

Simulation of shell and tube heat exchanger

¹Isah, I., ²Bonet, M. U., ³Mohammed, S. U. and ⁴Muhammad, M. M.

¹Centre for Atmospheric research, National Space Research and Development Agency, Prince Abubakar Audu University Campus, Ayingba, Nigeria

²Department of Aerospace Engineering, Airforce Training Institute of Technology Kaduna, Nigeria

³Department of Mechanical Engineering, Nigerian Defence Academy, Kaduna, Nigeria

⁴Department of Mechanical engineering, Federal University of Technology, Minna, Nigeria

mubonet@nda.edu.ng; sumuhammed@nda.edu.ng; masin.muhammadu@futminna.edu.ng

Paper History

Received: 01st October, 2025

Accepted: 15th October, 2025

Published: October, 2025

Abstract:

This study presents a numerical simulation of a shell and tube heat exchanger using ANSYS software to investigate its thermal and fluid dynamics performance in a power plant setting. The simulation predicts key aspects of the heat exchanger's behaviour, including temperature distribution, fluid flow patterns, pressure drop, and heat transfer rates. A counter-flow configuration is considered, with cold water flowing through the tubes and hot exhaust flue gases flowing through the shell. The ANSYS Fluent Computational Fluid Dynamics (CFD) tool was employed for simulating the heat exchanger, involving geometry creation in Solid Works, meshing, defining physics, applying boundary conditions, configuring solver settings, and post-processing results. Simulation results illustrated the heat exchanger's performance behaviours such as velocity and pressure flow trajectories, temperature distributions, and pressure cut plots. Also, the results of the simulation were in good agreement with literature values, validating the model's accuracy. Key results include temperature gradients for hot and cold fluids (813 K to 373 K for hot fluid; 288 K to 423 K for cold fluid), heat transfer rate of 4012.8 kW, overall heat transfer coefficient of 16 W/m²K, tube-side heat transfer coefficient of 1986.23 W/m²K, and pressure drops of 5.34 kPa (tube side) and 0.22 kPa (shell side). The study concludes that ANSYS software produces realistic outputs for heat transfer and fluid flow in shell and tube heat exchangers, allowing for model validation and revision before prototype production.

Corresponding author

Isah, I.

informisah@yahoo.com

Keywords: Fluid, Heat exchanger, Gas turbine, Shell, Tube, Temperature

1. Introduction

Heat exchangers are sophisticated devices which are designed to facilitate the efficient transfer of heat energy between a solid object and a fluid or between two or more fluids. These fluids can be separated by a solid wall to prevent them mixing or be in direct contact (Incropera, 2010). The versatility of heat exchangers makes them indispensable in a wide range of industries and applications such as space heating, refrigeration, air conditioning, power generation, chemical processing and many more. Their ability to optimize energy use and improve system efficiency has made them a crucial component in modern industrial processes (Cengel and Boles, 2008).

A shell and tube heat exchanger is a type of heat exchanger that has gained widespread acceptance and application in various industries, particularly in power plants, oil and refineries and large scale chemical plants. The design of this type of heat exchanger consists of a large cylindrical shell that houses a bundle of tubes, allowing one fluid to flow through the tubes while another

flows over the tubes through the shell. This configuration enables efficient heat transfer between the two fluids, making it an ideal solution for applications where high-pressure and high-temperature conditions are prevalent. The shell and tube design's robustness and reliability have made it a preferred choice for many industrial applications (Shah and Sekulic, 2003).

In a power generation plant, a shell and tube heat exchanger plays a vital role in enhancing the plant's efficiency and output. Specifically, the heat exchanger is used to recover heat from the high-temperature exhaust gases exiting the gas turbine. The heat exchanger uses these exhaust gases to heat up water, generating steam that is then used to drive a steam turbine, producing additional power. The combined cycle configuration enables the power plant to maximize energy extraction from the fuel, increasing overall plant efficiency (Kehlhofer, *et al.*, 2009).

The primary objective of this work is to conduct simulation of a shell and tube heat exchanger using ANSYS to investigate its thermal and fluid dynamics

performance. This simulation is aimed to predict various aspects of the heat exchanger's behaviour, including temperature distribution, fluid flow patterns, pressure drop and heat transfer rates within the heat exchanger. Through simulation of heat exchanger, significant understanding of its behaviour under various operating conditions can be obtained, thereby guiding design choices and strategies (Kakac and Liu, (2002).

The simulation of shell and tube heat exchangers has been extensively studied. Prithviraj and Andrews (2007) developed a 3D numerical model (HEATX) focusing on fluid mechanics in shell-and-tube heat exchangers, using volumetric porosities, surface permeabilities, and a modified k- ϵ turbulence model. Their model was validated against Argonne National Labs' experimental pressure drop data, showing good agreement, and they examined baffle cut and spacing effects on pressure drop. Correa and Marchetti (2007) created a multicell dynamic model for simulating transient operations of shell-and-tube heat exchangers, useful for design, transient analysis, and control system design, allowing time-dependent inlet conditions and disturbance simulations. Ozden and Tari (2010) performed CFD analysis on a shell-and-tube heat exchanger's shell-side, studying baffle spacing, baffle cut, and shell diameter impacts on heat transfer and pressure drop; their results were sensitive to turbulence model choice and validated against the Bell-Delaware method. Ambekar, et al. (2016) studied shell-and-tube heat exchangers with various baffle configurations using SOLIDWORKS Flow Simulation, finding single segmental baffles showed maximum heat transfer coefficient and pressure drop, while helical baffles had minimal stagnation zones suggesting reduced fouling and vibration.

Zhang (2009) presented 3D numerical simulations of shell-and-tube heat exchangers with middle-overlapped helical baffles and continuous baffles using GAMBIT 2.3 and FLUENT 6.3; validation showed good agreement for pressure drop and Nusselt number, and a periodic model efficiently captured heat transfer and pressure drop characteristics. Darbandi, et al. (2021) proposed a semi-full-scale CFD approach for shell-and-tube heat exchangers to reduce computational demands, simulating a smallest repetitive "block" and generating correlations for analysis/design, showing great agreement with experimental data. Ammar, et al. (2013) investigated shell-side heat transfer and pressure drop in twisted oval tube heat exchangers via 3D numerical simulation focusing on twisted pitch length and aspect ratio, with results showing Nusselt number and friction factor trends with geometrical parameters. Abd, et al. (2018) conducted a parametric study on shell-and-tube heat exchanger performance examining shell diameter, tube length, baffle spacing, and cutting space impacts, with key findings including effects on heat transfer coefficients and pressure drops for various configurations. Ali and Zaki (2017) developed a new correlation for predicting Nusselt number for the tube side of a shell-and-tube heat exchanger, reducing error compared to Kern's method based on simulation data for Clough Company. Pamuk (2020) simulated a shell-and-tube heat exchanger using ANSYS Fluent CFD software capturing physical phenomena like heat transfer,

temperature, velocity, and pressure distributions, allowing design without prototypes. Chen, et al. (2020) optimized a compact heat exchanger for waste heat recovery using CFD and response surface methodology achieving improved heat transfer and reduced pressure drop. Rao, et al. (2020) presented a design and optimization methodology for a heat exchanger used for waste heat recovery from a gas turbine using a combination of CFD analysis with ANSYS Fluent and GA optimization with MATLAB; their results showed the optimized design resulted in significant heat transfer rate improvement and pressure drop reduction. Ali et al. (2018) conducted a comprehensive parametric study on shell-and-tube heat exchangers investigating key design variables enhancing heat transfer efficiency and performance; they found increasing shell diameter with triangular pitch and pull-through floating head led to a 3% increase in heat transfer coefficient for a 0.05 m shell diameter increase. Isiaka et al. (2025) conducted parametric analysis of a shell-and-tube heat exchanger for a simple cycle gas turbine power plant indicating moderate heat transfer performance and relatively low and moderate shell and tube sides pressure drop, providing insights for design and optimization.

Santha, et al. (2020) analysed a shell-and-tube heat exchanger for waste heat recovery using ANSYS Fluent 15.0 CFD reporting on effectiveness, waste heat recovery, and logarithmic mean temperature distribution. Chen, et al. (2022) developed a flexible design approach for shell-and-tube heat exchangers using genetic algorithms with constraints and flexibility index showing improved design operability and lower total cost. Natesan and Karinka (2023) conducted a study on graphene-based thermal enhancement showing graphene's high thermal conductivity and stability make it suitable for heat transfer applications. Pontevedra (2018) studied shell-and-tube heat exchangers with different baffles finding increased baffle complexity can enhance heat transfer but lead to higher pressure drops. Silaipillayarputhur and Khurshid (2019) presented a comprehensive review of the design strategies for optimizing shell and tube heat exchangers. The paper considered a review for the design of a shell and tube heat exchanger. using analytical techniques such as log mean temperature difference (LMTD) and effectiveness-number of transfer units (ϵ -NTU). Franti and Martin (2017) presented the design process for shell and tube heat exchangers through systematic approach to optimize the heat transfer area for efficient thermal performance using the design calculation of U-tube heat exchanger (precisely 1-2 shell and tube heat exchanger with U-tubes). Perumal, et al. (2022) presented heat transfer analysis in counter flow shell and tube heat exchanger using design of experiments. The study used the Taguchi L9 orthogonal array execution to find the overall heat transfer coefficient with effective process parameters considering three major parameters which are coil diameter (25 mm, 30 mm, and 35 mm), baffle thickness (15 mm, 20 mm, and 25 mm) and baffle gap (200 mm, 300 mm, and 400 mm) with baffle plate thickness being a highly significant factor for the experiment. Mulyana, et al. (2011) presented four types of heat exchanger identification using ARX Model. Heat

exchangers are devices built for efficient heat transfer from one fluid to another and widely used in engineering process. The four heat exchangers consist of shell and tube, spiral, concentric (double pipe), and plate heat exchangers available in Univeriti Tun Hussein Onn Malaysia (UTHM) Laboratory.

Baina, *et al.* (2018) conducted analysis of a high-temperature heat exchanger for an externally-fired micro gas turbine. The work compares the performance of a corrugated plate heat exchanger and a two-tube-passes shell and tube heat exchanger considering the effect of thickness of deposit material with different thermal conductivities on pressure drop and effectiveness. Performance simulation was carried out using MATLAB and Aspen Plus software and the results obtained show that the effectiveness of the corrugated plate heat exchanger is more influenced at larger thicknesses of deposit materials than the two-tube-passes shell and tube heat exchanger. Lovaglio, *et al.* (2018) presented paper on the heat exchanger design and optimization using generic algorithm for externally fired micro-turbine. The study discussed a new configuration in which syngas which was a product of downdraft gasifier was externally fired by air turbine. MATLAB software was used to build code and modeFRONTIER was used for geometrical optimization through multi-objective maximize the overall heat transfer coefficient and minimize both costs and pressure drops across the equipment. You, *et al.* (2023) presented paper on performance of a high-temperature particle-based shell and tube crossflow heat exchanger suitable for CSP power generation application. The results obtained show high values of the overall heat transfer coefficient (up to $\sim 120 \text{ W/m}^2\cdot^\circ\text{C}$) in accordance with shell-and-tube indirect contact heat exchangers. Kara (2004) presented a computer program for designing shell-and-tube heat exchangers. Computer codes for design are organized to vary systematically the exchanger parameters such as, shell diameter, baffle spacing, number of tube-side pass to identify configurations that satisfy the specified heat transfer and pressure drops. Findings from the program were respectively overall shell dimensions, tube bundle dimensions, optimum heat transfer surface area and minimum or allowable shell-side pressure drop.

Hadidi, *et al.* (2013) presented a new design approach for shell-and-tube heat exchangers using imperialist competitive algorithm (ICA) from economic point of view. The results of the study show that a comprehensive computer code was developed to implement the proposed method and test cases demonstrated the algorithm's effectiveness and accuracy. Wang, *et al.* (2009) conducted an experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger. The study was a novel approach to enhancing heat transfer in shell-and-tube heat exchangers involves installing sealers in the shell-side. Finding from the study conducted showed heat transfer coefficient through shell-side coefficient increased by 18.2-25.5%, overall heat transfer coefficient increased by 15.6-19.7%, exergy efficiency increased by 12.9-14.1%, pressure losses increased by 44.6-48.8%, but with negligible impact on required pump power. Chukwudi and Ogunedo (2018)

presented study on design and construction of a shell and tube heat exchanger. The results of the study demonstrate effective heat transfer and acceptable pressure drops. Thet, *et al.* (2008) presented paper on heat exchanger design with the aim to design an oil cooler (heat exchanger), specifically a shell-and-tube heat exchanger, which is the most common type of liquid-to-liquid heat exchanger. MATLAB and AutoCAD software were used for design calculations and finding from the study included discussion of fundamental heat transfer concepts and complex relationships and computer program for designing and modifying shell-and-tube heat exchangers.

Wang, *et al.* (2018) conducted numerical study to investigate the effects of shell diameter on heat transfer performance in the shell-and-tube heat exchangers. The researchers utilized computational fluid dynamics (CFD) to simulate the flow and heat transfer in the heat exchanger. The shell diameter was varied and its impact on the heat transfer coefficient, pressure drop, and overall heat transfer performance examined. The results from the study show that the heat transfer coefficient increases with increasing shell diameter, the pressure drop decreases with increasing shell diameter and the overall heat transfer performance is optimized at a shell diameter of 0.3 to 0.4m.

Mishra and Das (2017) investigated the effect of mass flow rate on the performance of a shell and tube heat exchanger employing numerical model for simulation of heat transfer and flow of fluid in the heat exchanger. The results obtained show increasing the mass flow rate enhances the heat transfer coefficient, but also increases the pressure drop. Khan, *et al.* (2020) conducted analysis on heat transfer of a pipe-in-pipe heat exchanger with a focus on thermal performance and pressure drop characteristics. The research used experimentation and numerical methods to analyse heat transfer in pipe-in-pipe heat exchangers. Wang, *et al.* (2019) explored heat transfer rates and velocity distributions in various engineering applications focusing on convective heat transfer enhancement. The results of the study show an achievement of improved heat transfer rates and enhancement of convective heat transfer in various engineering applications. Gao and Chen (2018) conducted numerical investigation of flow and heat transfer in a shell-and-tube heat exchanger. The study investigated the velocity flow trajectory and transfer characteristics in a shell-and-tube heat exchanger using simulations. The finding from the study show that optimizing the tube arrangement and baffle design can significantly enhance heat transfer rates.

The present paper will conduct ANSYS simulation of shell and tube heat exchanger for a simple cycle gas turbine power plant.

2. Methodology

The input data for the study was based on assumptions. However, the hot fluid inlet temperature from the gas turbine exhaust waste was based on the exhaust outlet temperature of SGT5-2000E gas turbine model located in Geregu gas turbine power plant in Kogi State as

shown in Figure 1. Similarly, the Figure 2 shows the shell and tube heat exchange considered for the study.

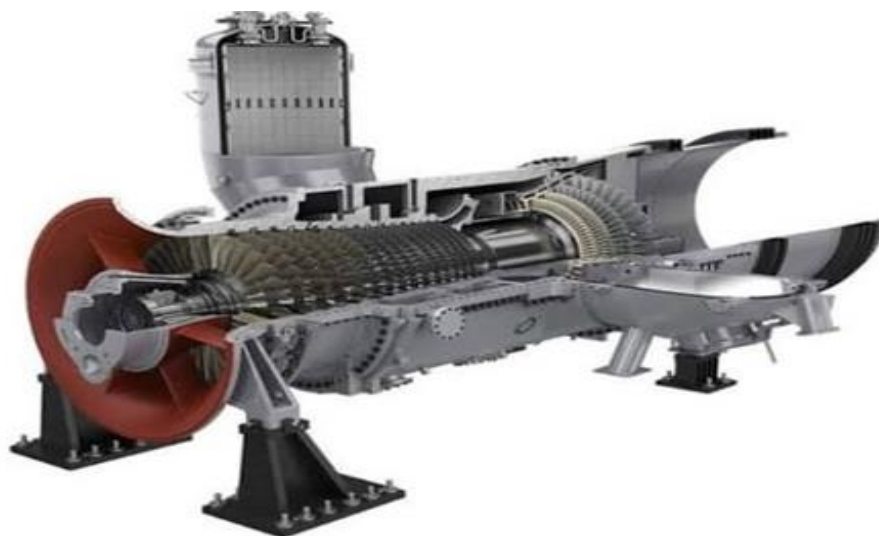


Figure 1: Siemens SGT5-2000E Gas Turbine Model

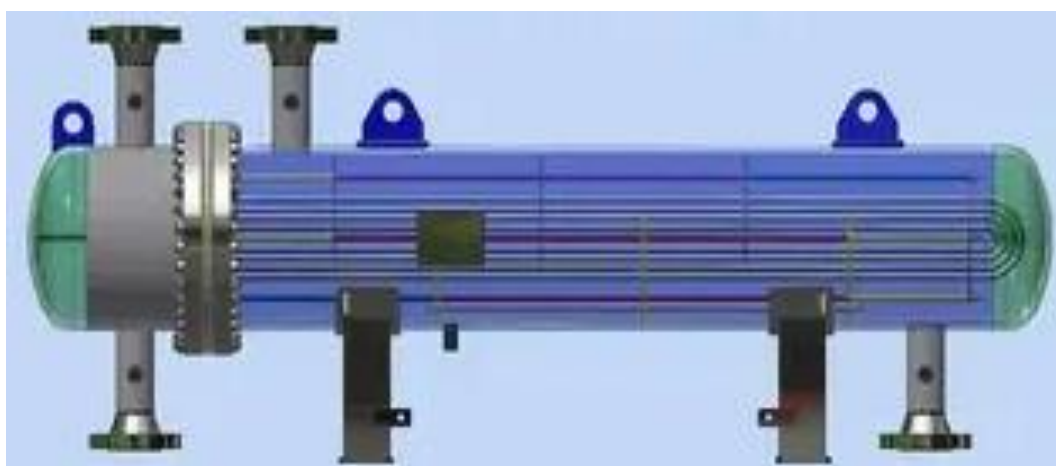


Figure 2: Shell and tube heat exchanger MechStudies (MechStudies, 2025)

The present study starts with ANSYS simulation of counter-flow shell and tube heat exchanger in order to predict the temperature distribution, fluid flow patterns, pressure drops and heat transfer within the heat exchanger. The cold water has been used as the tube side fluid and exhaust flue gases as the shell side fluid for heat transfer analysis. The hot fluid in the shell transfers heat to the cold fluid (water) that flow through the tubes.

ANSYS simulation provides flexibility to change design parameters without the use of hardware which may be regarded expensive. Therefore, it costs less to simulate instead of carrying out laboratory and experimental analyses. Cycle's design time and cost are also reduced by optimization through computer predictions so as to provide high level of confidence in prototype performance. Considering the above discussion, the present study has been undertaken to achieve objectives such as prediction of temperature distribution, fluid flow patterns, pressure drops and hat transfer within the heat exchanger.

The various mechanical and thermal design methods utilized for the study using relevant equations are as follows:

2.1 Mechanical parameters design

The mechanical parameters used in the design of the heat exchanger include:

Tube Pitch (t_p) was calculated using equation 1 (Chukwudi and Ogunedo, 2018)

$$t_p = 1.25 d_o \quad (1)$$

Baffle spacing (B) was calculated using equation 2:

$$B = 74 d_o^{0.75} \quad (2)$$

Number of Tube (N_t) was calculated using equation 3 (Sekulic, 2003)

$$N_t = \frac{CTP \pi D_{s,i}^2}{4 A_1} \quad (3)$$

Where: N_t = number of tubes, CTP = tube counting factor (assume 0.87 for 60° triangular layout) ("Standard of the Tubular Exchange Manufacturers Association" (TEMA) 10th Edition., 2019), $D_{s,i}$ = shell inside diameter (m), A_1 = cross sectional area (m²)

Tube surface area (A_o) was calculated using equation 4 (Chukwudi and Ogunedo, 2018):

$$A_o = \pi d_o N_t L \quad (4)$$

Where A_o = tube surface area (m^2), d_o = tube outer diameter (m), N_t = number of tubes, L = tube length (m)

Cross-sectional area of shell (m^2) was calculated using equation 5 as reported in ASME ("ASME Boiler and Pressure Vessel Code", Section VIII, Division 1, (Appendix A), 2019).

$$A_s = (\pi \times D_{s,i} \times B) - (N_t \times \pi \frac{d_o^2}{4}) \quad (5)$$

Where A_s = cross-sectional area of shell (m^2), $D_{s,i}$ = shell inside diameter (m), B = baffle spacing (m), N_t = number of tubes, d_o = tube outside diameter (m) and N_t = number of tube.

Shell outside diameter ($D_{s,o}$) was calculated using equation 6 as reported in Chukwudi and Ogunedo (Chukwudi and Ogunedo, 2018).

$$D_{s,o} = D_b + SBC \quad (6)$$

Where $D_{s,o}$ = shell outside diameter (m), D_b = bundle equivalent diameter (m) and SBC = shell bundle clearance

$$D_b = d_o \left(\frac{N_t}{K_1} \right)^{\frac{1}{n_1}} \quad (7)$$

Where $n_1 = 2.141$ and $K_1 = 0.319$ according to Chukwudi and Ogunedo (Chukwudi and Ogunedo, 2018)

Shell bundle clearance was calculated using equation 8

$$SBC = \frac{(D_{s,i} - d_o \text{ tb})}{2} \quad (8)$$

Where $d_o \text{ tb}$ = tube bundle outer diameter (m) which was calculated using equation 9.

$$d_o \text{ tb} = \frac{d_o + 2 \times (S - d_o)}{2} \quad (9)$$

Where S = tube pitch (t_p)

2.2 Thermodynamic design parameters

2.2.1 Overall heat transfer coefficient (U_o)

The overall heat transfer coefficient is the function of all the fouling resistance, individual heat coefficient and surface efficiency of the tubes. The fouling resistance are caused do to dirt's build-up on the heat exchanger surfaces and the equation to determine overall heat transfer coefficient is given in equation 10.

$$\frac{1}{U} = \frac{1}{\frac{1}{h_t} + R_{di} + R_w + R_{do} + \frac{1}{h_s}} \quad (10)$$

Where U = overall heat transfer coefficient (W/m^2K), h_t = tube-side heat transfer coefficient (W/m^2K), h_i = inside fouling resistance = 0.0001 (Chukwudi and Ogunedo, 2018) (m^2K/W), h_w = wall resistance (negligible because thin tube wall was considered) (m^2K/W), h_o = outside fouling resistance = 0.0002

(Chukwudi and Ogunedo, 2018) (m^2K/W), h_s = shell-side heat transfer coefficient (W/m^2K)

2.2.2 Heat Transfer Rate (Q)

Heat transfer rate in the heat exchanger according to (Lovaglio et al., 2018) is given as shown in equation 11:

$$q = m_h c_{p,h} (T_{h,i} - T_{h,o}) = m_c c_{p,c} (T_{c,i} - T_{c,o}) \quad (11)$$

Where m_h = mass flow of hot fluid, $c_{p,h}$ = specific heat capacity of hot fluid, $T_{h,i}$ = Temperature inlet of hot fluid, $T_{h,o}$ = Temperature outlet of hot fluid, m_c = mass flow of cold fluid, $c_{p,c}$ = specific heat capacity of cold fluid, $T_{c,i}$ = Temperature inlet of cold fluid and $T_{c,o}$ = Temperature outlet of cold fluid.

2.2.3 Energy balance in the heat exchanger

Heat loss by hot fluid = Heat gain by cold fluid which was calculated using equation 12.

$$q = m_h c_{p,h} (T_{h,i} - T_{h,o}) \quad (12)$$

2.2.4 Heat transfer area (A)

The heat transfer area (A) was determined using equation 13 as reported in (P, 2010)

$$A = \frac{Q}{U \Delta T_{LMTD}} \quad (13)$$

Where A = heat transfer area (m^2), Q = heat transfer rate (kW), U = overall heat transfer coefficient (W/m^2K) and ΔT_{LMTD} = logarithm mean temperature difference (K)

Logarithm mean temperature difference (ΔT_{LMTD}) was calculated using equation 14

$$\Delta T_{LMTD} = \frac{(T_{h,i} - T_{c,o}) - (T_{h,o} - T_{c,i})}{\ln \left(\frac{T_{h,i} - T_{c,o}}{T_{h,o} - T_{c,i}} \right)} \quad (14)$$

The mass flow rate of liquid can also be obtained from equation 15 to 17:

$$q = m_h c_{p,h} (T_{h,i} - T_{h,o}) = m_c c_{p,c} (T_{c,i} - T_{c,o}) \quad (15)$$

$$q = m_h c_{p,h} (T_{h,i} - T_{h,o}) = m_l (h_{c,o} - h_{c,i}) \quad (16)$$

$$m_l = \frac{m_h c_{p,h} (T_{h,i} - T_{h,o})}{(h_{c,o} - h_{c,i})} \quad (17)$$

2.2.5 Tube-side heat transfer coefficient

The tube-side Reynolds number depends on fluid properties such as fluid velocity, viscosity and characteristics diameter. Therefore, to calculate the tube-side heat transfer coefficient of the shell and tube heat exchanger, Dittus-Boelter correlation was used as shown in equation 18:

$$h_t = 0.023 \times \left(\frac{K}{d_o} \right) \times Re_t^{0.8} \times Pr_t^{0.3} \quad (18)$$

Where K = thermal conductivity of tube side fluid (W/m^2K), d_o = diameter of cold fluid (m), Re_t = tube-side Reynolds number (dimensionless) and Pr_t = tube-side Prandtl number (dimensionless)

2.2.6 Shell-Side heat transfer coefficient (h_s)

The shell-side heat transfer coefficient (h_s) for the heat exchanger can be calculated using the Bell-Delaware method in equation 19.

$$h_s = \frac{K}{D_h} \times 0.36 \times Re_s^{0.55} \times Pr_s^{0.333} \quad (19)$$

Where K = thermal conductivity, D_h = hydraulic diameter of shell (m), Re_s = Reynolds number of shell (dimensionless) and Pr_s = Prandtl number of shell (dimensionless)

2.2.7 Tube side pressure drop (ΔP_t)

The tube side pressure drop calculated using equation 20 as reported in (Chukwudi and Ogunedo, 2018)

$$\Delta P_t = f_t \times \left(\frac{L}{d}\right) \times \left(\rho \times \frac{V^2}{2}\right) \quad (20)$$

Where, ΔP_t = tube side pressure drop (kPa), f_t = tube side friction factor (dimensionless), L = tube length (m), d = tube inner diameter (m), ρ = density of fluid (kg/m³) and V = fluid velocity (m/s)

Friction factor f_t is a dimensionless quantity which represent ratio of shear stress at the tube wall to kinetic energy of the fluid per unit volume. It is therefore the fluid resistance to flow due to friction between fluid and the tube. Friction factor is calculated using the equation 21 as reported in (Kern, 1950).

$$f_t = 0.023 \times Re_t^{-0.2} \quad (21)$$

2.2.8 Shell side pressure drop (ΔP_s)

The shell side pressure drop was calculated using equation 22 as reported in (Chukwudi and Ogunedo, 2018)

$$\Delta P_s = f_s \times \left(\frac{D_s}{D_e}\right) \times \left(\rho \times \frac{V_s^2}{2}\right) \quad (22)$$

Where, ΔP_s = shell side pressure drop (Pa), f_s = shell side friction factor (dimensionless), D_s = shell diameter (m), D_e = equivalent or hydraulic diameter (D_h) (m), ρ = density of fluid (kg/m³) and V_s = shell side fluid velocity (m/s)

Thereafter, boundary conditions were defined and they include inlet/outlet temperature, pressure and mass flow rate of the fluids. The respective inlet/outlet temperatures and mass flow rate of the fluids are shown in Table 1.

Table 1: Boundary Conditions of Fluids

Parameters	Values	Units
Inlet temperature of hot fluid	813	K
Outlet temperature of hot fluid	373	K
Inlet temperature of cold fluid	288	K
Outlet temperature of cold fluid	423	K
Mass flow rate at shell side	8	kg/s
Mass flow rate at tube side	1.7	kg/s
Velocity at shell side	6.12	m/s
Velocity at tube side	5.4	m/s

3. Result and discussion

3.1 Simulation using ANSYS software

The ANSYS Fluent which is a Computational Fluid Dynamic (CFD) tool was used for simulating the shell and tube heat exchanger. The simulation method involves developing a 3-D geometry or model of heat exchanger using CAD software (Solidworks), shown in Figure 3.

The simulation of the shell and tube heat exchanger in this research was done to optimize the performance of the heat exchanger.

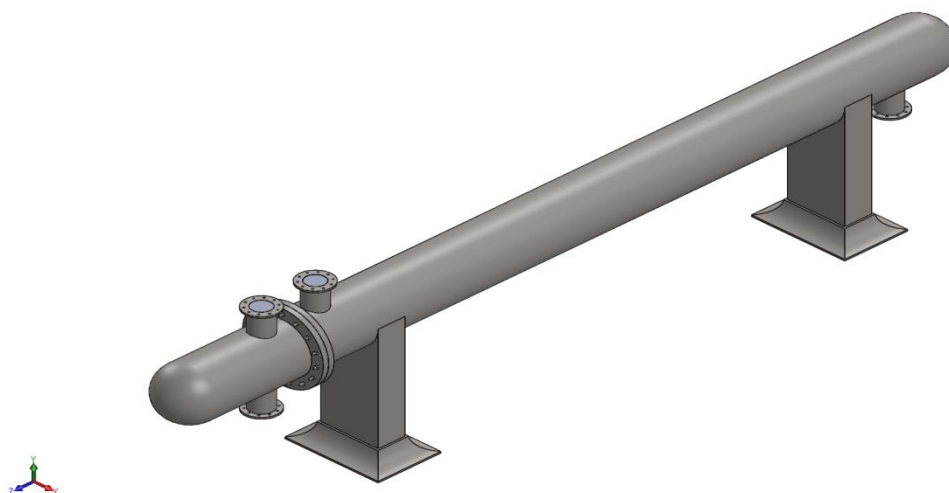


Figure 3: Designed Shell and Tube Heat Exchanger Geometry

The analysis involved simulating the heat exchanger to investigate its thermal and fluid dynamics performance. This simulation aimed to predict various aspects of the heat exchanger's behavior, including temperature distribution, fluid flow patterns, pressure drop and heat transfer rates within the heat exchanger. By simulating the heat exchanger, valuable insights into its performance

under different operating conditions which can inform design decisions and optimization strategies can be gained. The simulation of the shell and tube heat exchanger involved the following key steps.

After developing a 3-D model of the shell and tube heat exchanger in SolidWorks, the model was meshed by dividing it into smaller units, cells or members allowing for

more accurate calculations and also to create a mesh that can capture the desired level of details, with mesh size and type of between 0.01 - 0.04 m and Tetrahedron being

utilized as shown in Figures 4 to 7. The design data of the SGT5-2000E gas turbine are shown in Table 2.

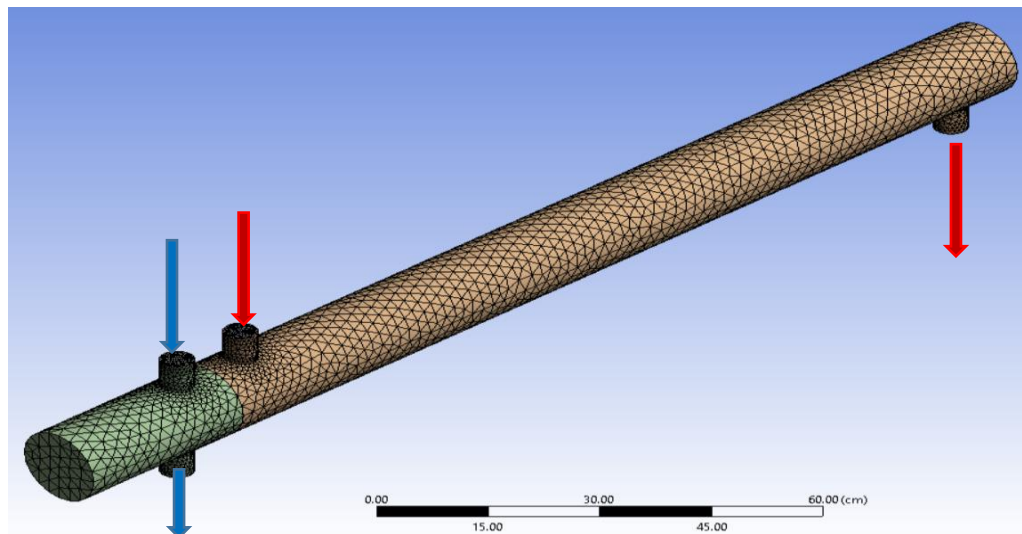


Figure 4: Shell Wall Mesh

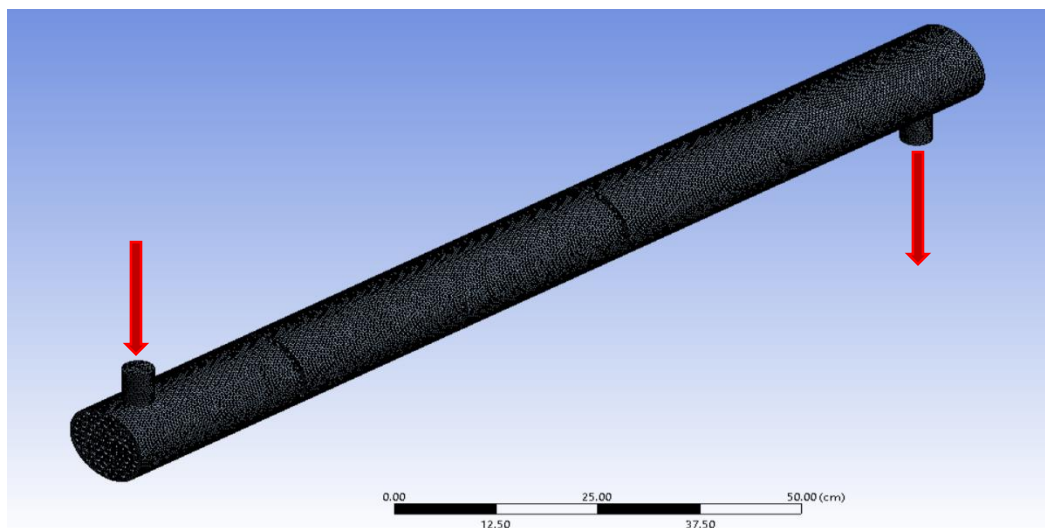


Figure 5: Shell Domain Mesh

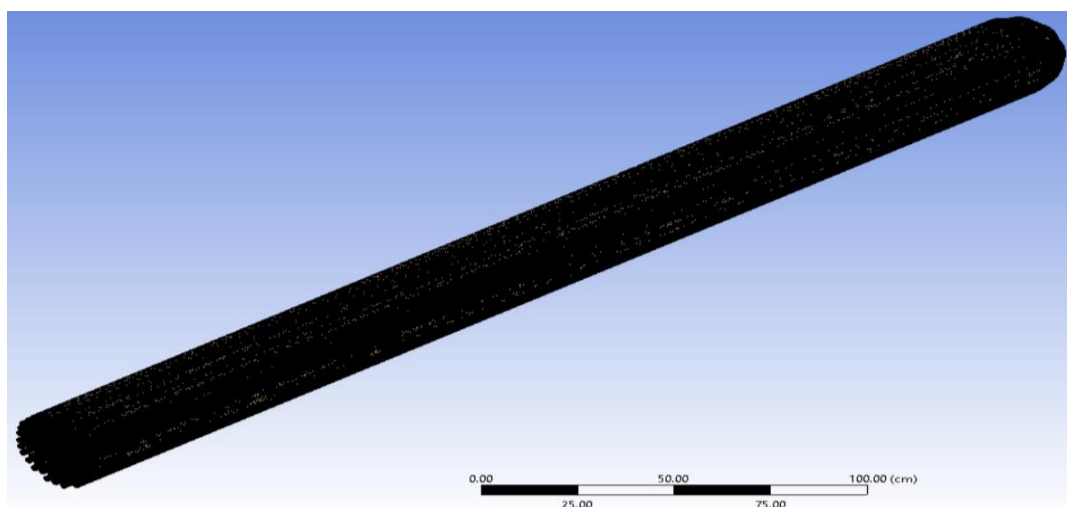


Figure 6: Tube Wall Mesh

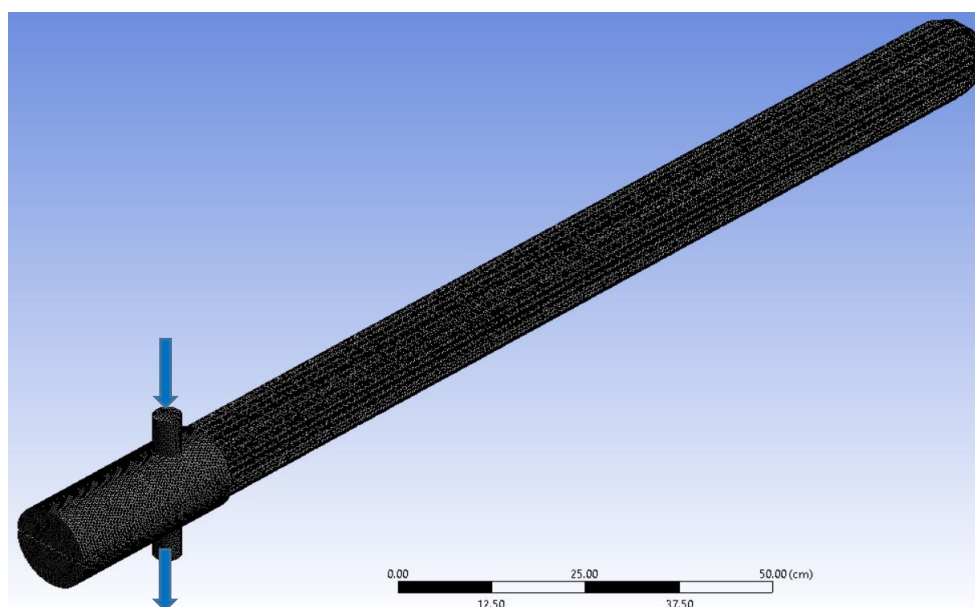


Figure 7: Tube Domain Mesh

Table 2: Results of Heat Exchanger Analysis

Parameter	Results of Heat Exchanger Analysis	
	Value	Unit
Mass flow rate at shell side	8	Kg/s
Mass flow rate at tube side	1.7	Kg/s
Velocity at shell side	6.12	m/s
Velocity at tube side	5.4	m/s
Enthalpy of fluid at temperature of 423 K	2776.2	kJ/kg
Enthalpy of fluid at temperature of 373 K	417.46	kJ/kg
Heat transfer rate, (q)	4012.8	kW
Heat transfer surface area	0.60	m ²
Number of tubes	130	
Shell diameter	0.3	m
Tube pitch	0.025	m
Tube surface area, (A _o)	28.59	m ²
Cross-sectional area of shell (m ²)	3.64	m ²
Shell outside diameter (D _(s,o))	0.35	m
Overall heat transfer coefficient (U _o)	16	W/m ² K
Tube-side heat transfer coefficient (h _t)	1986.23	W/m ² K
Shell-side heat transfer coefficient (h _s)	16.21	W/m ² K
Tube side pressure drop (ΔP _t)	5.34	kPa
Shell side pressure drop (ΔP _s)	0.22	kPa

Thereafter, physics of the problem was defined by specifying the fluid properties, including density, viscosity, and specific heat capacity. Applying boundary conditions like inlet/outlet conditions, wall temperature and heat transfer coefficient for the fluids. The solver settings were then configured by choosing a suitable solver, k-epsilon, being a widely used model for simulating turbulent flows in heat exchangers was utilized with energy equation in the study to compute fluid flow behaviour and model heat transfer phenomena. Turbulence model was selected and numerical schemes chosen for discretization and solution of the governing equations. The simulation was then run using a computational fluid dynamic (CFD) solver to iterate and converge to a solution. The CFD solver used

numerical methods to solve the governing equations such as the Navier-Stokes using steady-state energy equations to obtain results. Finally, the results of the simulation were analysed and visualized using post-processing tools.

3.2 Temperature distribution

The tools used allowed viewing of temperature distribution, fluid flow patterns and other results in a graphical format making it easier to understand the heat exchanger's behaviour as shown in Figures 8 to 13. By analysing these results, the areas of the heat exchanger's design that need to be improved and optimized for better performance could be identified.

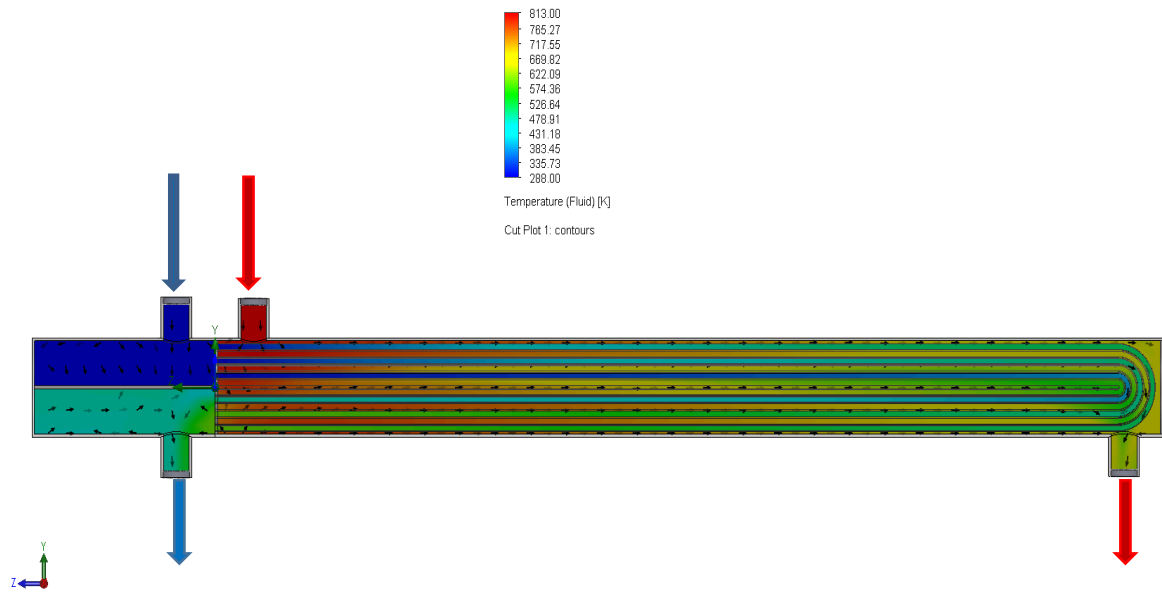


Figure 8: Temperature Cut Plot

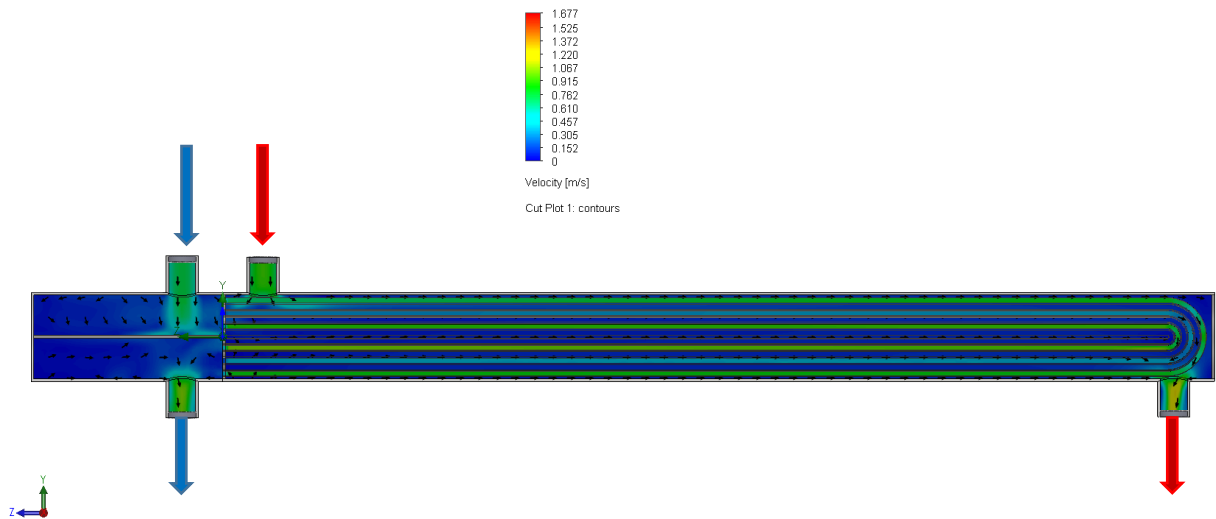


Figure 9: Velocity Cut Plot

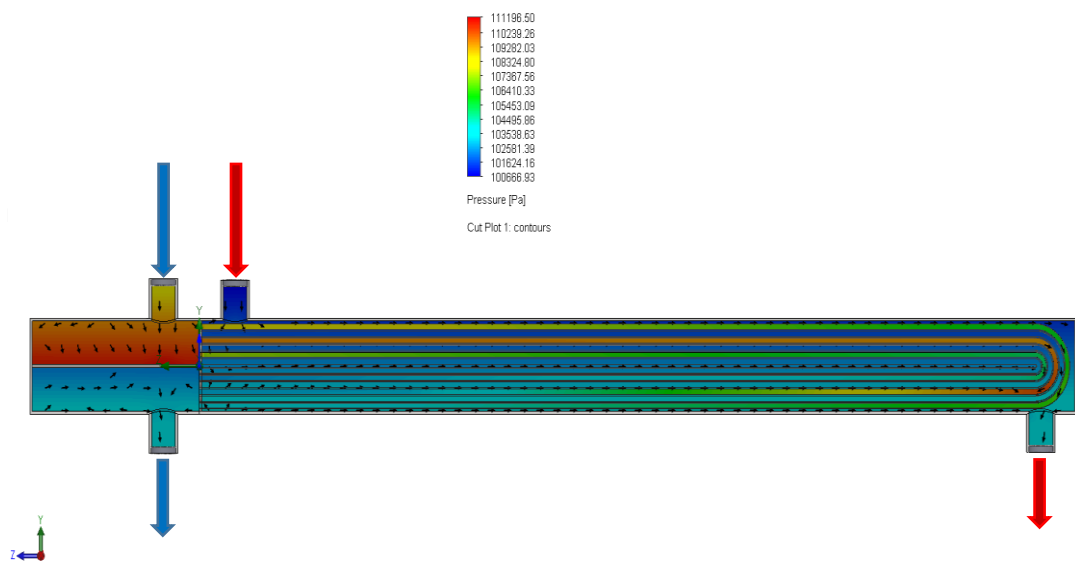


Figure 10: Pressure Cut Plot

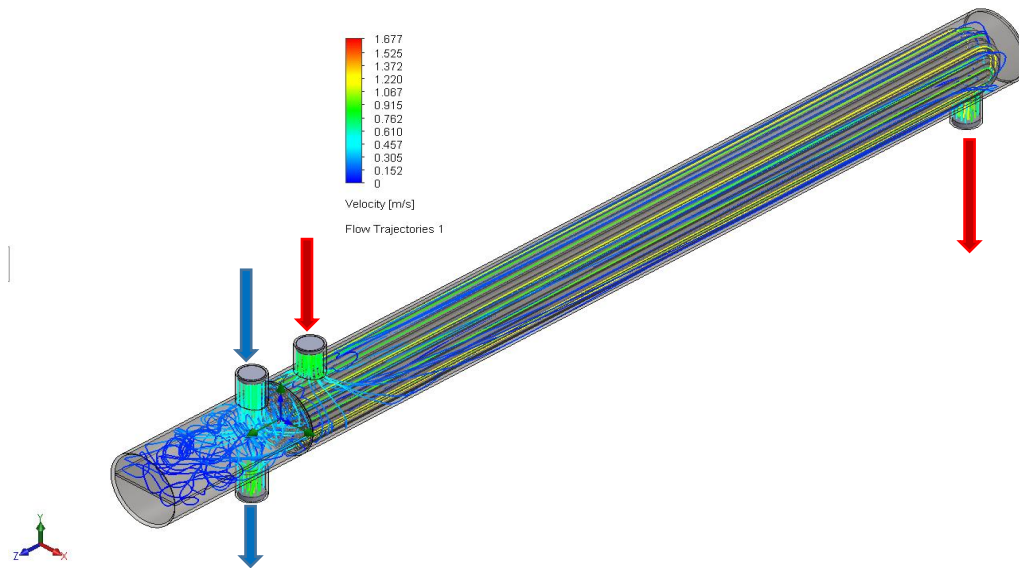


Figure 11: Velocity Flow Trajectory

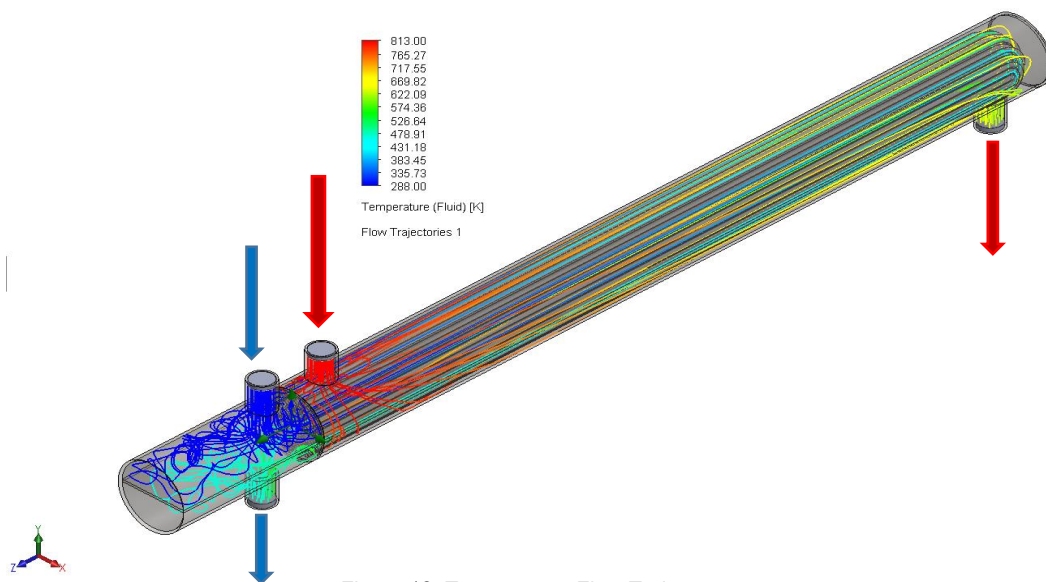


Figure 12: Temperature Flow Trajectory

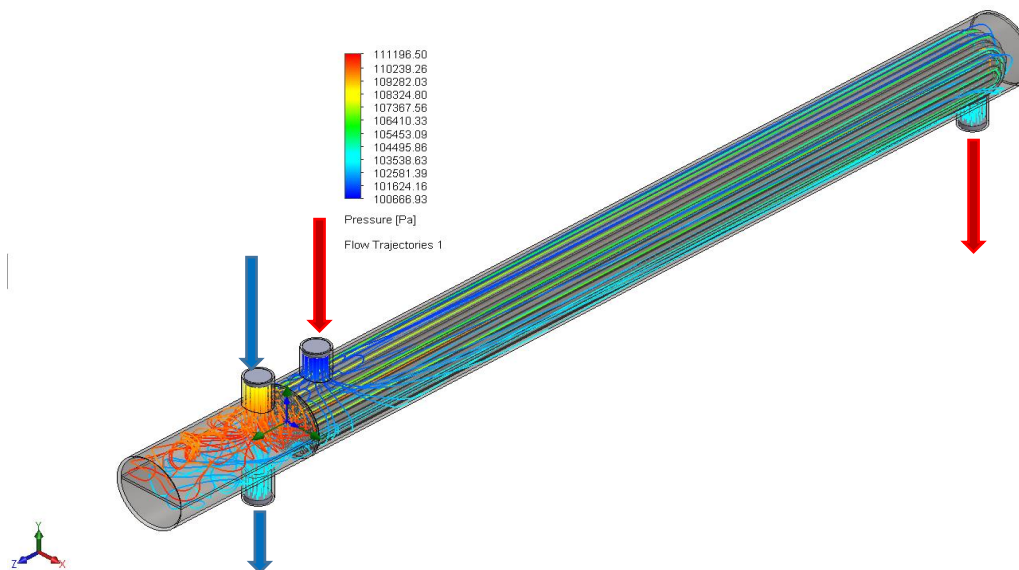


Figure 13: Pressure Flow Trajectory

Table 3: Validation of Simulation Results

Parameter	Simulation	Literature value	Percentage error (%)
Temperature gradient (cold fluid) (K)	288-383.45	290-390 (Khan et al. 2020)	0.69-1.68 %
Temperature gradient (hot fluid) (K)	765.27-813	770-820 (Khan et al. 2020)	0.61-0.85 %
Velocity range (m/s)	0-1.372	0-1.4 (Wang et al. 2019)	2 %
Higher velocity at inlet/outlet	Yes	Yes (Wang et al. 2019)	
Decreased velocity within tube/shell	Yes	Yes (Wang et al. 2019)	
Velocity flow trajectory	Zigzag/Counter-flow	Zigzag/Counter-flow (Shah, R. K and Sekulic, 2018)	
Pressure range at cold fluid inlet (Pa)	100666.93-102581.39	100000-105000 (Saunders, E. A., and Franz, 1999)	0.56-0.67 %
Fluid flow pattern on shell side	Zigzag/Counter-flow	Zigzag/Counter-flow (Gao, B., and Chen, 2018)	
Fluid flow pattern on tube side	Laminar/Turbulent	Laminar/Turbulent (Gao, B., and Chen, 2018)	
Pressure drop on shell-side (Pa)	1000-2000	1200-2500 (Kakac, S., Liu, H., and Pramuanjaroenkij, 2018)	16.67-20 %
Pressure drop on tube-side (Pa)	500-1000	600-1200 (Shah, R. K and Sekulic, 2018)	16.67-20 %

The simulation results in Table 3 show good agreement with literature values, with a temperature gradient for the cold fluid ranging from 288 K to 383.45 K, which is within 0.69-1.68% of the literature value of 290-390 K. The temperature gradient for the hot fluid ranges from 765.27 K to 813 K, with a percentage error of 0.61-0.85% compared to the literature value of 770-820 K. The velocity range is 0-1.372 m/s, which is close to the literature value of 0-1.4 m/s with a percentage error of 2%. The simulation also accurately captures the velocity flow trajectory, with zigzag and counter-flow patterns observed on both the shell and tube sides, consistent with literature findings. The pressure range at the cold fluid inlet is 100666.93-102581.39 Pa, with a percentage error of 0.56-0.67% compared to the literature value of 100000-105000 Pa. Additionally, the simulation results show higher velocity at the inlet/outlet, decreased velocity within the tube/shell, and laminar/turbulent flow patterns on the tube side, all consistent with literature findings. However, the pressure drop on both the shell-side and tube-side shows a higher percentage error of 16.67-20% compared to literature values. Overall, the simulation results demonstrate good agreement with literature values, validating the accuracy of the simulation model.

4. Conclusion

In conclusion, this study demonstrates the effectiveness of using computational fluid dynamics (CFD) simulations to analyze the thermal and fluid dynamics performance of shell and tube heat exchangers. The simulation results show good agreement with literature values, validating the accuracy of the model. Given the similarity between the current work and literature results, it is recommended that future studies focus on refining the model to reduce percentage errors, particularly for pressure drop predictions. Additionally, experimental validation of the simulation results and investigation of different design parameters would further enhance the understanding and optimization of heat exchanger performance. By implementing these recommendations, the design and performance of shell and tube heat exchangers can be further optimized, leading to improved efficiency and effectiveness in various industrial applications.

Acknowledgment

The authors thank the staff and management of Geregu II Gas Turbine Power Plant for providing data for the study.

References

- Shah, R. K and Sekulic, D. P. (2018). Heat Exchangers: Selection, Rating and Thermal Design,. CRC Press, Boca Raton, Florida
- Kehlhofer, R., Hannemann, F., Rukes, B., and Stirnimann, W. (2009) Combined-Cycle Gas and Steam Turbine Power Plants. PennWell Corporation Page 1-100
- Kakac, S., and Liu, H. (2002). Heat Exchangers: Selection, Rating and Thermal Design. CRC Press, Boca Raton, Florida
- Abd, A. A., Kareem, M. Q. and Naji, S. Z., (2018). Performance Analysis of Shell and Tube Heat Exchanger: Parametric Study. *Case Studies in Thermal Engineering*, 12, 563–568.
- Ambekar, A. S., Sivakumar, R., Anantharaman, N. and Vivekenandan, M., (2016). CFD Simulation Study of Small and Tube Heat Exchangers with Different Baffle Segment Configurations. *Applied Thermal Engineering*, 108, 999–1007. <https://doi.org/10.1016/j.applthermaleng.2016.08.013>
- Khan, (2020). Heat Transfer Analysis of a Pipe-in-Pipe Heat Exchanger,. *International Journal of Heat and Mass Transfer*, 151, 119–131.
- Wang, (2019). *Heat Transfer Rates and Velocity Distributions in Various Engineering Applications*, volume 861, page 119-137
- Ali, A., Qasim, M. and Zaki, S., (2018). Performance analysis of shell and tube heat exchanger: Parametric study. *Case Studies in Thermal Engineering*, 12(June), 563–568. <https://doi.org/10.1016/j.csite.2018.07.009>
- Ali, A., and Zaki, S., (2017). Analysis study of shell and tube heat exchanger for clough company with reselect different parameters to improve the design. *Case Studies in Thermal Engineering*, 10(March), 455–467. <https://doi.org/10.1016/j.csite.2017.10.002>
- Ammar, A. A., Mohammed, Q. K. and Samah, Z. N., (2013). 3D Numerical Simulation on the Shell Side Heat Transfer and Pressure Drop Performances of Twisted Oval Tube Heat Exchanger. *International*

- Journal of Heat and Mass Transfer*, 65, 244–253. <https://doi.org/https://doi.org/10.1016/j.ijheatmasstransfer.2013.06.011>
- ASME (2019). *ASME Boiler and Pressure Vessel Code*, Section VIII, Division 1, (Appendix A), ASME, New York, United States of America.
- Baina, F., Malmquist, A., Alejo, L. and Fransso n, T. H., (2018). *Analysis of a high-temperature heat exchanger for an externally-fired micro gas turbine*, *Journal of Applied Thermal Engineering*, Vol 7, Pg 410-420
- Chen, H. M., Hung, Y. H. and Wang, C. C., (2020). *Optimization of a Compact Heat Exchanger for Waste Heat Recovery from a Small Gas Turbine*. 142(10), 101802. <https://doi.org/10.1115/1.4047374>
- Surya, V., Adi, K., and Laxmidewi, R., (2022). Shell and tube heat exchanger flexible design strategy for process operability. *Case Studies in Thermal Engineering*, 37(June), 102163. <https://doi.org/10.1016/j.csite.2022.102163>
- Chukwudi, B. C. and Ogunedo, M. B., (2018). *Design and Construction of a Shell and Tube Heat Exchanger*. *Elixir International Journal*, 118, 50687–50691.
- Correa, D. J. and Marchetti, J. L., (2007). Dynamic Simulation of Shell-and-Tube Heat Exchangers. *Heat Transfer Engineering*, 8(1), 50–59. <https://doi.org/https://doi.org/10.1080/01457638708962787>
- Darbandi, M., Abdollahpour, M. and Hasanpour-Matkola, M., (2021). A Semi-Full-Scale CFD Approach for Shell-and-Tube Heat Exchangers (STHEs). *Chemical Engineering Science*, 245, 116836. <https://doi.org/https://doi.org/10.1016/j.ces.2021.116836>
- Rao, S., Khan, A., Modak, S. and Bani Venkatesh, S. P., (2020). Analysis of Shell and Tube Heat Exchanger for Waste Heat Recovery by using CFD, *Journal of Emerging Technologies and Innovative Research (JETIR)*, 7(4), 605-612. www.Jetir.Org,
- Franti, D. and Martin, J., (2017). *Shell and tube heat exchanger – the heat transfer area design process*, 67(2), 13–24. <https://doi.org/10.1515/scjme-2017-0014>
- Gao, B., and Chen, Q., (2018). Numerical Investigation of Flow and Heat Transfer in a Shell-and-Tube Heat Exchanger, *Interantional Journal of Heat and Mass Transfer*, 121, 588–599.
- Hadidi, A., Hadidi, M., and Nazari, A., (2013). *A new design approach for shell-and-tube heat exchangers using imperialist competitive algorithm (ICA) from economic point of view*. 67, 66–74.
- Isiaka, I., Bonet, M. U., Mohammed, S. U. and Muhammad, M. M., (2025). Parametric analysis of shell and tube heat exchanger for a simple cycle gas turbine power plant. *Savannah Journal of Science and Engineering Technology*, 03(02), 95–103.
- Zhang, J., He, Y. and Tao, W., (2009). 3D Numerical Simulation on Shell-and-Tube Heat Exchangers with Middle-Overlapped Helical Baffles and Continous Baffles-Part I: Numerical Mode and Results of Heat, *International Journal of Heat and Mass Transfer*, 52(23–24), 5371–5380. <https://doi.org/https://doi.org/10.1016/j.ijheatmasstransfer.2009.07.006>
- Kakac, S., Liu, H. and Pramuanjaroenkij, A., (2018). *Heat Exchangers: Selection, Rating and Thermal Design*, CRC Press, Boca Raton, Florida
- Kara, Y. A. (2004). A computer program for designing of shell-and-tube heat exchangers, *Journal of Applied Thermal Engineering*, 24, 1797–1805. <https://doi.org/10.1016/j.applthermaleng.2003.12.014>
- Kern, D. Q., (1950). *Process Heat Transfer*. McGraw-Hill, New York
- Lovaglio, N., Pina, A., Ferrão, P., Fournier, J., Lacarrière, B. and Corre, O. Le. (2018). Heat exchanger design and optimization by using genetic algorithm for externally fired micro-turbine, *Energy Procedia*, 148, 720–727. <https://doi.org/10.1016/j.egypro.2018.08.162>
- MechStudies. (2025). *What are Heat Exchangers? Definition, Parts, Types, How Does It Works, Equation*. Retrieved from <https://www.mechstudies.com/heat-exchangers-definition-parts-types/>. access on 13/02/2025.
- Mishra, M. and Das, P. K., (2017). Effect of Mass flow Rate on the Performance of a Shell and Tube Heat Exchange. *Journal of Thermal Science and Engineering Applications*, 9(2), 021012.
- Mulyana, T., Suhaimi, F. N., Hanafi, D., Nor, M. and Than, M., (2011). *ARX Model of Four Types Heat Exchanger Identification*. 1–5.
- Natesan, K. and Karinka, S., (2023). A comprehensive review of heat transfer enhancement of heat exchanger , heat pipe and electronic components using graphene. *Case Studies in Thermal Engineering*, 45, 102874. <https://doi.org/10.1016/j.csite.2023.102874>
- Ozden, E. and Tari, I., (2010). Shell Side CFD Analysis of a Small Shell-and-Tube Heat Exchanger. *Energy Conversion and Management*, 51(5), 1004–1014. <https://doi.org/https://doi.org/10.1016/j.enconman.2009.12.003>
- Incropera, H. J. P., (2010). *Heat Transfer*, Wiley, Hoboken, New Jersey, p. 365.
- Pamuk, M. T., (2020). *CFD Simulation of a Shell and Tube Heat Exchanger*. 1–8. <https://doi.org/10.11159/iccpe20.105>
- Perumal, S., Sundaresan, D., Sivanraju, R., Tesfie, N., Ramalingam, K. and Thanikodi, S., (2022). *Heat transfer analysis in counter flow shell and tube heat exchanger*. 26(2), 843–848.
- Pontevedra, Y., (2018). Study of Shell and Tube Heat Exchanger with the Effect of Types of Study of Shell and Tube Heat Exchanger with the Effect of Types of Baffles of Zanin Afonso Costing models for capacity optimization in Industry 4 . 0: Trade-off between used capacity and pressure drop. *Procedia Manufacturing*, 20, 195–200. <https://doi.org/10.1016/j.promfg.2018.02.028>
- Prithviraj, M. and Andrews, M. J., (1998). Three Dimensional Numerical Simulation of Shell-And-Tube Heat Exchangers. Part I: Foudation and Fluid

- Mechanics. *International Journal of Computation and Methodology*, 33(2), 139-155
<https://doi.org/https://doi.org/10.1080/10407789808913968>
- Rao, S. S., Das, P. K. and Mitra, S. K., (2020). Design and Optimization of a Heat Exchanger for Waste Heat Recovery from a Gas Turbine, *Applied Thermal Engineering*, 173, 115243.
<https://doi.org/10.1016/j.applthermaleng.2020.115243>
- Saunders, E. A. and Franz, B., (1999). Shell-and-Tube Heat Exchangers, *Heat Exchanger Design Handbook*, Begell House., 3.1-3.24.
- Sekulic, S. and Shah, R. K. (2003). *Shell-and-Tube Heat Exchangers*. *Journal of Heat Transfer Engineering*, 24(4) 123-134
- Silaipillayarputhur, K., and Khurshid, H. (2019). *The design of shell and tube heat exchangerS: A review*, 9(1), 87–102.
- TEMA (2019). *Standard of the Tubular Exchange Manufacturers Association10th Edition, TEMA*, 17.
- Thet, S., Than, M., Lin, K. A. and Mon, M. S., (2008). *Heat Exchanger Design*. World Academy of Science, Engineering and Technology, Vol 47, page 604–611
- Wang, Q., Chen, G. and Xu, X., (2018). Numerical Investigation on the Effects of Shell Diameter on Heat Transfer Performance in Shell-and-Tube Heat Exchangers, *Applied Thermal Engineering*, 131, 531–539.
- Wang, S., Wen, J. and Li, Y., (2009). An experimental investigation of heat transfer enhancement for a shell-and-tube heat exchanger, *Applied Thermal Engineering*. 29, 2433–2438.
<https://doi.org/10.1016/j.applthermaleng.2008.12.008>
- You, A., Be, M. and In, I., (2023). *Performance of A High-Temperature Particle-Based Shell-Shell-and-Tube Solid Particle Thermal Energy Storage System, Thermal Energy Storage System. Sustainability*, 17, 7244