 6250 mm in length and 2000 mm in width. The mesh is generated on the geometry of the furnace with 22277 quadrilateral cells of a base size of 0.3 m. The prism layer cells are used at the furnace walls to capture the physics near the walls. The mesh is refined at the critical regions of the furnace to capture the combustion process in the furnace accurately. The boundary conditions and the proximate & ultimate analysis of coal are depicted in Table 3 and Table 4.

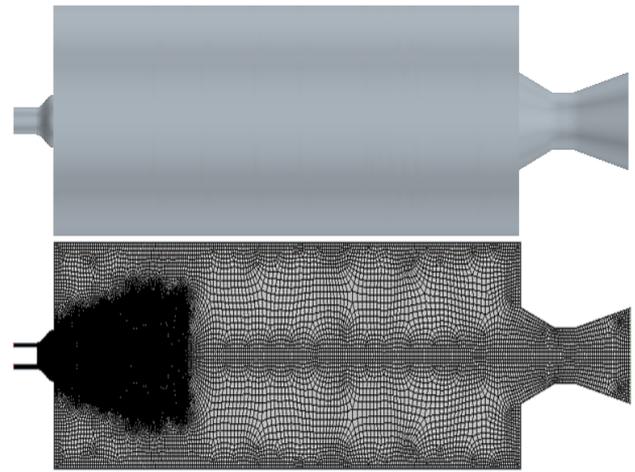


Figure 6. 3D CAD model and discretization of the furnace

Table 3. Proximate and ultimate analysis of coal [24]

Proximate Analysis		Ultimate Analysis	
Fixed carbon	54.3	Carbon	80.36
Volatile matter	37.4	Hydrogen	5.08
		Sulphur	0.94
		Nitrogen	1.45
Ash	8.3	Oxygen	12.17
		LCV (MJ/kg-dry)	32.32

Table 4. Boundary conditions [24]

Inlet for Combustion Air	
Mass flow rate (dry)	2684 kg/hr
Temperature	573.15 K
Transport Air Inlet and Pulverized Coal	
Coal mass flow rate (dry)	263 kg/hr
Air mass flow rate (dry)	421 kg/hr
Temperature	343.15

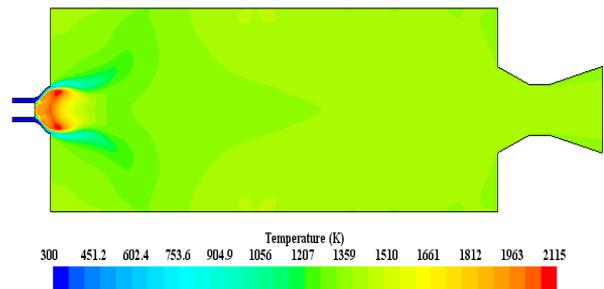


Figure 7. Temperature contour

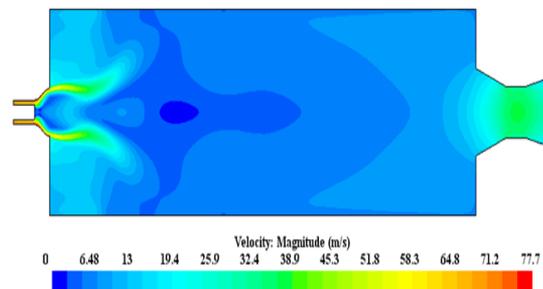


Figure 8. Velocity contour

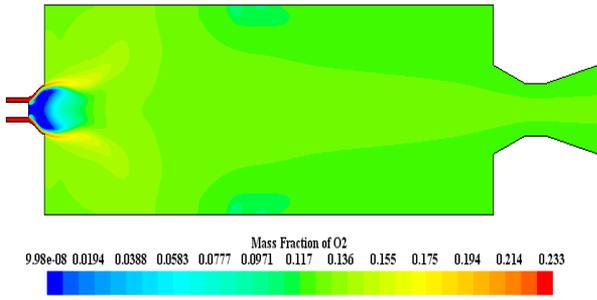


Figure 9. O₂ Mass fraction contour

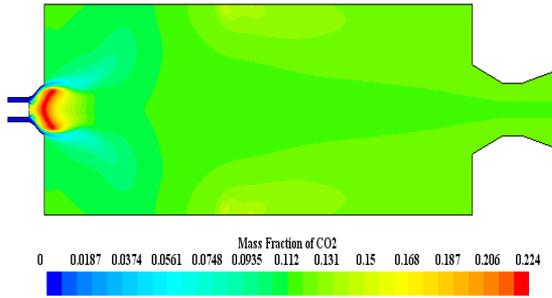


Figure 10. CO₂ mass fraction contour

The CFD-predicted outcomes are compared with the experimental findings of Peters and Weber [29]. The proposed CFD methodology is validated with the help of a grid independence study. The CFD-predicted velocity, temperature, and species distribution in the furnace are shown in Figures 7-10. The CFD-predicted temperature profile is in close agreement with the experimental findings [29] as shown in Figure 11. The computed results are in close agreement with the experimental findings, and the deviation is within 3%.

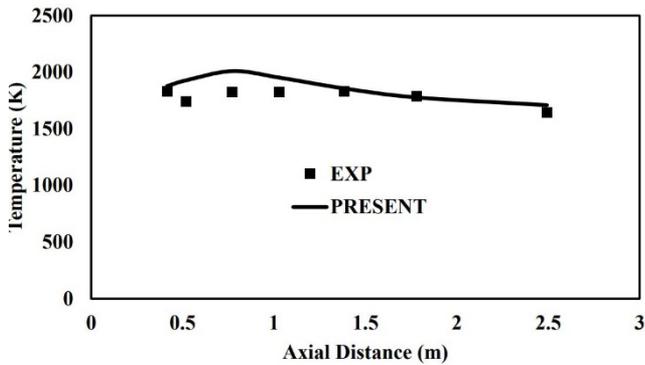


Figure 11. Comparison of temperature variation along the axial distance

3. RESULTS AND DISCUSSION

3.1 Orientation of planes

Different planes are considered (shown in Figure 12) in the boiler to analyze the results. The two vertical planes represent the variation of any variable along the height of the furnace, and six horizontal cross-sectional planes in the furnace along its height represent the variation of various parameters. The elevation of these planes from the bottom of the boiler is section (i) ($Z = 8.73$ m), section (ii) ($Z = 9.84$ m), section (iii) ($Z = 11.1$ m), and section (iv) ($Z = 12.36$ m), section (v) ($Z =$

13.6 m) and section (vi) ($Z = 14.73$ m).

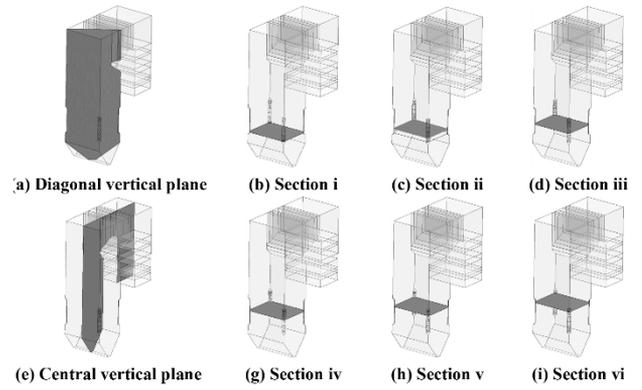


Figure 12. Orientation of different planes in the boiler for the illustration of outcomes

3.2 Gas velocity and its vectors

The velocity contours and their vector plots in different planes along the height of the furnace are shown in Figures 13 and 14. In the regions near the burners, it was observed that the flow was more active than in the other regions. Due to the arrangement of the burner angles, an anticlockwise vortex flow is generated at the furnace center. The vortex flow is more Weber robust in the lower levels than in the upper levels. The vortex flow is reduced remarkably in the upper levels of the furnace. The velocity variation at the centerline of the furnace along the height is shown in Figure 15. A significant drop in the velocity in the combustion zone is observed because the velocity at the center of the vortex is low.

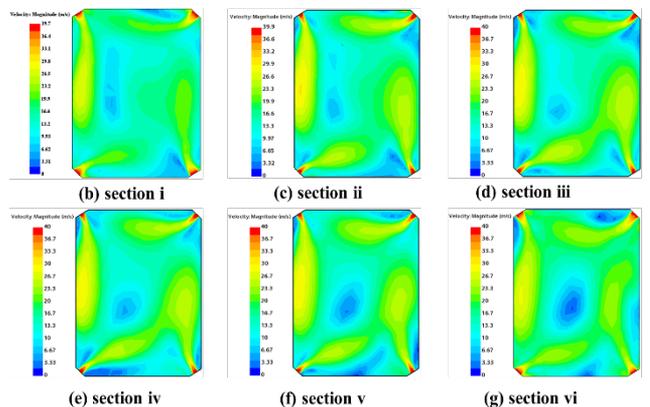
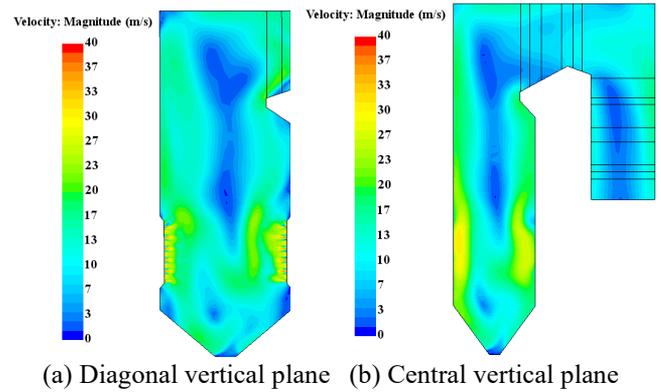


Figure 13. Velocity distribution on different planes in the furnace

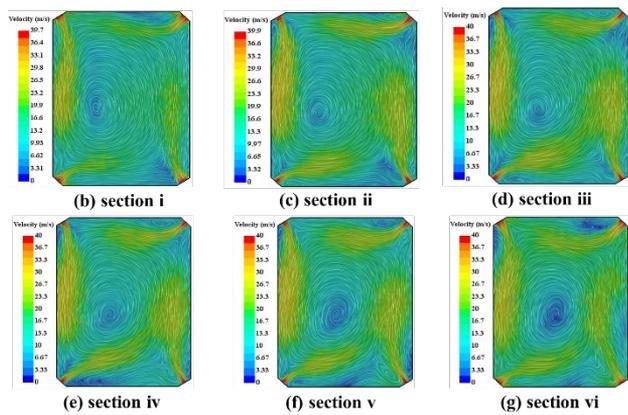
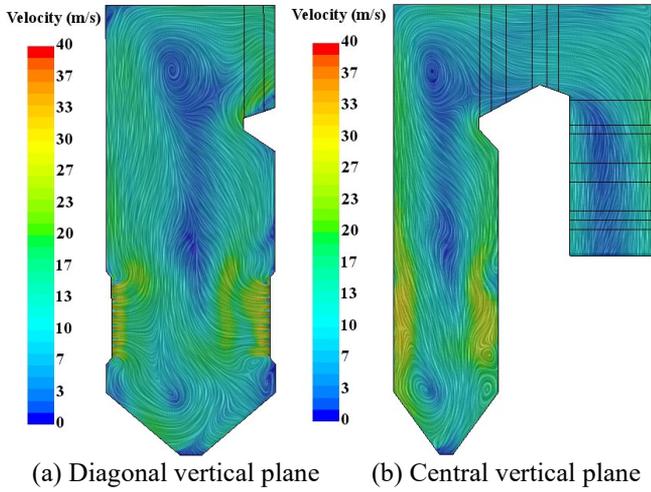


Figure 14. Velocity vectors on different planes in the furnace

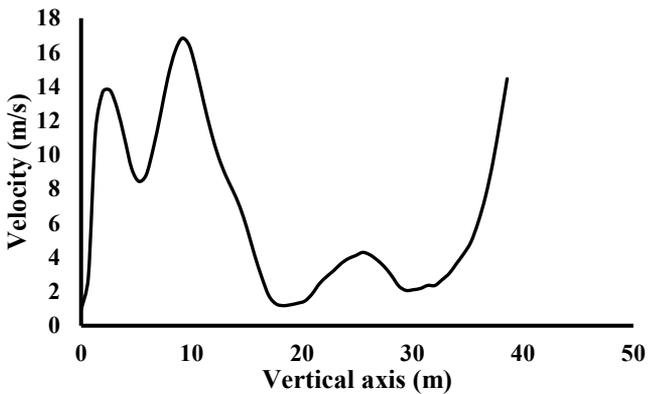


Figure 15. Variation of velocity along the height of the furnace

3.3 Streamlines and particle trajectories

Figures 16 and 17 illustrate the three-dimensional streamlines and coal particle trajectories inside the furnace, revealing a highly complex and strongly turbulent flow structure characteristic of tangentially fired systems. The streamlines indicate the formation of intense swirling motion and large-scale recirculation zones, which play a crucial role in promoting effective mixing between the combustion air and pulverized coal. This enhanced mixing improves local oxygen availability around coal particles, thereby supporting stable ignition and sustained combustion.

The CFD-predicted flow field demonstrates that turbulence

generated by the bulk rotational motion significantly enhances convective and radiative heat transfer within the furnace. Elevated turbulence levels increase interphase momentum and heat exchange, resulting in more uniform temperature distributions and improved thermal interaction between the flue gas and furnace walls.

Coal particle trajectory analysis shows that particles largely follow the general gas-phase flow pattern, confirming strong aerodynamic coupling between the phases. However, noticeable deviations from the gas streamlines are observed due to particle inertia, density differences, and local flow unsteadiness. These deviations influence particle residence time and spatial distribution, which in turn govern char burnout, unburned carbon levels, and ash deposition tendencies. Overall, the combined streamline and particle trajectory analysis provides valuable physical insight into the interaction between flow aerodynamics, particle dynamics, and combustion performance within the boiler.

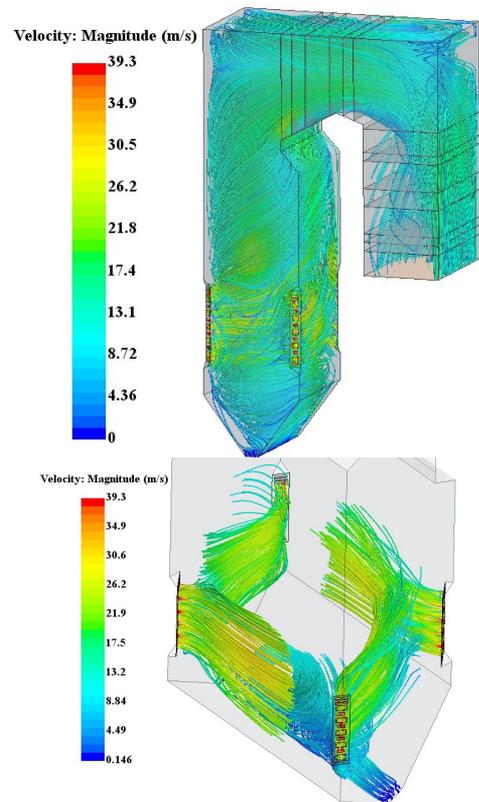
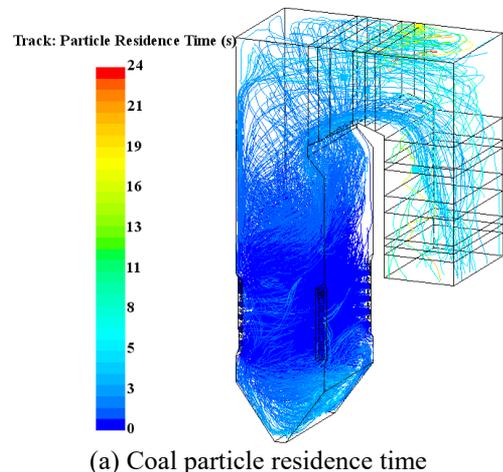
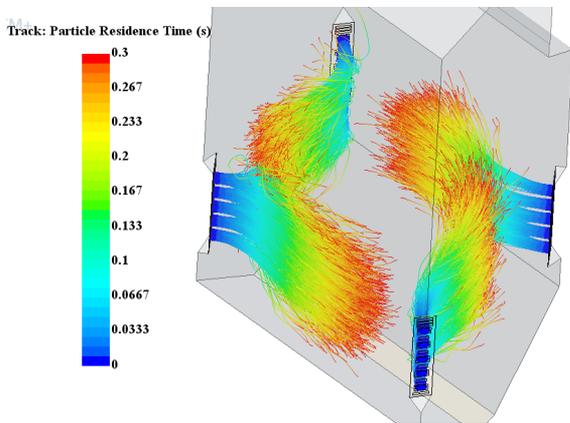


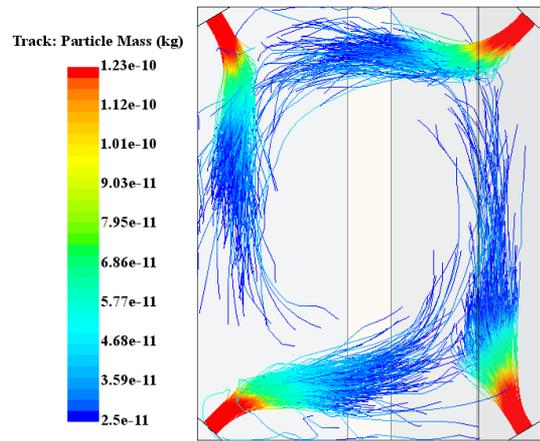
Figure 16. Streamlines of the flue gas in the boiler and furnace region



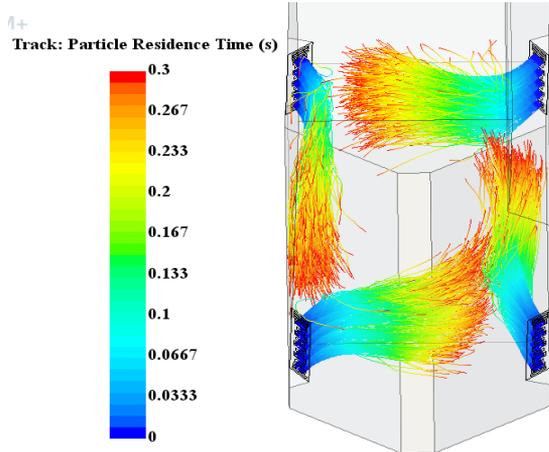
(a) Coal particle residence time



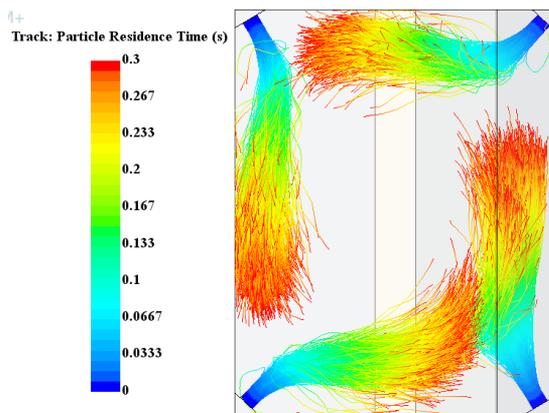
(b) Residence time (Isometric view)



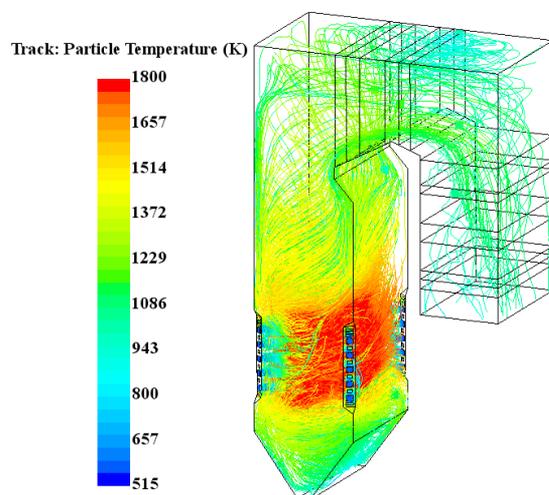
(f) Mass of coal particles



(c) Residence time (Inclined view)



(d) Residence time (Top view)



(e) Temperature of the coal particles

Figure 17. Coal particle trajectories

With the introduction of coal particles and air through the lower burners, in the beginning, the flow moves around the furnace's bottom and the bottom of the ash hopper. Subsequently, the particles ascend through the high-temperature vortex, or fireball, formed in the central furnace area. In contrast, coal particles and flue gas from the upper burners circulate around the surface of the fireball. This results in shorter residence times for particles injected from higher positions compared to those injected from lower burners. On average, coal particles remain within the system for 24 s.

3.4 Temperature profile

Figure 18 presents the CFD-predicted temperature contours across multiple transverse and longitudinal planes of the furnace, providing detailed insight into the thermal structure of the tangentially fired boiler. A pronounced temperature peak is observed in the central furnace region, which corresponds to the zone of maximum combustion intensity created by the interaction of tangentially injected coal–air jets and the resulting strong recirculation. This high-temperature core confirms effective ignition, rapid volatile release, and sustained char oxidation within the primary combustion zone. As the combustion products move upward, the CFD results clearly capture the progressive reduction in flue gas temperature caused by efficient heat extraction to the furnace water walls and downstream heat exchangers, including the economizer, superheater, and reheater sections. The predicted temperature fields demonstrate a systematic variation with furnace height, reflecting the spatial redistribution of heat release and heat absorption along the gas flow path. Air entering the furnace through the burners at approximately 573 K undergoes rapid heating and reaches a peak temperature of about 2008 K in the central furnace, highlighting the effectiveness of turbulent mixing and radiative heat transfer in establishing a stable and intense flame.

Figure 19 further quantifies this behaviour by depicting the axial variation of gas temperature along the furnace height and its comparison with real-time plant data. The highest temperatures are attained within the main furnace region, followed by a gradual decline as the flue gas transfers energy to the surrounding walls and to the heat exchange surfaces in the crossover and rear passes. The close agreement between CFD predictions and measured temperatures validates the model's ability to accurately resolve the coupled processes of combustion, turbulence, and heat transfer. Overall, the

temperature analysis demonstrates the strength of the CFD approach in elucidating the physical mechanisms governing heat release, heat absorption, and thermal performance in large-scale tangentially fired boilers.

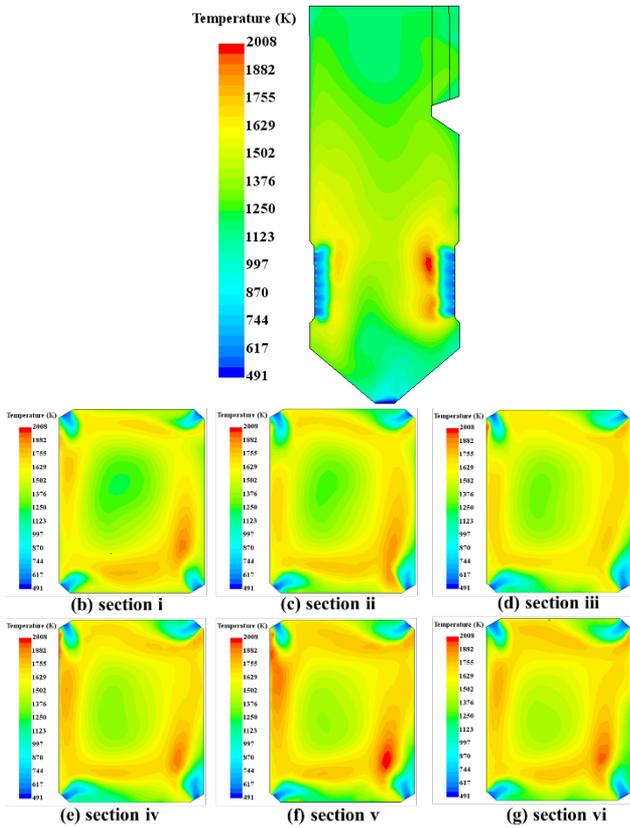


Figure 18. Temperature distribution along the furnace height and on various planes

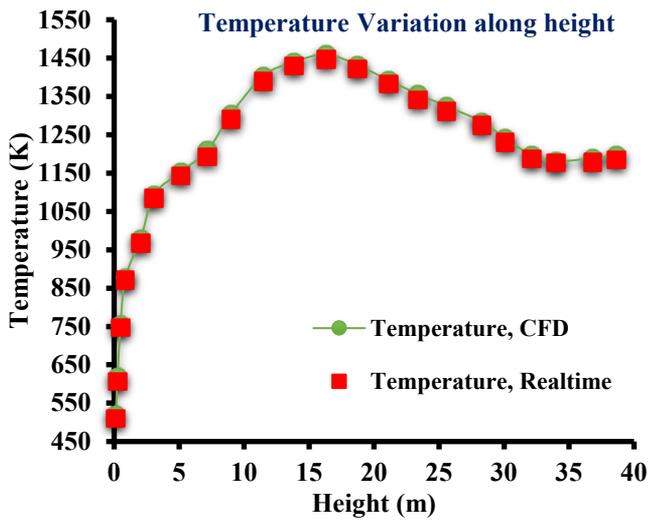


Figure 19. Variation of temperature along the height of the furnace

3.5 Species profile

The mass fraction profiles of O_2 , CO_2 , and CO concentration on different planes are shown in Figures 20-22, respectively. It is observed that near the tip of the burners, the concentration of O_2 is high, and it is quickly consumed as char and volatile matter react with O_2 (Figure 20). An opposite trend of the CO_2 mass fraction to the O_2 mass fraction. The values of oxygen, carbon dioxide, and carbon monoxide observed at the exit of

the furnace are 3.1% (mass), 8.9% (mass), and 14.29% (mass). It is observed that the regions of low oxygen are the high-temperature regions.

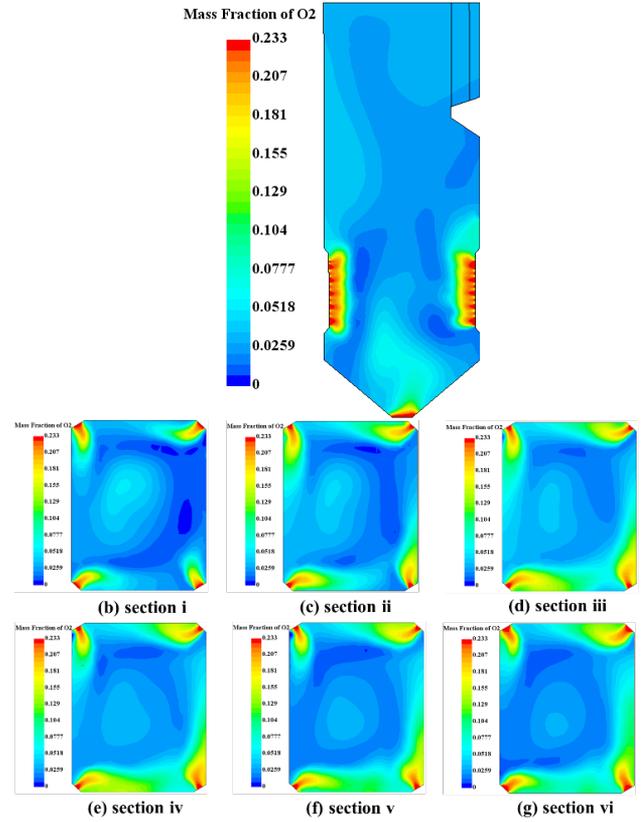


Figure 20. O_2 mass fraction distribution along the furnace height and on various planes

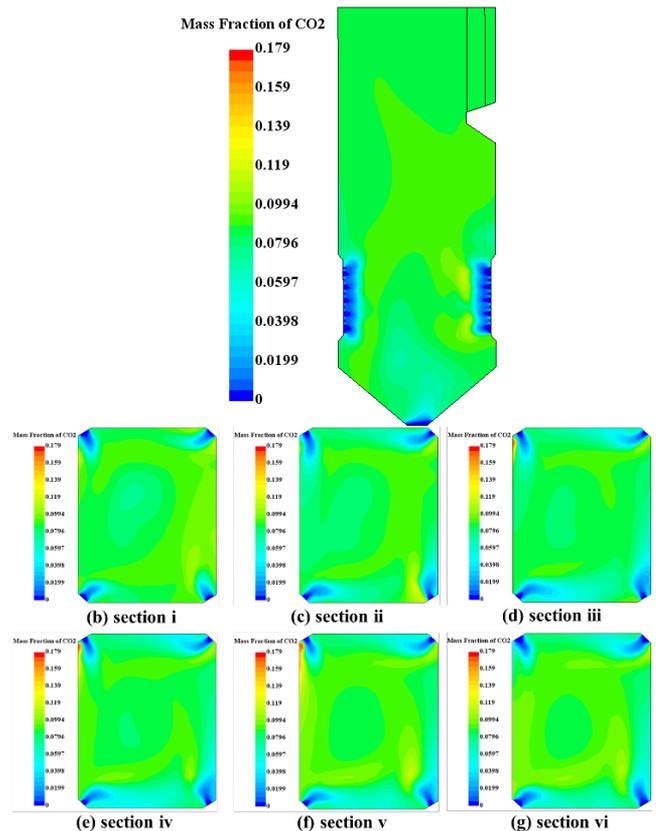


Figure 21. CO_2 mass fraction distribution along the furnace height and on various planes

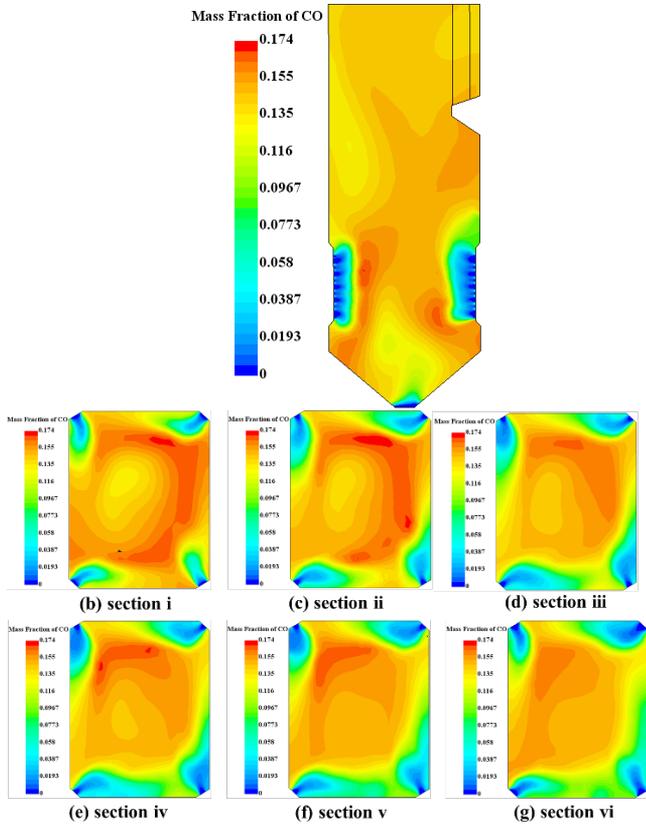


Figure 22. CO Mass Fraction distribution along the furnace height and on various planes

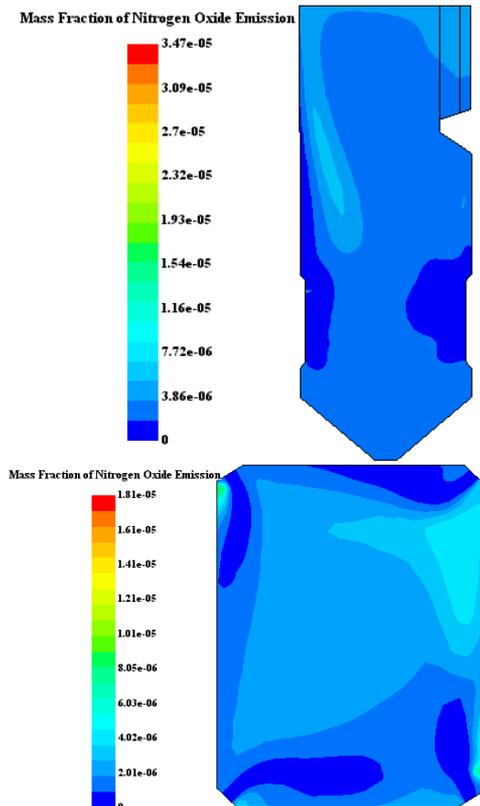


Figure 23. NO_x Mass Fraction contour on the diagonal and horizontal plane

3.6 NO_x emissions

Figure 23 depicts NO_x emissions in the vertical sectional

plane and horizontal cross-section. The concentration of NO_x around the burner end is lower than in the core region of the furnace.

3.7 3D flames and fireball

Figure 24 depicts the three-dimensional flame propagation and fireball in the furnace. The temperature of the fireball region is found to be lower than the temperature of the flames (Figure 24). The surface temperature of the fireball is between 1440 K and 1500 K.

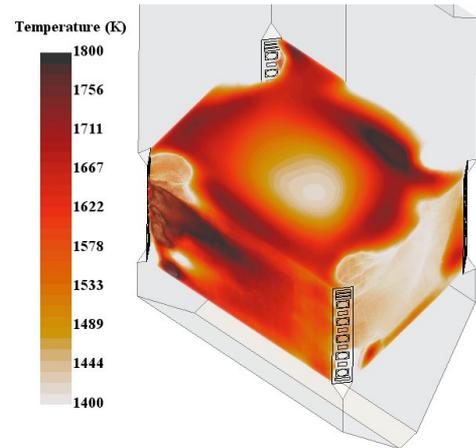


Figure 24. 3D Flame in the furnace

3.8 Radiation heat transfer on the furnace walls

Figure 25 depicts the distribution of radiation heat flux on the furnace walls. Because of intense combustion and extreme heat in that zone, the radiation heat flow is higher near the combustion zone. The most significant recorded radiated heat flux is 379 kW/m².

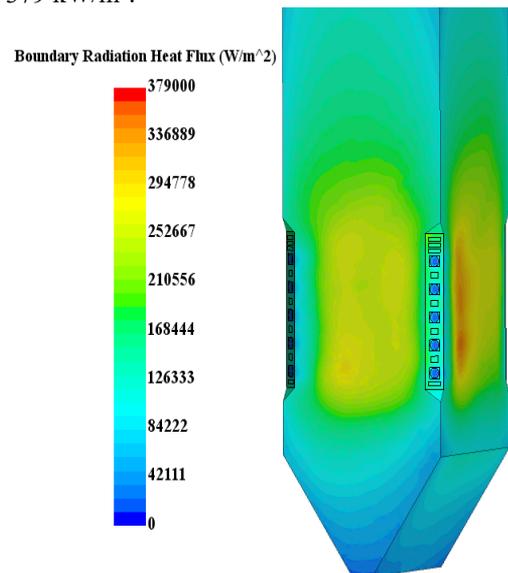


Figure 25. Radiation heat flux on the furnace walls

3.9 Results comparison

The validation of numerical results with real-time functioning plant data (Aditya Aluminium, Sambalpur) is shown in Table 5. From this study, it is concluded that the numerical results for pure coal combustion agree with plant data. This indicates that our models are performing adequately.

Table 5. Comparison of plant data with numerical results

Description	Plant Data	Numerical Results
O ₂ in the gas at the economizer outlet	3.52%	3.36%
CO ₂ in the gas at the economizer outlet	16.34%	15.52

3.10 Engineering applicability

This study developed a validated CFD method that is robust, general, and easy to scale up, so it can be used on tangentially fired boilers of all sizes and operating conditions. It can also be easily adapted to fuel-blending setups that use more than one fuel. These setups are becoming more common in industrial power plants because they make the plants more flexible and lower costs and emissions.

In terms of usefulness, this flexibility makes the suggested CFD method a useful engineering tool for analysing real-world boilers. It lets plant designers and operators carefully look at how changes to the design, operating strategies, and fuel types affect combustion performance, heat transfer behaviour, and pollutant emissions, especially NO_x. Because of this, the method helps people make smart choices about how to improve boiler design, retrofit it, and control emissions without having to do a lot of trial-and-error experiments.

In general, the demonstrated scalability and versatility make the CFD framework much more useful and relevant in industry. It is now a reliable platform for both performance improvement and emission reduction studies in large-scale tangentially fired boiler systems.

4. CONCLUSION

A state-of-the-art CFD methodology has been developed to perform an in-depth analysis of an industrial-scale 150 MWe tangentially fired pulverized coal boiler. This advanced modeling approach is capable of capturing the complex physics of combustion, fluid flow, heat transfer, and pollutant formation within large-scale boiler systems.

To ensure accuracy and reliability, the CFD model underwent rigorous validation against well-established experimental data from the International Flame Research Foundation (IFRF) furnace. The validation process confirmed that the numerical predictions—such as temperature profiles, velocity distributions, and species concentrations—were in close agreement with the benchmark experimental measurements, establishing the credibility of the model. Additionally, a comprehensive grid independence study was performed to eliminate the influence of mesh size on the results, thereby enhancing the robustness and consistency of the simulations. Following validation, the CFD methodology was applied to simulate the operational behaviour of a 150 MWe tangentially fired boiler in service at Aditya Aluminium, Sambalpur, Odisha. This real-world application involved detailed modelling of:

- Combustion dynamics, including ignition, flame stability, and burn-out behaviour of pulverized coal,
- Flow field analysis, capturing the recirculating flow patterns typical of tangential firing,
- Heat transfer mechanisms, with particular emphasis on the transfer of thermal energy from flue gases to boiler surfaces,

- Pollutant formation and emission, including NO_x and unburned carbon assessment,
- Coal particle tracking, to evaluate the velocity distribution and residence time of coal particles throughout the furnace.

The simulation results demonstrated a high degree of correlation with operational plant data, specifically in terms of temperature distributions and species mass fractions, underscoring the model's predictive capability. The CFD analysis also enabled visualization and quantification of coal particle trajectories and velocities, which are crucial for optimizing burner configuration and combustion efficiency.

Overall, this validated and industrially applied CFD methodology serves as a powerful and reliable tool for conducting detailed combustion performance evaluations of tangentially fired pulverized coal boilers. Its successful deployment in a full-scale operational environment highlights its potential for aiding in the design, optimization, and environmental assessment of large industrial combustion systems. Also, it can be used for the detailed analysis of a tangentially fired boiler with fuel blending.

This CFD methodology is only applicable for the combustion analysis of a tangentially fired boiler. It cannot be used for the analysis of fluidized bed boilers.

ACKNOWLEDGMENT

The authors extend their sincere gratitude to Dr. Meghana Thimmappa, Assistant Professor in the Department of HSS at ITER, SOA University, Bhubaneswar, for her invaluable assistance in refining the manuscript. Her meticulous editing and keen eye for detail significantly improved the English language and corrected any grammatical inconsistencies, ultimately enhancing the quality of the work.

REFERENCES

- [1] Silva, C.V.D., Indrusiak, M.L.S., Beskow, A.B. (2010). CFD analysis of the pulverized coal combustion processes in a 160 MWe tangentially-fired-boiler of a thermal power plant. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 32(4): 427-436. <https://doi.org/10.1590/S1678-58782010000400004>
- [2] Choi, C.R., Kim, C.N. (2009). Numerical investigation on the flow, combustion and NO_x emission characteristics in a 500MWe tangentially fired pulverized-coal boiler. *Fuel*, 88(9): 1720-1731. <https://doi.org/10.1016/j.fuel.2009.04.001>
- [3] Fan, J.R., Liang, X.H., Chen, L.H., Cen, K.F. (1998). Modeling of NO_x emissions from a W-shaped boiler furnace under different operating conditions. *Energy*, 23(12): 1051-1015. [https://doi.org/10.1016/S0360-5442\(98\)00059-0](https://doi.org/10.1016/S0360-5442(98)00059-0)
- [4] Xu, M., Azevedo, J.L.T., Carvalho, M.G. (2000). Modelling of the combustion process and NO_x emission in a utility boiler. *Fuel*, 79(13): 1611-1619. [https://doi.org/10.1016/S0016-2361\(00\)00019-3](https://doi.org/10.1016/S0016-2361(00)00019-3)
- [5] Backreedy, R.I., Jones, J.M., Ma, L., Pourkashanian, M., Williams, A., Arenillas, A., Arias, B., Pis, J.J., Rubiera, F. (2005). Prediction of unburned carbon and NO_x in a tangentially fired power station using single coals and

- blends. *Fuel*, 84(17): 2196-2203. <https://doi.org/10.1016/j.fuel.2005.05.022>
- [6] Huang, L., Li, Z., Sun, R., Zhou, J. (2006). Numerical study on the effect of the Over-Fire-Air to the air flow and coal combustion in a 670 t/h wall-fired boiler. *Fuel Processing Technology*, 87(4): 363-371. <https://doi.org/10.1016/j.fuproc.2005.10.008>
- [7] Le Bris, T., Cadavid, F., Caillat, S., Pietrzyk, S., Blondin, J., Baudoin, B. (2007). Coal combustion modelling of large power plant, for NOx abatement. *Fuel*, 86(14): 2213-2220. <https://doi.org/10.1016/j.fuel.2007.05.054>
- [8] Díez, L.I., Cortés, C., Pallarés, J. (2008). Numerical investigation of NOx emissions from a tangentially-fired utility boiler under conventional and overfire air operation. *Fuel*, 87(7): 1259-1269. <https://doi.org/10.1016/j.fuel.2007.07.025>
- [9] Nimalipuri, P., Singh, V., Vitankar, V., Das, H.C., Pradhan, M.K. (2024). Numerical prediction of fire dynamics and the safety zone in large-scale multiple pool fire in a dike using flamelet model. *The Canadian Journal of Chemical Engineering*, 102(1): 11-29. <https://doi.org/10.1002/cjce.25094>
- [10] Nimalipuri, P., Singh, V., Das, H.C., Pradhan, M.K., Vitankar, V. (2023). Consequence analysis of heptane multiple pool fire in a dike. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 238(5): 1811-1827. <https://doi.org/10.1177/09544062231181813>
- [11] Nimalipuri, P., Das, H., Pradhan, M. (2020). Simulation of emission from coal-fired power plant. In: Biswal, B., Sarkar, B., Mahanta, P. (eds) *Advances in Mechanical Engineering. Lecture Notes in Mechanical Engineering*. Springer, Singapore. https://doi.org/10.1007/978-981-15-0124-1_87
- [12] Pradhan, M.K., Nimalipuri, P., Malaya, K., Pradhan, Chandra, H., Rabindra, D., Mahapatra, N., Das, B. (2018). Prediction of air pollutants emitting from chimney of a CHP using CFD. *International Journal of Scientific and Engineering Research*, 9(4): 105-110.
- [13] Yin, C., Caillat, S., Harion, J.L., Baudoin, B., Perez, E. (2002). Investigation of the flow, combustion, heat-transfer and emissions from a 609 MW utility tangentially fired pulverized-coal boiler. *Fuel*, 81(8): 997-1006. [https://doi.org/10.1016/S0016-2361\(02\)00004-2](https://doi.org/10.1016/S0016-2361(02)00004-2)
- [14] Yin, C., Rosendahl, L., Condra, T.J. (2003). Further study of the gas temperature deviation in large-scale tangentially coal-fired boilers*. *Fuel*, 82(9): 1127-1137. [https://doi.org/10.1016/S0016-2361\(02\)00418-0](https://doi.org/10.1016/S0016-2361(02)00418-0)
- [15] Kokkinos, A. (2002). Reducing NOx emissions in tangentially-fired boilers-a new approach. *Fuel and Energy Abstracts*, 43(4): 290. [https://doi.org/10.1016/S0140-6701\(02\)86525-0](https://doi.org/10.1016/S0140-6701(02)86525-0)
- [16] Pallarés, J., Arauzo, I., Díez, L.I. (2005). Numerical prediction of unburned carbon levels in large pulverized coal utility boilers. *Fuel*, 84(18): 2364-2371. <https://doi.org/10.1016/j.fuel.2005.05.011>
- [17] Spitz, N., Saveliev, R., Perelman, M., Korytni, E., Chudnovsky, B., Talanker, A., Bar-Ziv, E. (2008). Firing a sub-bituminous coal in pulverized coal boilers configured for bituminous coals. *Fuel*, 87(8-9): 1534-1542. <https://doi.org/10.1016/j.fuel.2007.08.020>
- [18] Korytnyi, E., Saveliev, R., Perelman, M., Chudnovsky, B., Bar-Ziv, E. (2009). Computational fluid dynamic simulations of coal-fired utility boilers: An engineering tool. *Fuel*, 88(1): 9-18. <https://doi.org/10.1016/j.fuel.2008.08.010>
- [19] Kumar, A., Nimalipuri, P., Singh, V., Vitankar, V., Das, H.C., Pradhan, M.K., Panda, M.K. (2023). Simulation of natural gas combustor using CFD. In: Pradhan P, Pattanayak B, Das HC, Mahanta P, editors. *Recent Advances in Mechanical Engineering*, Singapore: Springer Nature, pp. 667-679. https://doi.org/10.1007/978-981-16-9057-0_72
- [20] Nimalipuri, P., Singh, V., Vitankar, V., Das, H.C., Pradhan, M.K., Jena, S. (2025). Predicting the safety zone and fire dynamics of heptane multiple pool fire in a square dike using flamelet-generated manifold model. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 47: 44. <https://doi.org/10.1007/s40430-024-05363-2>
- [21] Gharebaghi, M., Irons, R.M.A., Ma, L., Pourkashanian, M., Pranzitelli, A. (2011). Large eddy simulation of oxy-coal combustion in an industrial combustion test facility. *International Journal of Greenhouse Gas Control* 5(supplement 1): S100-S110. <https://doi.org/10.1016/j.ijggc.2011.05.030>
- [22] Toporov, D., Bocian, P., Heil, P., Kellermann, A., Stadler, H., Tschunko, S., Förster, M., Kneer, R. (2008). Detailed investigation of a pulverized fuel swirl flame in CO₂/O₂ atmosphere. *Combustion and Flame*, 155(4): 605-618. <https://doi.org/10.1016/j.combustflame.2008.05.008>
- [23] Ma, L., Jones, J.M., Pourkashanian, M., Williams, A. (2007). Modelling the combustion of pulverized biomass in an industrial combustion test furnace. *Fuel*, 86(12-13): 1959-1965. <https://doi.org/10.1016/j.fuel.2006.12.019>
- [24] Gubba, S.R., Ingham, D.B., Larsen, K.J., Ma, L., Pourkashanian, M., Tan, H.Z., Williams, A., Zhou, H. (2012). Numerical modelling of the co-firing of pulverised coal and straw in a 300MWe tangentially fired boiler. *Fuel Processing Technology*, 104: 181-188. <https://doi.org/10.1016/j.fuproc.2012.05.011>
- [25] Singh, V., Nimalipuri, P., Das, H.C., Vitankar, V. (2025). Experimental and numerical study on multiphase fluid dynamics of coal and blends with biomass (sugarcane bagasse) in a lab-scale full-loop circulating fluidized bed gasifier. *International Journal of Coal Preparation and Utilization*, 45(4): 36-72. <https://doi.org/10.1080/19392699.2024.2350448>
- [26] Singh, V., Nimalipuri, P., Das, H.C., Vitankar, V. (2024). Co-combustion of coal and sawdust in a three-dimensional industrial scale BFB boiler (BG-043, 67TPH): A numerical approach. *International Journal of Coal Preparation and Utilization*, 46(1): 126-173. <https://doi.org/10.1080/19392699.2025.2452298>
- [27] Singh, V., Nimalipuri, P., Das, H.C., Vitankar, V. (2025). An Eulerian-Eulerian multifluid simulation for co-combustion of coal and sawdust in industrial scale circulating fluidized bed boiler. *Cleaner Energy Systems*, 10: 100169. <https://doi.org/10.1016/j.cles.2025.100169>
- [28] Jena, S., Singh, V., Nimalipuri, P., Das, H.C., Pradhan, M.K., Vitankar, V., Hotta, T.K. (2025). Hydrodynamics and Mixing analysis of different bed materials with coal in an industrial scale bubbling fluidized bed boiler. *Case Studies in Thermal Engineering*, 69: 105989. <https://doi.org/10.1016/j.csite.2025.105989>
- [29] Peters, A.A.F., Weber, R. (1997). Mathematical modeling of a 2.4 MW swirling pulverized coal flame.

NOMENCLATURE

ν kinematic viscosity, m^2/s
 V velocity, m/s

T local temperature, K
 h specific enthalpy, kJ/kg
 g gravitational acceleration, m/s^2
 t time, s
 μ dynamic viscosity, Ns/m^2
 ρ density, kg/m^3
 P pressure, N/m^2