CHAPTER 6

THERMAL PERFORMANCE ANALYSIS OF FUEL GAS PREHEATERS USING CFD TECHNIQUES

Thivaakar Rajaratinam[1], Kamariah Md Isa[2], Nur Hazwani Mokhtar[3], Ab Aziz Mohd Yusof[2], Aminuddin Saat[1], Kahar Osman[1], Nofrizalidris Darlis[4]

[1]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310, Johor Bahru, Johor, Malaysia

[2]Faculty of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, UiTM, Johor Branch, Masai, Malaysia

[3] Fakulti Teknologi dan Kejuruteraan Mekanikal, Universiti Teknologi Melaka, Melaka, Malaysia

[4]Centre of Automotive & Powertrain Technology, Faculty of Engineering Technology, University Tun Hussein Onn Malaysia, Edu Hub Pagoh, 84600, Muar, Johor, Malaysia

ABSTRACT

Gas turbines have been a cornerstone of power generation technology since their introduction in 1939, operating through the continuous interaction of air compression, fuel combustion, and turbine expansion. Modern designs employ axial compressors equipped with inlet guide vanes (IGV) and variable guide vanes (VGV) to enhance airflow control and efficiency. This study focuses on the optimisation of fluid velocities within a fuel gas preheater, an essential component in gas turbine systems. Using computational fluid dynamics (CFD) analysis, various flow conditions were evaluated to determine their impact on thermal performance. The results revealed that setting the natural gas velocity at 5 m/s and the water velocity at 1 m/s produced the most favourable outcomes, yielding the highest temperature rise, superior heat transfer rates, and a notable pressure drop on the shell side, indicative of enhanced energy exchange. Among the CFD models tested, the k–ε turbulence model coupled with a pressure-based solver demonstrated the most accurate and physically consistent predictions. These findings underscore the importance of optimised flow dynamics in improving heat exchanger efficiency and provide valuable insights for advancing gas turbine

performance and energy system design.

Keywords: Gas Turbine, Fuel Gas Preheater, Computational Fluid Dynamics (CFD), Heat Transfer

6.1 INTRODUCTION

Gas turbines, a popular energy generation technology since 1939 which require air and fuel to operate. A simple gas turbine's main components are shown in Fig. 6.1. The compressor, combustion chamber, and turbine are all mounted on the same shaft line. Modern gas turbines use axial compressors for efficiency and higher flow rates. The compressor consists of rotating and stationary blades, with an inlet guide vane (IGV) controlling air entry. Modern gas turbines also have variable guide vanes (VGV) to improve airflow through the compressors.

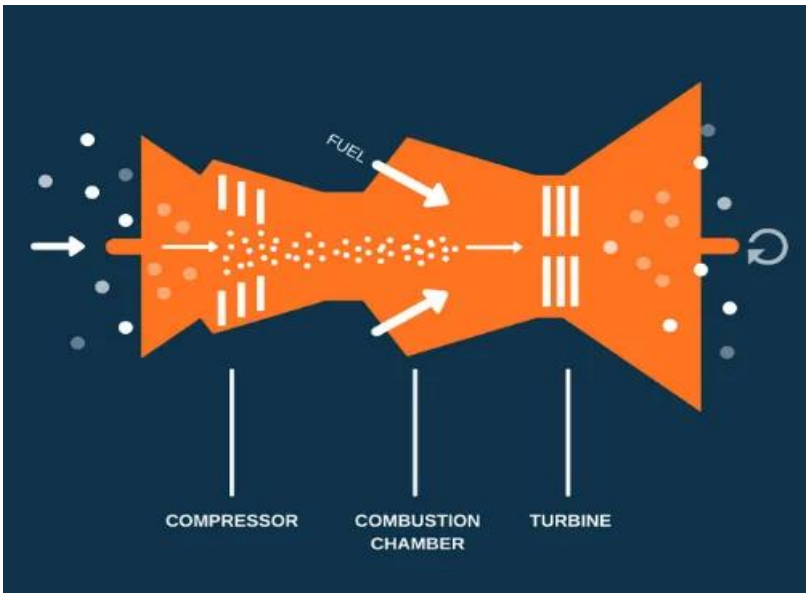

Fig. 6.1: Main Components of a Gas Turbine, Wartsila (2023)

In modern energy systems, pursuing higher efficiency and reduced emissions has led to the widespread adoption of combined cycle power plants, where gas turbines are integrated with Heat Recovery Steam Generators (HRSG) to harness waste heat. Within this configuration, the fuel gas preheater is critical in enhancing combustion efficiency by raising natural gas's temperature before entering the chamber. Effective preheating improves thermal efficiency, fuel-air mixing, ignition delay, and lower pollutant formation.

The performance of a fuel gas preheater is inherently tied to its design, particularly in terms of heat transfer and flow dynamics. Inappropriate sizing or flow conditions may lead to thermal inefficiencies, pressure losses, or uneven

temperature distribution, ultimately affecting the performance of the entire turbine system. To address these challenges, advanced numerical tools such as Computational Fluid Dynamics (CFD) have become indispensable for analysing complex heat exchanger behaviour under various operating conditions.

This study was carried out to investigate the influence of fuel gas preheater design on the performance of gas turbines in a combined cycle power plant with Heat Recovery Steam Generators (HRSG). Using Computational Fluid Dynamics (CFD), several scenarios were simulated to analyse variations in natural gas temperature, pressure, and velocity after preheating. The primary focus was identifying the optimal preheater temperature by examining how water heating affects natural gas properties before entering the combustion chamber. Key monitored parameters include the inlet and outlet temperatures of heating water and natural gas, alongside changes in pressure and velocity.

The first practical gas turbine was built in Paris in 1903 with a three-cylinder, multistage reciprocating compressor, combustion chamber, and impulse turbine. The combustion chamber burnt liquid fuel and compressor air. After cooling with water, the gasses were delivered to an impulse turbine. This technology showed that a gas turbine engine was possible with a thermal efficiency of 3%.

Heavy-duty gas turbines have since been used for power generation purposes. Modern gas turbines can generate more than 600 mw of power while maintaining >60% efficiency in combined cycle arrangements. Air and fuel are the two most important factors in operating a gas turbine. Like all the other internal combustion engines, the gas turbine performance depends mainly on optimising fuel and air ratios. This research will emphasise the significance of the fuel gas temperature.

6.2.1 Using CFD In Shell and Tube Heat Exchanger Analysis

Dario M. Godino et al. (2018) demonstrated the used of the robust capabilities of Computational Fluid Dynamics (CFD) in examining thermal-hydraulic behaviour within complex geometry. Their research produced qualitative and quantitative insights, facilitating an accurate representation of flow patterns, recirculation zones, constricted flow regions, and elevated temperature locations along the heat exchanger tubes. Furthermore, CFD facilitated accurate estimates of global heat transmission and pressure loss across the system.

Heather K. Wiest and Stephen D. Heister (2014) examined the effects of elevated fuel temperatures on the performance and emissions of a modified Rolls-Royce fuel injector under simulated engine conditions using a single-can combustion rig. The findings indicated that increased fuel temperatures augmented vaporisation

and flow enthalpy, enhancing combustion efficiency. However, under test conditions when combustion pressure was inferior to fuel vapour pressure, elevated fuel temperatures resulted in reduced combustion efficiency, ascribed to modified spray atomisation and fuel-air mixing caused by injector flashing. Emission trends were observed to correlate directly with combustion efficiency: enhanced efficiency diminished CO and UHC emissions, whereas NOx levels increased. Moreover, the changed injector exhibited elevated NOx emissions compared to a traditional fuel injector evaluated under the same conditions. Variations in fuel composition resulting from different deoxygenation procedures exhibited no impact on performance.

The study concluded that although fuel temperature significantly influences combustion efficiency, it is not the only determining element, and merely elevating fuel temperature does not ensure corresponding enhancements in performance.

6.2.2 Effects of fuel, inlet air and gas temperature on heat exchanger

O.R. Darbyshire et al. (2006) investigated the effects of fuel gas preheating on the homogeneity of the fuel-air mix in the lean pre-mix burner of a ground-based gas turbine. The results showed an increase in the mix of the fuel and air as the amount of fuel gas preheat was increased. This effect is most likely due to the raised temperature increasing the momentum of the fuel gas jet and, hence, an increase in the penetration of the jet into the mainstream and subsequent improvement in mixing performance.

Similarly, a decrease in the temperature of the inlet air showed an improvement in the mixedness. This can be attributed to the fact that a decrease in temperature will decrease the velocity. As the momentum of the fuel gas remains the same, the ratio of the jet to mainstream momentum must increase, resulting in an improvement in the mixing. Overall, they found that increasing the amount of fuel gas preheat appears to have a beneficial effect on mixing the fuel and air within the burner and the amounts of NOx emissions produced.

David W. Naegeli And Lee G. Dodge (2007) done the study related to fuel gas temperature more critical than inlet air temperature. The low-temperature ignition performance appeared to depend more on the effect of fuel temperature, i.e., the fuel viscosity, rather than the burner inlet air temperature. This observation has the important practical implication that heating the fuel or using a lower-viscosity fuel would be a much more efficient way to improve low-temperature ignition than heating the inlet air, a guideline particularly suited to combustors employing pressure-swirl atomisers.

Marin G.E et al. (2019), mentioned that the fuel gas composition affects the characteristics and mode of operation of the gas turbine. In particular, the methane content in the fuel gas composition has a decisive influence on the energy characteristics of the gas turbine and the performance of the entire installation. Therefore, it is important also to pay attention to the choice of fuel (its gas field) and its characteristics that may also change over time and, therefore, affect the various indicators of a particular installation or the entire equipment in the complex (for example. if the GTU is an integral part CCP). The different compositions of the fuel affect the efficiency of the gas turbine; therefore, it is important to consider the characteristics of the fuel to increase efficiency.

In addition to efficiency, the composition of the fuel affects emissions and, accordingly, the environment, which encourages the choice of fuel with less environmental damage. Most of the costs during the operation of a gas turbine are fuel costs, and if its consumption is reduced, then the cost of produced energy will decrease. Therefore, it is important to consider the ratio of the fuel price, its consumption, and composition, which affects the operation of gas turbines.

6.3 METHODOLOGY

Pre-processing, processing, and post-processing are the three general simulation tasks that make up any system's computational fluid dynamic study [8, 9]. The first step is pre-processing, which involves defining and creating the geometry and mesh for the heat exchanger domain. The processing step, which defines the problem's governing equations, materials, numerical models, and boundary conditions, comes next. After that, the solution computations are started using the specified setup. The computed results, including velocity and thermal profiles, can be automatically inspected and collected at the post-processing stage. The three-step procedure is repeated with different boundary conditions and process parameters to produce a set of findings that can be compared and examined.

6.3.1 Pre-processing

The drawing was done on Solidworks and exported to STEP file format before being imported into Ansys FLUENT. Ansys itself has a Design Modeler function that can also be used to sketch the model. The model developed for this study is shown in Fig. 6.2.

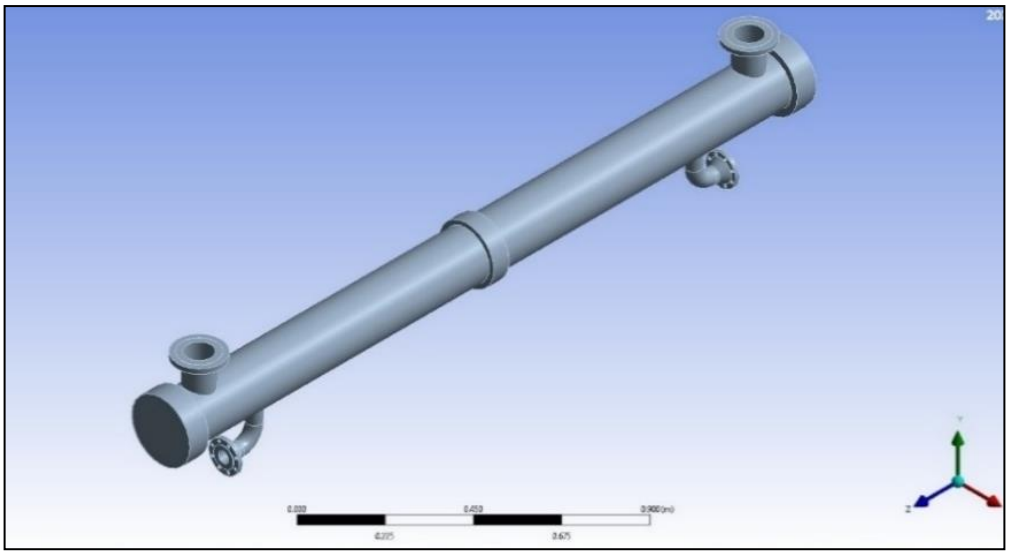

Fig. 6.2: Heat Exchanger Drawing Imported to Ansys Fluent

In a general Finite Element Method, reducing mesh element size improves analysis accuracy, but only up to a specific limit beyond which further refinement yields minimal benefits. Sizing options offer enhanced control over mesh characteristics, including the rate at which element size transitions between fine and coarse regions (mesh growth), refinement based on surface curvature and angles between adjacent element normal (curvature-based sizing), and the number of elements placed in narrow spaces between geometric features (proximity-based sizing).

The model generated 943,171 elements and 5,032,795 nodes in the initial meshing attempt. Multiple body sizing strategies were applied to enhance result accuracy, as outlined in Table 6.1, with the final selected body sizing set at 0.00392 m. This refinement resulted in 1,323,946 nodes and 6,761,541 elements, meeting key quality criteria: skewness ranging from less than 0.94 to a maximum of 0 and orthogonal quality ranging from above 0.15 to a maximum of 1.

Table 6.1: List of Elements and Nodes Before and After Body Sizing

Size	Nodes	Element	Skewness		Orthogonal Quality	
			Min	Max	Min	Max
Sizing/Limits			<0.94	0	>0.15	1
0	943171	5032795	0.9689	2.03E-07	0.031097	0.99943
0.01	939992	4983555	0.92	6E-08	0.08	0.99943
0.008	966196	5084896	0.9244	5.87E-07	0.0756	0.99943
0.007	989124	5177606	0.878	8.12E-07	0.12163	0.99943
0.006	1028565	5345707	0.8898	1.16E-07	0.1102	0.99943

0.004	1304353	6668122	6.31E-09	0.8526	0.1474	0.99943
0.0039	1329750	6788138	1.04E-09	0.85029	0.14971	0.99943
0.00395	1316232	6723675	1.65E-09	0.85155	0.14845	0.99943
0.00393	1322053	6753366	7.19E-09	0.85501	0.14499	0.99943
0.00392	1323946	6761541	0.84946	1.39E-09	0.15054	0.99943
0.0038	1357705	6924769	1.01E-09	0.84403	0.15597	0.99943
0.003	1781638	9089714	2.14E-10	0.82088	0.179	0.99943
0.00298	1801829	9197352	3.65E-10	0.8306	0.1694	0.99943

A comparison of mesh generated with or without the body sizing function activated is shown in Fig. 6.3.

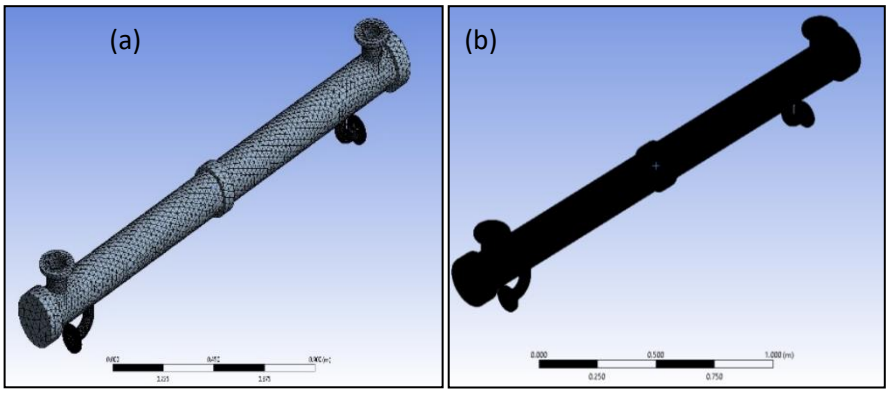

Fig. 6.3: Mesh generated for the heat exchanger model (a) without body sizing (b) with body sizing

6.3.2 Processing

A predetermined set of parameters was established, as shown in Table 6.2 below. These parameters were derived from actual data from a Singaporean power plant that is currently in operation. The following values were used to compare reference data from a running unit with a CFD simulation.

Table 6.2: Parameters Set on The First Trial Run or Base Run

Heat exchanger section	Velocity (m/s)	Pressure (Pa)	Temperature K
Water Inlet	5	2500000	488
Water Outlet	-	2300000	328
Natural gas	5	3200000	303
Natural gas	-	3100000	444

6.3.3 Post-processing

The post-processing phase determines the intended results of the simulations. ANSYS generates automated reporting for temperature and velocity variables after solution calculations. The overall heat transfer rate, shell exit temperature, and pressure drop are derived from the ANSYS Fluent findings. Alongside these post-processing data, temperature contours and flow profiles are established to facilitate the evaluation of the shell and tube heat exchanger .

6.4 RESULTS AND DISCUSSION

This study presents a numerical analysis of convection heat transfer over an isothermal porous flat plate, focusing on the effects of varying the velocities of natural gas and water across a series of simulations. Nine test cases (Series 1 to 9) were conducted with different gas and water velocity combinations. In Series 1 through Series 4, the gas velocity was incrementally increased from 1 m/s to 10 m/s, while the water velocity was maintained at 5 m/s. Series 5 evaluated the condition where gas and water velocities were set at 5 m/s. From Series 6 to Series 9, the gas velocity was constant at 5 m/s, while the water velocity was progressively reduced from 4 m/s to 1 m/s. The gas and water velocity series is shown in Table 6.3 below for easier understanding.

Table 6.3: The Values and Parameters That Were Used

Series	Velocity of Natural Gas (m/s)	Velocity of Water (m/s)
1	1	5
2	2	5
3	3	5
4	4	5
5	5	5
6	5	4
7	5	3
8	5	2
9	5	1

Temperature contours were generated for each configuration, shown in Fig. 6.4 to 6.12, revealing clear differences in heat transfer behaviour under varying flow conditions. As the gas velocity increased while maintaining a constant water flow (Series 1 to 4), a more uniform and enhanced temperature distribution was observed across the heat exchanger. This effect is attributed to the improved

mixing and momentum of the gas, which enhances convective heat transfer near the porous surface. In Series 5, where both fluid velocities were equal at 5 m/s, the contour exhibited a less favourable thermal profile, indicating reduced heat exchange efficiency, likely due to balanced convective forces that minimise the thermal gradient.

The most pronounced thermal performance was achieved in Series 9, where the gas velocity remained at 5 m/s, and the water velocity was minimised to 1 m/s. This combination resulted in the steepest temperature rise across the exchanger, as shown in Fig. 6.9. The dominance of gas flow momentum and the slower water stream created favourable conditions for extensive heat transfer, with greater residence time and thermal interaction.

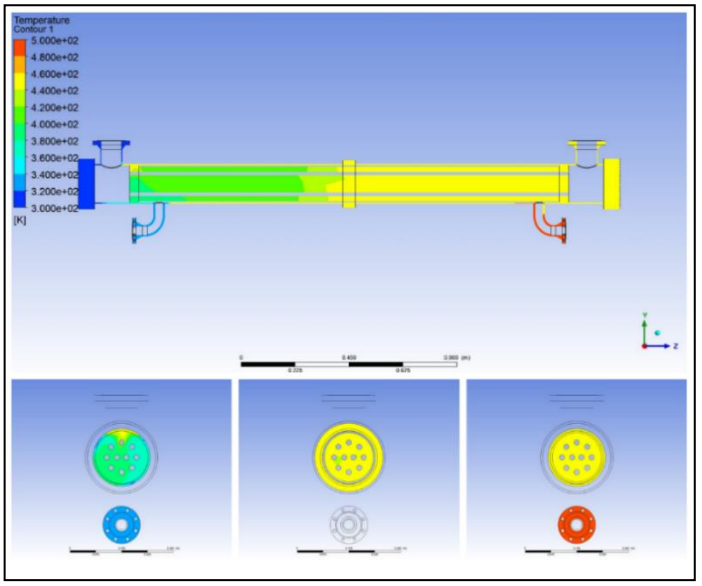

Fig. 6.0: Temperature contour for gas velocity 1 m/s and water 5 m/s (Series 1)

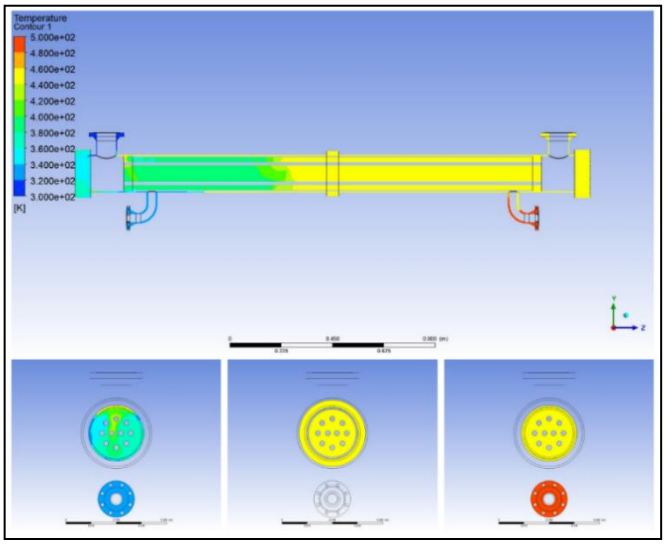

Fig. 6.5: Temperature contour for gas velocity 2 m/s and water 5 m/s (Series 2)

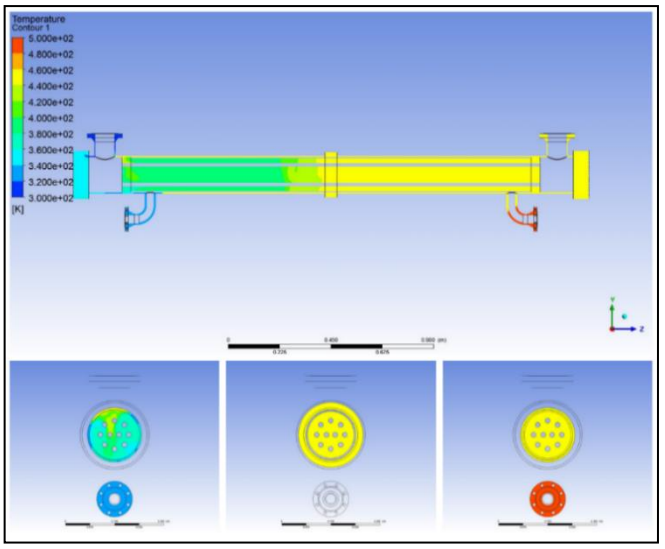

Fig. 6.6: Temperature contour for gas 3 m/s and water 5 m/s (Series 3)

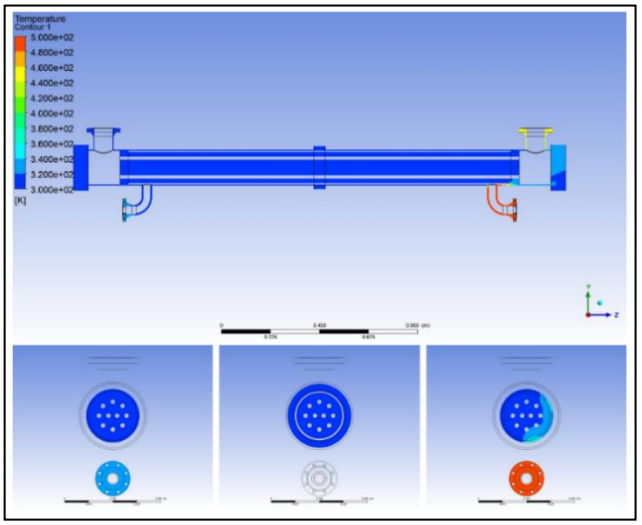

Fig. 6.7: Temperature contour for gas 4 m/s and water 5 m/s (Series 4)

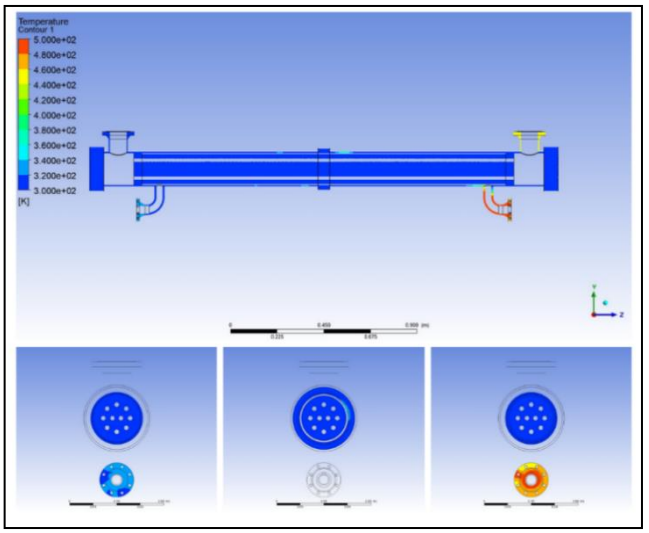

Fig. 6. 8: Temperature contour for gas 5 m/s and water 5 m/s (Series 5)

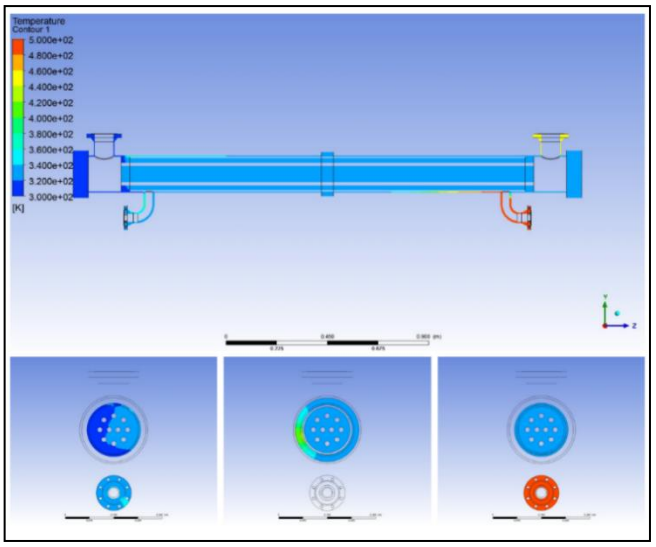

Fig. 6.9: Temperature contour for gas 5 m/s and water 4 m/s (Series 6)

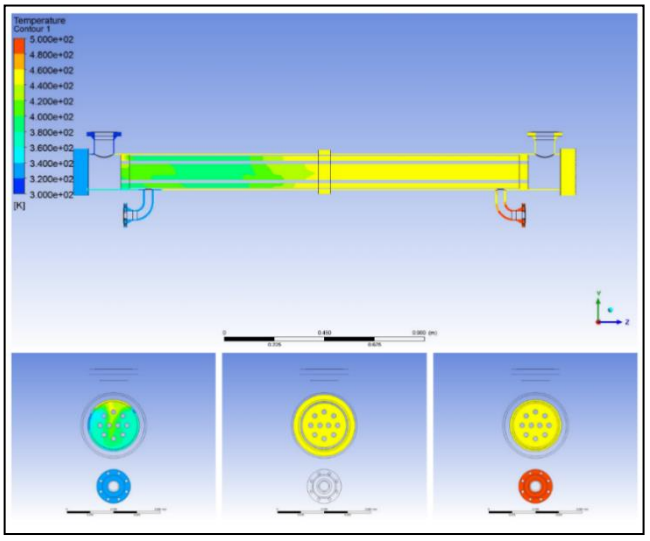

Fig. 6.10: Temperature contour for gas 5 m/s and water 3 m/s (Series 7)

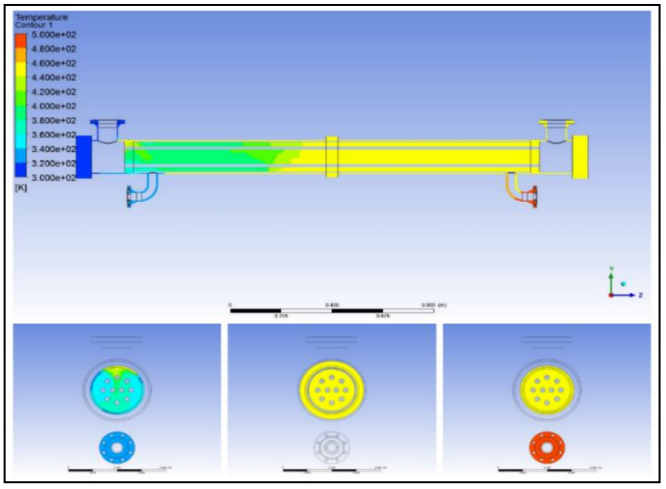

Fig. 6.11: Temperature contour for gas 5 m/s and water 2 m/s (Series 8)

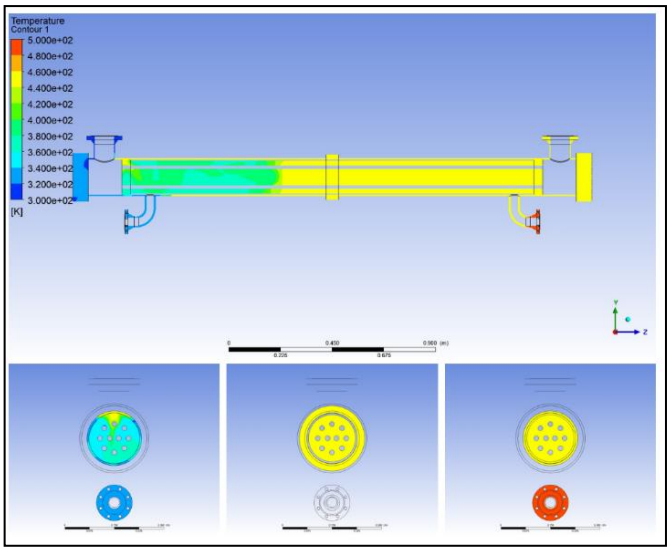

Fig. 6.12: Temperature contour for gas 5 m/s and water 1 m/s (Series 9)

Fig. 6.13 plots the temperature profile across the Z plane for all velocity settings to support this observation. The temperature increase is the most rapid in Series 9, confirming it as the most thermally efficient setup. Conversely, Series 5 demonstrates the weakest thermal performance due to symmetrical high-velocity flow conditions that limited the development of a strong thermal boundary layer. The numerical results show that the best heat transfer performance occurs when gas flow dominates, particularly in Series 9. In contrast, Series 5, 6, and 4 show the least favourable outcomes. This confirms that reducing water velocity while maintaining high gas velocity enhances thermal exchange, emphasising the importance of velocity control in optimising heat exchanger performance.

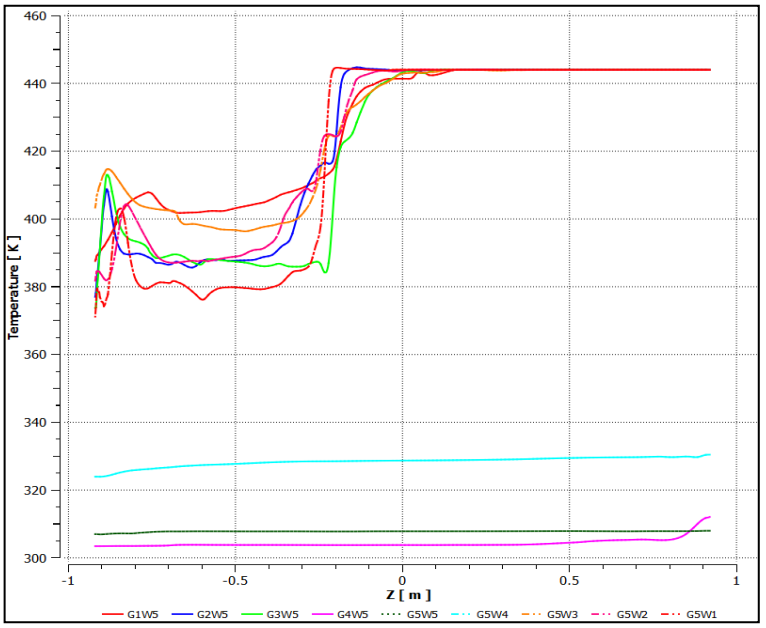

Fig. 6.13: Temperature Profile Across the Heat Exchanger for All Velocity Settings Combined

6.5 CONCLUSION AND RECOMMENDATIONS

This chapter demonstrates that optimising fluid velocities, specifically setting natural gas at 5 m/s and water at 1 m/s, significantly improves the thermal performance of a fuel gas preheater. This configuration delivered the highest temperature increase, optimal heat transfer characteristics, and substantial pressure drop across the shell side, indicating enhanced efficiency. Among the CFD methods applied, the k-epsilon turbulence model with a coupled solver yielded the most reliable and physically consistent result. These findings highlight the critical role of flow dynamics in heat exchanger design and offer practical insight for improving energy systems, particularly in gas turbine applications.

6.6 REFERENCE

D. Eckardt, *Gas turbine powerhouse: the development of the power generation gas turbine at BBC-ABB-Alstom*. Walter de Gruyter, 2013.

S. R. Burden, "Evaluation and application of CFD for turbine rim sealing," University of Surrey, 2024.

D. M. Godino, S. F. Corzo, N. M. Nigro, and D. E. Ramajo, "CFD simulation of the preheater of a nuclear facility steam generator using a thermal coupled model," *Nuclear Engineering and Design,* vol. 335, pp. 265-278, 2018.

H. K. Wiest and S. D. Heister, "Experimental study of gas turbine combustion with elevated fuel temperatures," *Journal of Engineering for Gas Turbines and Power,* vol. 136, no. 12, p. 121507, 2014.

O. Darbyshire, C. Wilson, A. Evans, and S. Beck, "CFD based analysis of burner fuel air mixing over a range of air inlet and fuel preheat temperatures for a Siemens v94. 3A gas turbine burner," in *Turbo Expo: Power for Land, Sea, and Air*, 2006, vol. 42363, pp. 709-714.

D. W. Naegeli and L. G. DODGE, "Ignition study in a gas turbine combustor," *Combustion science and technology,* vol. 80, no. 4-6, pp. 165-184, 1991.

G. Marin, D. Mendeleev, and A. Akhmetshin, "Analysis of changes in the thermophysical parameters of the gas turbine unit working fluid depending on the fuel gas composition," in *2019 International Multi-Conference on Industrial Engineering and Modern Technologies (FarEastCon)*, 2019: IEEE, pp. 1-4.

B. Andersson, R. Andersson, L. Håkansson, M. Mortensen, R. Sudiyo, and B. Van Wachem, *Computational fluid dynamics for engineers*. Cambridge university press, 2011.

J. T. Masilungan-Manuel, M. C. E. Manuel, P. T. Lin, and A. N. Soriano, "Optimisation of the drying parameters for the short-form spray dryer producing powdered egg with 20% tapioca starch additive," *Advances in Mechanical Engineering,* vol. 7, no. 9, p. 1687814015602603, 2015.

P. A. D. Cruz, E.-J. E. Yamat, J. P. E. Nuqui, and A. N. Soriano, "Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger," *Digital Chemical Engineering,* vol. 3, p. 100014, 2022/06/01/ 2022.