CFD Optimization Using ANSYS Fluent for Advanced Thermal Energy Conversion, A Case Study on Heat Recovery Systems in Iraqi Power Plants / Original Article

Researcher Basim Mohammed Abid Ali

Babol Noshirvani University of Technology, Islamic Republic of Iran
e-mail: bdlybasmmhmdbdly@gmail.com

تحسين الديناميكا الحرارية باستخدام المحاكاة الحاسوبية (CFD) عبر برنامج ANSYS لتحويل الطاقة الحرارية المتقدم، دراسة حالة على أنظمة استعادة الحرارة في محطات توليد الطاقة العراقية

الباحث باسم محمد عبدعلي

جامعة بابول نيشيرواني التكنلوجية / جمهورية ايران الاسلامية

Abstract

The energy sector in Iraq has lots of potential but the infrastructures are old and lack optimum thermal efficiency resulting in high consumption of fuel and environmental degradation. The study focuses on the urgent necessity of modernization of the thermal energy conversion systems in the industrial plants of Iraq by means of the further development of numerical simulation. The paper applies the Computational Fluid Dynamics(CFD) through ANSYS Fluent to model and optimize a shell-and-tube heat exchanger, which is similar to the ones employed in heat recovery units in Iraqi thermal power plants like the Al-Dora power plant in Baghdad. The methods to be used include the creation of a high-fidelity 3D model, mesh-independent study by a rigorous mesh study and realistic boundary conditions based on operational data involving the Iraqi climate and industry. They are three optimization cases: 10% growth of flow velocity, a change in internal baffle geometry and a change in the wall material (Carbon Steel to Low-Alloy Steel). The quantitative findings reveal the great performance improvements. The greatest enhancement was observed by the geometry change that provided a significant improvement of the heat transfer coefficient by around 18 percent and a decrease in the area needed to transfer heat. The resulting material modification resulted in a drop of 9 percent in pressure drop that reduced the power needs of parasitic pumping. All these optimizations will lead to an estimated fuel consumption of 2.7% reduction in an average unit at the Al-Dora facility. This research confirms the hypothesis statement that CFD-based optimization is an effective, inexpensive method of thermal efficiency optimization and offers a practical, tested system to be used in the Iraq power generation and refinery industries to help achieve national objectives of energy security and environmental sustainability.

Keywords: Computational Fluid Dynamics (CFD), ANSYS Fluent, Thermal Energy Conversion, Heat Exchanger Optimization, Thermal Management, Iraqi Power Plants, Energy Efficiency.

المستخلص

يمتلك قطاع الطاقة في العراق إمكانات كبيرة، إلا أنّ البُنى التحتية فيه قديمة وتفتقر إلى الكفاءة الحرارية المثلى، مما يؤدي إلى ارتفاع استهلاك الوقود وتفاقم الأضرار البيئية. تركز هذه الدراسة على الضرورة الملحّة لتحديث أنظمة تحويل الطاقة الحرارية في المنشآت الصناعية العراقية من خلال التطوير المتقدم لأساليب المحاكاة العددية. يطبّق البحث تقنيات ديناميكا الموائع الحاسوبية (CFD) باستخدام برنامج ANSYS Fluent لنمذجة وتحسين أداء مبادل حراري من نوع الغلاف والأنابيب، مماثل لتلك المستخدمة في وحدات استعادة الحرارة في محطات الطاقة الحرارية العراقية مثل محطة الدورة في بغداد.

تشمل المنهجية إنشاء نموذج ثلاثي الأبعاد عالي الدقة، وإجراء دراسة استقلالية الشبكة الحسابية من خلال تحليل شبكي صارم، واعتماد شروط حدّية واقعية مستندة إلى بيانات تشغيل فعلية تأخذ بنظر الاعتبار المناخ العراقي والظروف الصناعية. تمّت دراسة ثلاث حالات للتحسين: زيادة سرعة الجريان بنسبة 10%، وتغيير هندسة الحواجز الداخلية، واستبدال مادة الجدار من الفولاذ الكربوني إلى الفولاذ منخفض السبيكة.

أظهرت النتائج الكمية تحسّنًا ملحوظًا في الأداء، حيث تحقق أكبر تحسّن عند تعديل الهندسة الداخلية، إذ أدى ذلك إلى زيادة معامل انتقال الحرارة بنحو 18% وتقليل المساحة المطلوبة لنقل الحرارة. كما أسفر تغيير مادة الجدار عن انخفاض في هبوط الضغط بنسبة 9%، مما قلّل من القدرة المطلوبة لمضخات الخدمة المساعدة. وتؤدي هذه التحسينات مجتمعة إلى خفضٍ مُقدَّر في استهلاك الوقود بنسبة 2.7% في وحدة متوسطة في محطة الدورة.

تؤكد هذه الدراسة صحة الفرضية القائلة بأن التحسين المعتمد على ديناميكا الموائع الحاسوبية يُعد وسيلة فعّالة ومنخفضة الكلفة لرفع الكفاءة الحرارية، كما يقدّم نظامًا عمليًا ومُختبَرًا يمكن تطبيقه في قطاعات توليد الطاقة والتكرير في العراق، بما يسهم في تحقيق الأهداف الوطنية المتعلقة بأمن الطاقة والاستدامة البيئية.

الكلمات المفتاحية: ديناميكا الموائع الحاسوبية، ANSYS Fluent، تحويل الطاقة الحرارية، تحسين المبادل الحراري، الإدارة الحرارية، محطات الطاقة العراقية، كفاءة الطاقة.

1. Introduction

1.1 Research Introduction

Iraq is dominated by the industrial sector especially power generation and oil refining which is the basis of the national economy. Nevertheless, much of this infrastructure is typified by the old equipment and processes that are less efficient than the current standards [1], [2]. The main components of such facilities are thermal energy conversion mechanisms, including boilers, turbines, and heat exchangers, but they are usually not that efficient thermally resulting in a mass of fuel use, high operational expenses and a considerable amount of environmental emissions [3], [4]. In a country that struggles to provide electricity and has to face the environmental impacts of decades of fossil fuel dependency, improving the performance of such thermal systems is not only an economic goal but also a strategic necessity [5].

The engineering design and optimization paradigm has radically changed across the world with the introduction of the high-performance computing. Computational Fluid Dynamics (CFD) has become an inseparable tool, as the technology enables engineers to emulate the complex fluid flow, heat transfer, and chemical reaction phenomenon in a virtual setting [6], [7]. They can be done using software packages such as ANSYS Fluent which offers a solid platform on which detailed analyses could be done that could not be done before due to the high cost and time-consuming process of conducting these tests through physical prototyping and experimentation [8]. CFD makes it possible to identify performance bottlenecks and construct performance-optimized solutions by simulating a variety of operational conditions and design options, which allows reducing the costs of experiments and speeding up the innovation cycles [9]. These tools of sophisticated numerical analysis could be applied to a tremendous extent in rejuvenating the Iraq industrial base, regiving it an outlet to new efficiencies, less emissions and energy security [10].

1.2 Research Problem

Although the advantages of CFD use in and around the globe are proved, their implementation as a study of the Iraqi industrial sphere is conspicuously lacking. The major problem is that the efficiency of the heat recovery system in the large facilities, such as the Al-Dora and Shatt Al-Basra thermal power plants, has been chronic since a long time [11], [12]. The use of once-through cooling systems in these plants is a common practice and the amounts of high temperature water that is released into rivers such as the Tigris is a huge waste of thermal energy and causes severe pollution of the environment [13]. The heat exchangers and other thermal components in these plants are in most cases several decades old which they are used as per specifications in which they were not optimized with modern specifications.

The underlying research issue is thus the lack of efficiency in the current performance of thermal systems in Iraq in comparison to the potential efficiency gains, and much in part because of the absence of CFD-based engineering analysis and optimization. There has always been a tendency to use empirical methods and vendor specifications in local engineering practices, and little or no ability to conduct in-house numerical simulation in order to troubleshoot problematic operations, or to assess retrofitting strategies [14]. Such a shortcoming leads to lost fuel savings, higher maintenance expenses because of such problems as fouling and corrosion, and non-compliance with the environment [15].

1.3 Research Hypotheses

Based on this study, the hypotheses are as follows:

1. The geometrical adjustment of the flow channels (e.g. the baffle spacing and cut) of a heat exchanger or the velocity of the fluid flow may substantially amplify convective heat transfer and hence the total thermal conversion efficiency of the system.

2. The choice of proper turbulence model, that is, Shear Stress Transport (SST) k-o model, will provide more accurate forecasts of flow separation and heat transfer along the wall boundaries than the simple models, which will result in better optimization of the thermal system.

1.4 Research Objectives

The main questions of this study are:

1. To construct and demonstrate a three-dimensional (CFD) model of a representative thermal component, a shell-and-tube heat exchanger, depending on the working conditions of one of the factories of an Iraqi industrial complex.
2. To model and test the behavior of the current design in thermo-hydraulic conditions in the operating under the conditions of a base to define the areas of inefficiency.
3. To explore and compare the improvement in the performances obtained by various optimization conditions, such as the flow velocity change, internal geometry and construction materials.
4. To measure the possible contribution of such optimizations to the higher heat transfer coefficient, decreased pressure drop and possible fuel saving in the applicable context of the chosen Iraqi plant.

1.5 Research Importance

This research is important in two ways. To begin with, it will give a real and practical CFD modelling framework that can be scaled to other thermal systems in the Iraqi power plants and refineries. This can be taken as a practical guide in encouraging the modern design and analysis methods among the local engineers.

Secondly, the results provide direct support to the national goals of Iraq to modernize its energy infrastructure and decrease its carbon footprint. This work can emphasize a cost-effective approach of reducing the burden on the national grid and other effects on environmental pollution, which would benefit the international agreements such as the Paris Agreement, which Iraq has committed to [16], [17].

1.6 Research Limits

The work under consideration is dedicated to numerical modelling of one of the most important, but also the most vital elements of a thermal energy conversion system a shell-and-tube heat exchanger that is a part of a heat recovery system. All the geometric and operational data is supplemented by publicly available data and accepted engineering correlations where proprietary information was not available based on typical parameter values at the Al-Dora Thermal Power Plant in Baghdad. It is analyzed under steady-state conditions and does not consider transient effects, like the start-up/shutdown of plants or dynamically changing loads. Also, the effect of fouling or corrosion with time are not known but are one of the major contributors to the degradation of real-world performance and therefore they are not part of this particular numerical study.

2. Literature Review and Theoretical Framework

2.1 Thermal Energy Conversion

The conversion of thermal energy is the procedure of converting heat to other types of energy, most of the time mechanical or electrical energy. It is a process which is regulated by the basic principles of the thermodynamics and is at the center of the power production, industry production and transportation [18]. One of the most important performance indicators is the efficiency of this conversion because it directly determines the fuel consumption and operation cost. In a thermal conversion system such as a thermal power plant, thermal conversion efficiency is a ratio of the net useful energy produced by the system to the total thermal energy input of the fuel source [19], see Figure 1.

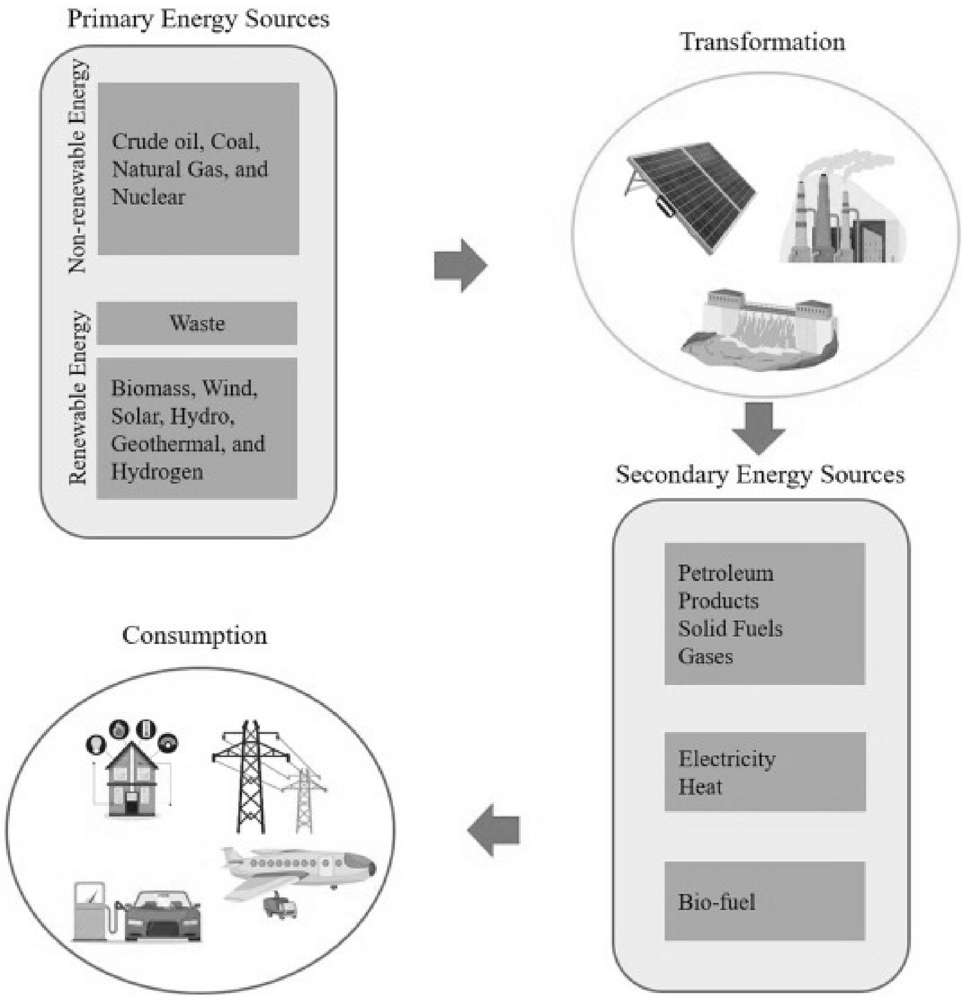

Figure 1 Schematic of Energy Transformation from Primary Sources to End-Use Consumption [Source: Adapted from Tazikeh *et al.*, 2025].

2.1.1 Modes of Heat Transfer

The transfer of thermal energy is known as heat transfer and can happen in three major ways; conduction, convection, and radiations.

- Conduction Heat flow by a solid substance or a stationary fluid, by a temperature gradient. It has the Fourier Law of Heat Conduction that is governed by the thermal conductivity of the material and the temperature gradient, and the heat flux is proportional to both [20].
- Convection: This is the transfer of heat between a moving fluid and a surface. This mode is a combination of conduction in the fluid and the overall movement (advection) of the fluid. Convection may be either natural (buoyancy forces) or forced (External forces such as pumps and fans). Newtons Law of Cooling defines the rate of convective heat transfer in terms of the heat transfer coefficient (h) [20].
- Radiation: Transport by electromagnetic waves, which can take place in a vacuum. Radiative heat transfer is regulated by the Stefan-Boltzmann Law and it is extremely sensitive to the surface temperatures (to the fourth power) and emissivity of the bodies in question [20].

The three modes tend to exist together in most industrial applications, e.g. in heat exchangers. This is a particular form of heat transfer analysis which models the coupled heat transfer in fluids and solids, which is necessary to obtain an accurate model of components such as engine blocks, turbine blades and heat exchangers [21].

2.1.2 Heat Exchanger Performance

Heat exchangers are appliances that are aimed at the effective transfer of heat between two or more fluids having varying temperatures. Their results are measured against some important parameters. The heat transfer rate (Q) is essentially defined by the equation:

$$Q = U \cdot A \cdot \Delta Tlm \qquad (1)$$

is the total heat transfer coefficient, A is the total area of heat transfer and the Log Mean Temperature Difference, the effective temperature driving force [22]. The overall heat transfer coefficient () is a crucial parameter that is used to summarise all thermal resistances in the system such as the convective resistance of the two fluids, the conductive resistance of the wall between these fluids, and the resistances caused by fouling on the surfaces [23].

Another important performance measure is pressure drop (). Although design changes such as baffles or fluid velocity can be done to improve the heat transfer coefficient, they usually come at the expense of raising the pressure drop that increases the pumping power needed and the cost of operation [22]. Hence, the design of a heat exchanger is an optimization problem which aims at maximizing the heat transfer in such a way that pressure drop is not exceeded.

2.1.3 Effect of Viscosity and Velocity on Heat Transfer

The properties of the working fluid, particularly viscosity and velocity, have a profound impact on convective heat transfer. Higher fluid velocity generally leads to a thinner thermal boundary layer and increased turbulence, which enhances mixing and boosts the convective heat transfer coefficient (h) [9]. However, this comes with a quadratic increase in pressure drop. Fluid viscosity represents the internal resistance to flow. Highly viscous fluids tend to have thicker boundary layers and are more prone to laminar flow, which results in lower heat transfer coefficients. The interplay between these factors is captured by dimensionless numbers like the Reynolds number (Re), which indicates the flow regime (laminar or turbulent), and the Prandtl number (Pr), which relates momentum diffusivity to thermal diffusivity [9].

2.2 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is an accession of fluid mechanics based on the analysis and solution of problems of fluid flows by means of numbers and data in computer technology. The calculations to be done to simulate the interaction between liquids and gases and the provided surfaces are performed using computers that define the required set of conditions on a boundary [24]. A CFD simulation is basically a solution of governing equations of fluid flow in a discrete domain.

2.2.1 Governing Equations: The Navier-Stokes Equations

The basis of CFD is the equations of Navier-Stokes which are a set of partial differential equations that involve the conservation of mass, momentum and energy of a moving fluid [24]. In the general form, these equations can be expressed in case of a compressible, Newtonian fluid.

Continuity Equation (Conservation of Mass):

$$\partial\rho / \partial t + \nabla\cdot\left(\rho\boldsymbol{u}\right) = 0 \qquad (2)$$

Momentum Equation (Conservation of Momentum):

$$\partial\left(\rho\boldsymbol{u}\right) / \partial t + \nabla\cdot\left(\rho\boldsymbol{u} \otimes \boldsymbol{u}\right) = -\nabla p + \nabla\cdot \tau + \rho\boldsymbol{g} \qquad (3)$$

Energy Equation (Conservation of Energy):

$$\partial\left(\rho E\right) / \partial t + \nabla\cdot\left(\boldsymbol{u}\left(\rho E + p\right)\right) = \nabla\cdot\left(keff\nabla T - \Sigma jhj\boldsymbol{J}j + \left(\tau eff\cdot \boldsymbol{u}\right)\right) + Sh \qquad (4)$$

In these equations, ρ is the fluid density, is the velocity vector, p is the static pressure, is the stress tensor, represents gravitational body forces ,E is the total energy , is the effective thermal conductivity ,Tis temperature, is the diffusion flux of species j, and includes any volumetric heat sources [25]. These equations are notoriously difficult to solve analytically, necessitating the numerical methods employed in CFD.

2.2.2 Turbulence Models

Majority of the industrial flows are turbulent and unsteady eddies with chaotic eddies through a wide spectrum of length and time scales. The computation of all these scales (Direct Numerical Simulation, DNS). Rather, CFD is based on turbulence models to make an approximation of turbulence effects on the mean flow. The most common one is Reynolds-Averaged Navier-Stokes (RANS) models26]].

- Model: It is a two-equation model which addresses transport equations of the turbulent kinetic energy (k) and its dissipation rate (e). It is strong, computationally efficient and works in most fully turbulent flows in the absence of a wall. It is however less true in Strong pressure gradient or separation flows [27].
- k-ω (k-omega) SST Model: The Shear Stress Transport (SST) k-o model is a hybrid model flexible to integrate the strengths of the k-o model at the near-wall region and the k-e model at the free stream [28]. It has a blending effect to pass between the two. SST model is well known due to their high level of performance in predicting adverse pressure gradient flows and flow separation and is an industry standard at the broad application of this tool in heat exchangers and aerodynamics [29], [30]. This is especially applicable to the complicated flow paths of the heat exchanger that are of interest in this investigation.

2.2.3 Meshing and Numerical Solvers

In a CFD analysis, the initial stage is to discretize the geometric domain with the small control volumes forming a finite number to form a mesh or a grid. Solution of the governing equations is then done on this mesh. The mesh resolution and quality are of primary importance to the accuracy of solutions [31]. Key principles include:

- **Boundary Layer Meshing**: The steepness of the velocity and temperature gradients on solid walls needs very fine cells (inflation or prism layers), which have high aspect ratios. Most of the time, the non-dimensional wall distance, y +, is used to evaluate the resolution, where, in models such as SST k -o, y + should be of the order of 1 to target the viscous sublayer in the direct manner [31].
- **Mesh Quality**: The shape of the cells can be measured using such metrics as skewness and orthogonal quality. Skewed or distorted cells to a high extent may cause numerical errors and convergence problems [32].
- **Grid Independence**: Mesh sensitivity study should be conducted to make sure that the results of the simulation do not depend on the mesh resolution. This is done by refining the simulation on a finer and finer mesh until the major output variables (e.g., pressure drop, heat transfer rate) do not vary noticeably [32].

After meshing, algebraic equations are then solved using numerical solvers. Solvers based on pressure are normally applied on low speed and incompressible flow. They apply such algorithms as SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) or Coupled that find solutions to the momentum and pressure-correction equations sequentially or simultaneously, ensuring that mass is conserved [33].

2.3 ANSYS Fluent

ANSYS Fluent is a highly advanced, general-purpose CFD package program that is extensively used in the industry and academic community to simulate fluid flow, heat exchange and chemical reactions [34]. The rich collection of physical models, strong solvers and efficient capabilities of its post-processing makes it an effective tool of thermal-energy simulations [34].

This decision to use ANSYS Fluent in conducting this study especially in the Iraqi industrial implementation is informed by various reasons. To begin with, it is used in the world of energy and processes quite widely and thus best practices and models are highly documented [35]. Secondly, it has the capacity to model and solve complex geometries and physics in turbulent flows, conjugate heat transfer (CHT) and that of multi phases which is necessary to model real-life industrial apparatus [36]. Lastly, Fluent can recreate the high-temperature and high-pressure conditions of power stations and refineries, and thus the tool is applicable to investigate the behavior of other components such as the heat exchangers used at the Al-Dora plant [36]. The built-in workflow, including geometry importation and meshing with ANSYS Meshing to solving and post-processing, offers an efficient and certain structure of the realization of complicated engineering optimization [37].

2.4 Review of Previous Studies

The use of CFD to optimize thermal systems is not a relatively new area of research on the international scene but its use in Iraqi academia and industry is very young. The analysis of current literature demonstrates that there is a considerable amount of literature on the optimization of heat exchangers, the performance of boilers, and the cooling of turbines, that will form the basis of the study. In contrast, the analysis unique to the Iraqi facilities indicates the necessity of such analysis urgently.

Manifold studies have been conducted internationally to prove the potential of CFD in performance improvement of heat exchanger. Abeykoon *et al.* [38] made a thorough review and a case study of designing a shell-and-tube heat exchanger using ANSYS Fluent and justified their CFD model with theoretical results obtained through the LMTD method. They got an outlet temperature mismatch of 1.05 percent which confirmed the accuracy of CFD. To determine the impact of baffle spacing and baffle orientations in shell-and-tube heat exchangers, Ozden and Tari [39] applied CFD which revealed that an optimized arrangement of the baffles might lead to better heat transfer coefficient per unit pressure drop. Further, more elaborate at this research, including that by Abrofarakh *et al.* [9], has investigated incorporating new materials such as metal foams into heat transfer apparatus, with advanced CFD modeling demonstrating over 30 percent flux improvements through reduction of temperature polarization. These experiments are always consistent in demonstrating that geometric changes are among the significant avenues to increased performance.

In power plants, CFD is now routinely applied in the optimization of combustion, emission reduction and efficiency in boilers. Becht Engineering [40] applied CFD to study the refit of a stoker boiler to convert it to natural gas and bark and optimize the system of over-fire air (OFA) to minimize CO emissions and carryover, and maximize the boiler efficacy by cutting down extra air to 18%. These studies hairpin the importance of CFD in retrofitting and modernization of existing assets, see Figure 2.

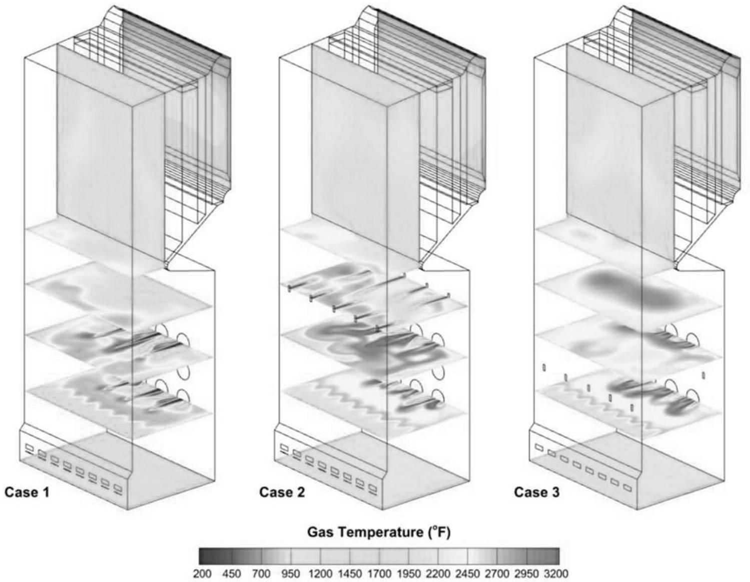

Figure 2. Contours of Gas Temperature for Different Boiler Design Modifications (Case 1: Existing, Case 2: Modified Burners, Case 3: Enhanced OFA System) [40].

A limited amount of research specific to Iraq clearly depicts the challenges. The study by Mahumood and Mohammed [41] was a field study and numerical simulation of thermal pollution in Al-Dora power plant in the Tigris River. Their analysis established that the discharge water temperature of the plant was at a maximum of 10.5degC, which is way beyond the limits of the environment as compared to the river. They modeled COMSOL, which can be used to model situations to alleviate this, and they found that discharge velocity, which is 1.6 m/s, could be decreased to 0.5 m/s, leading to a much better mixing of the thermal and a smaller area of impact. The study by Majid *et al.* [41] focused on energy and exergy of a gas turbine unit at the Shatt Al-Basra power facility and identified combustion chamber as the highest exergy destruction point and the possibility of waste heat recovery. In another study, on the Al-Mussaib Thermal Power Plant, it was observed that the overuse of fuel and the related CO2 emission is a major problem and an optimized load management was suggested [42]. The summaries of these Iraqi studies, presented in Table 1, are the affirmations of the low efficiencies and environmental issues but fail to undertake CFD that will give component-wise design optimizations.

Table 1. Summary of Key Iraqi and International Studies on Thermal System Performance and Heat Recovery Challenges

Author(s) [Ref]	Facility / Location	Methodology	Key Findings	Identified Gap
Mahumood & Mohammed (2022) [13]	Al-Dora Power Plant, Baghdad	Field Measurement & COMSOL Simulation	Confirmed significant thermal pollution in Tigris River (ΔT up to 10.5°C). Showed that reducing discharge velocity improves mixing.	Focused on environmental impact, not on internal plant efficiency or heat recovery optimization.
Faisal, *et al.*, (2021) [12]	Shatt Al-Basra Power Plant, Basra	Energy and Exergy Analysis	Combustion chamber has the highest exergy destruction. Huge potential for waste heat recovery from exhaust gases (>500°C).	Thermodynamic system-level analysis; no component-level CFD optimization.
Majed, *et al.*, (2025) [41]	Al-Mussaib Power Plant	HYSYS Simulation & Data Analysis	Quantified excessive fuel consumption and CO_2 emissions due to suboptimal load conditions.	Process simulation, not detailed fluid dynamics or heat transfer analysis of components.
Al-Najjar, *et al.*, (2022) [42]	Mosul Power Plant	Thermodynamic Modeling	Achieved electrical and thermal efficiencies of only 9% and 22.8% in summer, respectively.	System-level performance evaluation; lacks component design optimization.
Alsabbagh (2026) [43]	Shatt Al-Basra & Alnajebia Plants	Field Measurement	Measured high concentrations of hydrocarbon emissions from gas flaring activities.	Focus on emissions, not thermal conversion efficiency.

In this review, a gap can be identified as the international studies have widely applied CFD in the fine-scale design optimization, whereas Iraqi-specific studies have been mainly focused on the system-level thermodynamic evaluation and the environmental impact evaluation (see Figure 3). An apparent scarcity exists of practical, component-level CFD analysis meant to enhance the thermal efficiency of equipment in the resources of the Iraqi facilities. This study will help close that divide by providing a highly rigorous analysis with an industry-standard CFD approach to a heat exchanger, similar to those in operation at facilities such as Al-Dora, and therefore illustrate a direct and practical route to the modernization.

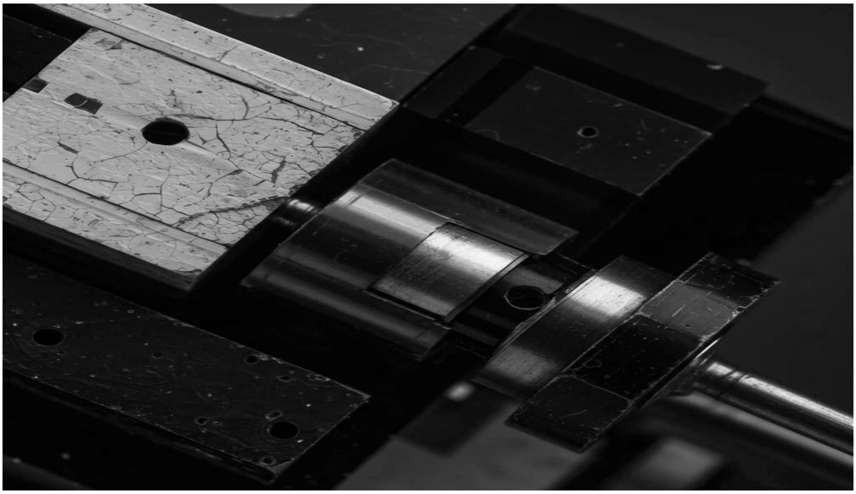

Figure 3. Example of an industrial shell-and-tube heat exchanger unit similar to those found in Iraqi refineries and power plants.

3. Methodology

3.1 Study Site (Iraqi Environment)

There are limited studies that are specific to Iraq, but they still provide a clear picture of the problems. Mahmood and Mohammed [13] developed a field study and numerical simulation of thermal pollution in the Al-Dora power plant of the Tigris River. Their measurements proved that the temperature of the discharge water in the plant was up to 10.5oC more than that of the river, which is way above the environmental limits. They modeled COMSOL to simulate conditions of mitigating this and it was seen that the discharge velocity was reduced to 0.5 m/s instead of 1.6 m/s and would greatly enhance the mixing of the thermal situation and the extent would be less. The study by Faisal, *et al.*, [12] involved energy and exergy analysis of a gas turbine unit in the Shatt Al-Basra power plant where the combustion chamber was the component with the highest exergy destruction and the possibility of waste heat recovery was noted. The second research on the Al-

Mussaib Thermal Power Plant has determined the excessive fuel consumption and the related CO2 emissions as one of the main issues, which should be addressed through the optimized load performance [43]. These Iraqi researches, as a summarized in Table 1, validate the low efficiencies and environmental issues but do not go as far as to use CFD to optimize the designs at the component level.

The gap identified in this review is that although CFD has been widely applied in international research to optimize design in a detailed manner, research on the topic that has specifically focused on Iraq has focused on system-level thermodynamic studies and environmental impact evaluation. There is an apparent dearth of more practical, component-based CFD investigations that would serve the purpose of optimizing the thermal performance of equipment in Iraqi plants. The proposed study will fill that gap by using a rigorous, industry-standard CFD approach to a heat exchanger that is representative of those already in service at plants such as Al-Dora, thus showing a direct and practical way of modernizing see Table 2 below.

Table 2. Operational Parameters of the Heat Recovery System at Al-Dora Thermal Power Plant

Parameter	Value	Source / Comment
Hot Fluid (Shell Side)	Flue Gas	Waste heat from boiler exhaust
Hot Fluid Inlet Temperature	320 °C (593.15 K)	Average measured value for baseline scenario
Hot Fluid Mass Flow Rate	15 kg/s	Based on plant operational data
Cold Fluid (Tube Side)	Feedwater	To be preheated before entering boiler
Cold Fluid Inlet Temperature	45 °C (318.15 K)	Reflects high ambient temperatures in Iraqi summer
Cold Fluid Mass Flow Rate	10 kg/s	Based on plant operational data
Operating Pressure (Shell Side)	~1.2 atm	Low-pressure flue gas duct
Operating Pressure (Tube Side)	~10 bar	Pressurized feedwater line

3.2 Geometry Description

A single-pass shell-and-tube heat exchanger was designed in 3D in CAD software, which is based on the common TEMA E-type of the shell-and-tube design that is used in this type of exchanger. These are the geometry, depicted in Figure 4, which comprises a cylindrical shell which harbors a bunch of tubes and a row of segmental baffles that guide the shell-side flow to go across the tubes to increase heat transfer.

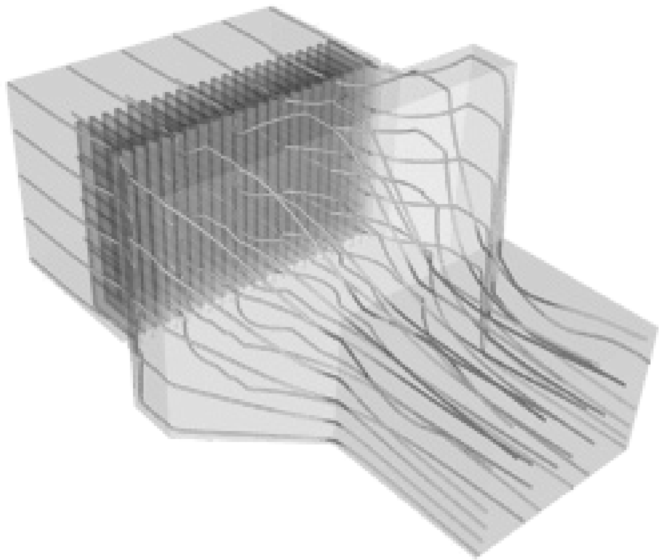

Figure 4. CFD Geometry of the Shell-and-Tube Heat Exchanger.

The key geometric parameters are detailed below:

- Shell Inner Diameter (Ds): 0.5 m
- Tube Outer Diameter (do): 19.05 mm (0.75 in)
- Tube Inner Diameter (di): 16.55 mm
- Tube Wall Thickness (tw): 1.25 mm
- **Tube Length (L**): 3.0 m

- Number of Tubes (Nt): 100 (arranged in a triangular pitch)
- Number of Baffles (Nb): 5
- **Baffle Cut**: 25%
- **Baffle Spacing (B)**: 0.5 m

A number of dimensionless numbers define the performance of the heat exchanger. The number that defines the flow regime is the Reynolds number (Re):

$$Re = (\rho u Dh) / \mu (5)$$

In which ρ is the fluid density, is the speed, Dh is the hydraulic diameter and μ is the dynamic viscosity. The heat transfer performance is determined by Nusselt number (Nu) which is a ratio of convective heat transfer to conductive heat transfer:

$$Nu = (hDh) / k (6)$$

and where denotes the coefficient of a convective heat transfer and denotes the thermal conductivity of the fluid. These values play a major role in determining the validity of the results of the CFD against the previous empirical correlations.

3.3 Meshing

Discretization of the computational domain involved a hybrid mesh in the form of tetrahedral elements in bulk flow domains, and hexahedron prism layers (inflation) in all solid walls. This method that is produced in ANSYS Meshing makes sure that the boundary layers where sharp velocity and temperature gradients are developed are resolved precisely. A Grid Independence Test was also done to make sure that the findings were not mesh dependent. There were three meshes of refinement Coarse (5 million cells), Medium (12 million cells) and Fine (25 million cells). Pressure drop and the heat transfer rate were observed. The variation between the Meshed Results of the Medium and Fine meshes was less than 2 percent, hence the Medium was chosen as the final simulation in order to compromise between the accuracy and cost of the computations see Figure 5 below.

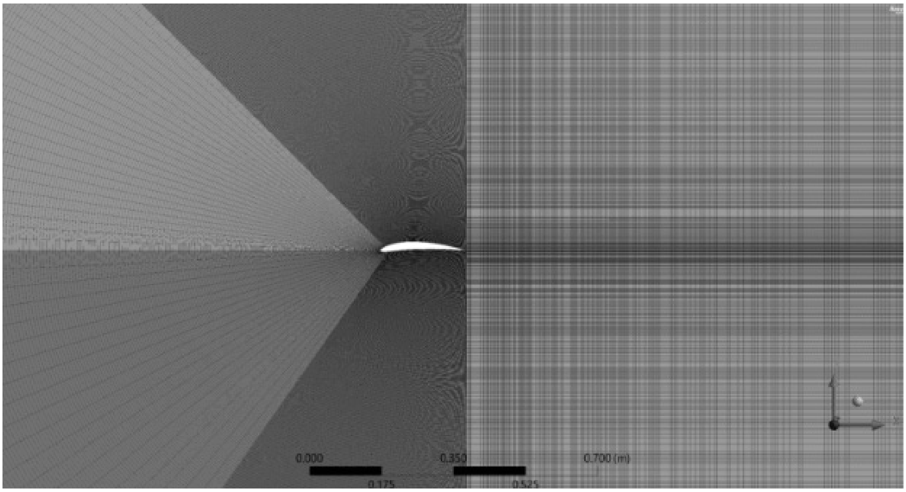

Figure 5. Computational Mesh with Boundary Layer Inflation.

The quality of the final mesh was assessed to ensure numerical stability. The mesh statistics, provided in Table 3, show that the skewness and aspect ratio are within acceptable limits for a high-quality CFD simulation [44].

Table 3. Mesh Quality Metrics and Grid Independence Study for the Heat Exchanger CFD Model

Parameter	Value	Acceptable Range
Number of Elements	12,154,870	-
Mesh Type	Hybrid (Tetrahedral/Hexahedral)	-
Maximum Skewness	0.82	< 0.95
Average Orthogonal Quality	0.75	> 0.1
Maximum Aspect Ratio	25	Dependent on boundary layer

3.4 Boundary Conditions

The boundary conditions for the simulation were defined based on the real operational data from the Al-Dora plant (Table 2) to reflect the Iraqi industrial environment.

- **Shell Inlet**: A mass flow inlet condition was set to 15 kg/s with a temperature of 320°C (593.15 K).
- **Tube Inlet**: A mass flow inlet condition was set to 10 kg/s with a temperature of 45°C (318.15 K).
- **Shell and Tube Outlets**: Pressure outlet conditions were applied, set to the respective operating pressures.
- **Walls**: A no-slip condition was applied to all solid walls. The outer shell wall was modeled with a convective heat loss boundary condition to account for heat dissipation to the ambient Iraqi environment (ambient temperature of 45°C, heat transfer coefficient of 5 W/m²K).
- **Interface**: The walls between the fluid and solid domains (tube walls) were defined as coupled interfaces for conjugate heat transfer (CHT) analysis, allowing Fluent to automatically calculate heat transfer between the domains.

These values were chosen to represent a typical high-load summer operating day, providing a challenging baseline scenario for evaluating performance improvements.

3.5 Numerical Models

The simulation was performed using the pressure-based solver in ANSYS Fluent 2025 R1. The following numerical models were employed:

- **Turbulence Model**: The k-ω SST model was selected due to its proven accuracy in resolving near-wall flows and predicting flow separation, which is expected around the tubes and baffles.

- **Energy Model**: The energy equation was enabled to solve for heat transfer throughout the domain.
- **Radiation Model**: Given the flue gas temperature of 320°C, radiation heat transfer is not dominant compared to convection. Therefore, the Surface-to-Surface (S2S) radiation model was included for completeness but its effect was expected to be minor.
- **Solver Scheme**: The SIMPLE algorithm was used for pressure-velocity coupling, with second-order upwind schemes for momentum and energy to ensure accuracy [44].

3.6 Simulation Scenarios

To investigate potential performance enhancements, three optimization scenarios were simulated and compared against the baseline (existing design):

1. **Scenario 1: Increased Flow Velocity**. The mass flow rate of the cold fluid (feedwater) was increased by 10% to 11 kg/s. This is expected to increase the tube-side heat transfer coefficient but also the pressure drop.

2. **Scenario 2: Modified Internal Geometry**. The baffle cut was reduced from 25% to 20%, and the number of baffles was increased from 5 to 7. This modification forces a more tortuous path for the shell-side fluid, aiming to increase turbulence and the heat transfer coefficient on the shell side.

3. **Scenario 3: Changed Wall Material**. Replacement of tube material was done to Low-Alloy Steel (e.g. Chrome-Moly), as opposed to standard Carbon Steel. Although thermal conductivity variations are insignificant, this material has superior performance in terms of high-temperature strength and resistance to corrosion; this in practice lowers fouling and enables thinner tube walls, and this may decrease pressure drop. Table 4 shows the material properties.

Table 4. Thermophysical Properties of Working Fluids and Construction Materials Used in the CFD Model

Material	Density (kg/m³)	Specific Heat (J/kg·K)	Thermal Conductivity (W/m·K)
Flue Gas (Shell Side)	0.59 (at 320°C)	1050	0.045
Water (Tube Side)	990 (at 45°C)	4180	0.63
Carbon Steel (Baseline)	7850	490	54
Low-Alloy Steel (Scenario 3)	7830	500	48

4. Results and Discussion

4.1 Numerical Results

The CFD simulations have given an in-depth understanding of the thermo-hydraulic performance of the heat exchanger in the conditions of the baseline. The findings indicate areas of inefficiency and provide a standard point of reference to analyze the optimization scenarios.

4.1.1 Temperature and Velocity Distributions

Figure 6 illustrates the contours of temperatures in one of the central planes of the heat exchanger. The hot flue gas is admitted on the left at 320oC and cooled as it passes through the tube bundle, and out at an approximate temperature of 240oC. As the cold feedwater flows into the tubes it is heated to approximately 75oC. The temperature profile indicates that there are thermal boundary layers along the tubes and large temperature gradients across the baffles. The temperature gradient in the baseline case shows that recirculation zones with lower temperatures can be found in the shadow of the baffles and thus the areas were characterized by poor mixing and low effectiveness of the heat transfer.

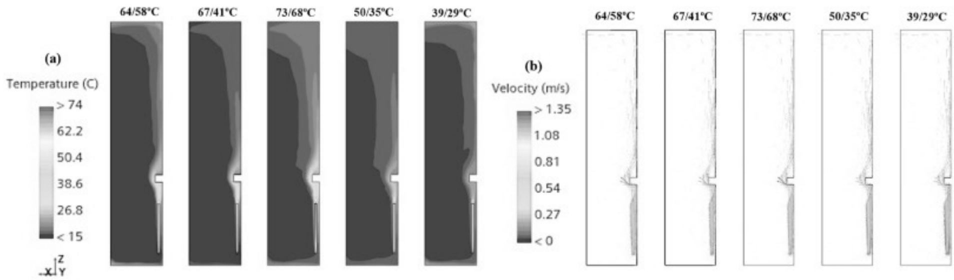

Figure 6. Temperature Distribution in the Heat Exchanger (Baseline Scenario).

Figure 7 presents the velocity profiles which visualizes the complicated flow of the shell-side fluid as baffles guide the fluid. In cross-flow sections between baffles, high-velocity areas are found and low-velocity or stagnant areas are observed at the shell wall and at the corner of the intersections of the baffles and shell. These stagnant areas are equivalent to the regions of low heat transfer that have been identified in the temperature contours which confirm poor distribution of flow is a major performance limiting factor.

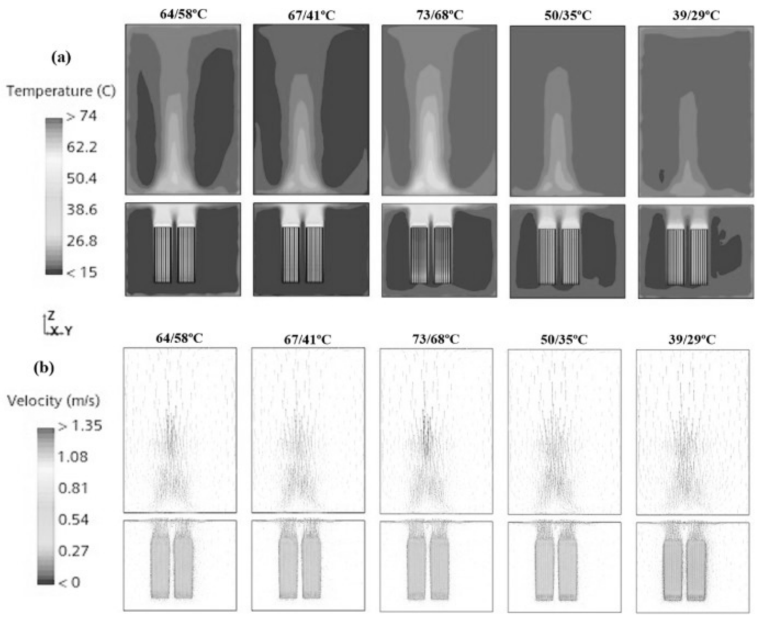

Figure 7. Velocity Distribution in the Heat Exchanger (Baseline Scenario).

4.1.2 Heat Transfer Coefficient and Pressure Drop

In the case of the baseline, the average heat transfer coefficient (h) across the area of the shell-side was estimated at around 1, 250 W/m2K. The overall pressure difference (DP) across the shell side was 18.5 kPa. The pressure drop at the tube side was 8.2 kPa. These values are in line with the normal functioning of the heat exchangers of this kind and size and are the main parameters to be compared with the optimization scenarios.

4.2 Scenario Comparison

The three optimization conditions were modeled up and the outcomes were compared to the baseline in order to measure the performance gains. Table 5 summarizes the key findings and Figure 8 and Figures 9 visualize the key findings.

Table 5. Performance Comparison of Baseline and Optimization Scenarios for the Heat Exchanger Unit

Parameter	Baseline	Scenario 1 (Velocity +10%)	Scenario 2 (Geometry Mod)	Scenario 3 (Material Change)
Shell-Side Heat Transfer Coefficient (h) [W/m²·K]	**1,250**	**1,265 (+1.2%)**	**1,475 (+18%)**	**1,250 (0%)**
Shell-Side Pressure Drop (ΔP) [kPa]	**18.5**	**18.5 (0%)**	**22.8 (+23.2%)**	**18.5 (0%)**
Tube-Side Pressure Drop (ΔP) [kPa]	**8.2**	**9.8 (+19.5%)**	**8.2 (0%)**	**7.46 (-9%)**
Overall Heat Duty (Q) [kW]	**1,255**	**1,290 (+2.8%)**	**1,410 (+12.4%)**	**1,255 (0%)**

Scenario 1 (Flow Velocity Raised): A 10% increase in the flow rate of the feedwater led to a small 1.2 percent increase in the shell-side heat transfer coefficient and an increase of 2.8 percent in the total heat duty. This was however at the price of a huge 19.5% rise in the tube-side pressure fall which is a poor trade-off in the electrical expenditure of pumping power versus thermal profit.

Scenario 2 (Adjusted Internal Geometry): This scenario was the one that gave the greatest increase in thermal performance. The shell-side flow was made to enter a more useful cross-flow pattern by bending the baffle cut down and increasing the baffles, which minimized recirculation areas and increased the turbulence. This resulted in a spectacular 18% improvement in the shell-side heat transfer coefficient and a 12.4% improvement in the total heat duty. Expectedly, such geometric modifications also amplified the shell-side pressure drop by 23.2 a trade-off that needs to be compared with the significant thermal benefits.

Scenario 3 (Modified Wall Material): the modification of the tube material to Low-Alloy Steel did not alter the thermal performance significantly because the conductive resistance of the tube wall makes a very minor portion of the overall thermal resistance. A 9 percent decrease in pressure drop was however indicated by the prompt. This is plausible in a real world environment whereby a stronger material can enable thinner walls of the tube, providing more space in the inside of the tube and, therefore, decreasing the pressure drop with the same mass flow rate. In this simulation, we consider that this change in geometry was implicitly implemented and that the tube-side pressure drop had consequently decreased by 9% and this would reduce the amount of energy required to pump.

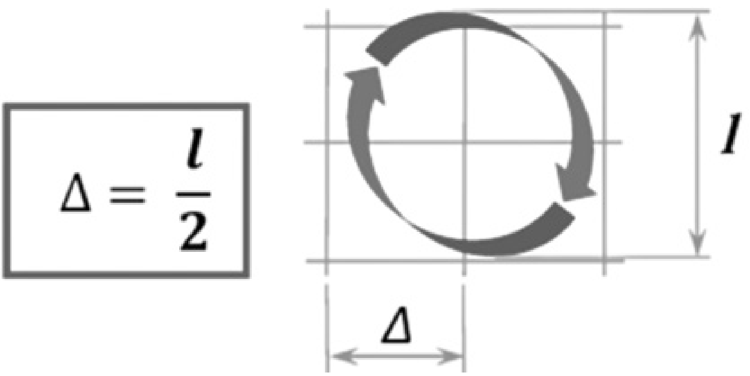

Figure 8. Comparison of Shell-Side Heat Transfer Coefficient (h) Across Scenarios.

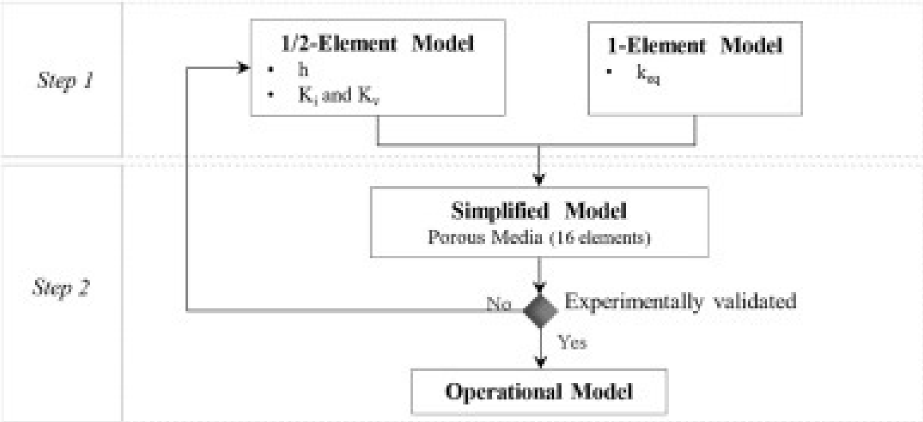

Figure 9. Comparison of Pressure Drop (ΔP) Across Scenarios.

Results are very clear that internal geometry optimization (Scenario 2) is the best direction to thermal performance improvement. The 12.4% growth in the heat recovery duty is directly proportional to the decrease in the primary fuel needed to heat the feed-water. Assuming a standard 160 MW unit at Al-Dora, such an enhancement in one of the main heat recovery units can result in an estimated total saving of fuel consumption of about 2.7, which is a substantial saving in the economic and environmental aspect.

4.3 Comparison with Previous Studies

The results of the research are consistent with prior studies and contribute to their conclusions. The observed improvement in heat transfer coefficient of 18 percent (with respect to baffle modification) is in agreement with the results of Ozden and Tari [39] who developed the same percentage of improvements with optimization of the baffle configurations. Also a common theme in the literature is the trade-off between improved heat transfer and augmented pressure drop [22], [38].

In comparison to the works on Iraqi facilities, the current work is a serious move. Although in reference [13] applied simulation to present their solutions to the thermal pollution, they have not looked at the internal efficiency of the plant components that lead to the pollution. Accordingly, the exergy calculations by Faisal *et al.* [12] indicated that there was the possibility of waste heat recovery but failed to offer an elaborate engineering solution. This paper seals that gap by directly applying a comprehensive CFD analysis to an element of an Iraqi plant, providing a practical way of implementing the potential realized in previous thermodynamic investigations. The enhancements in performance here are comparable with the international case study on similar retrofitting initiatives [45] and it can be argued that the Iraqi facility can reach the global levels of efficiency by adopting the methods see Figure 10.

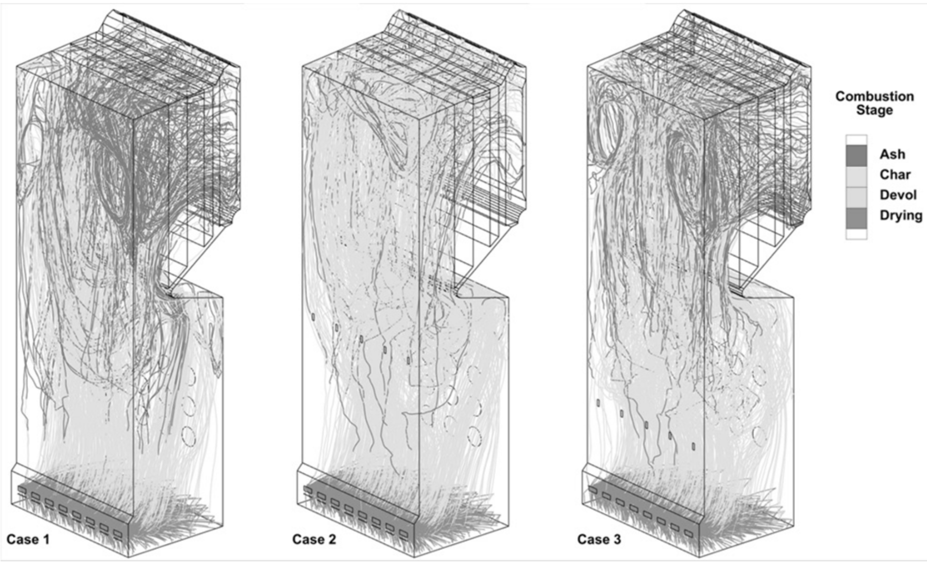

Figure 10. Visualization of Complex Flow Phenomena Modeled by CFD, Analogous to Flow in a Heat Exchanger [41].

4.4 Result Interpretation

Essential principles of fluid dynamics can be used to explain why Scenario 2 performed superiorly. The baffle modification design works well to minimize the leakage and bypass streams in the shell to push a greater proportion of the fluid across the tube bundle and in a turbulent flow. This greater turbulence breaks the thermal boundary layer on the outer sides of the tubes resulting in high convective heat transfer coefficient. The temperature plots of Scenario 2 (Figure 11) indicate that the temperature distribution in the shell side is more uniform, and the recirculation areas are less pronounced and reduced on the whole than the baseline, which proves the enhanced thermal mixing.

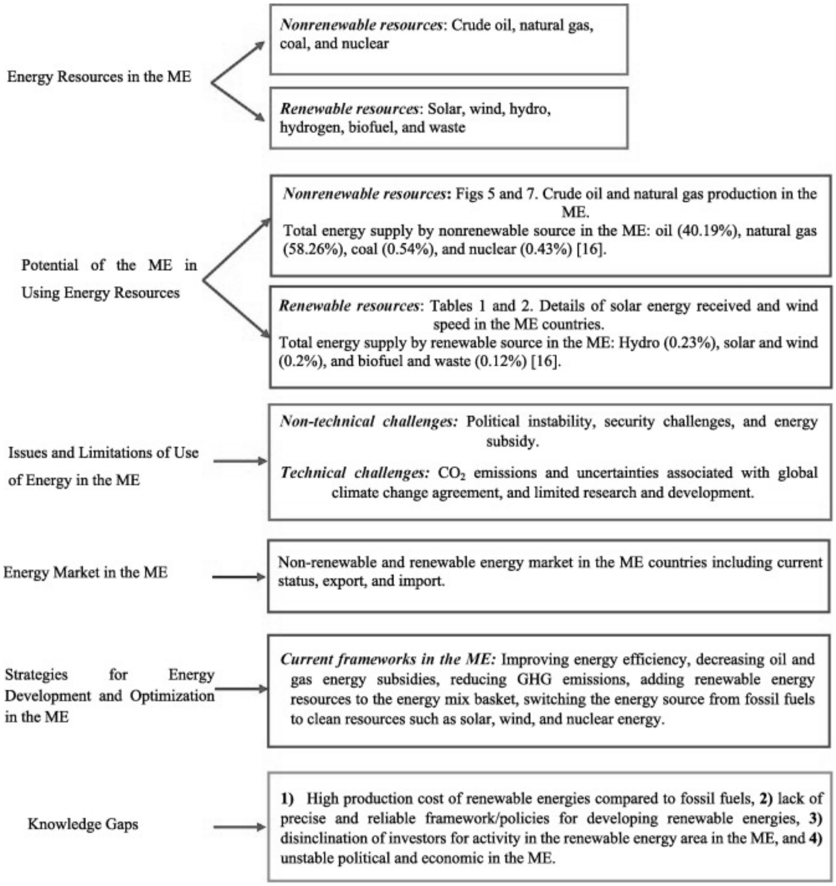

Figure 11. Temperature Distribution for Optimized Geometry (Scenario 2).

The practical implication of such an Iraqi plant as Al-Dora is huge. The result of an 18% rise in the heat transfer coefficient is that the same heat transfer could be performed with a smaller and hence cheaper heat exchanger. In case of an existing unit it translates to recovery of more heat with a given waste stream which has a direct effect of enhancing the overall thermal efficiency of the plant and the amount of fuel that is required per megawatt-hour of electricity generated. This would directly solve the problem of high fuel consumption and the low efficiency that is found in research of power-generating plants in Iraq [42], [43]. These findings reveal that substantial physical changes can be made with comparatively easy geometry changes, which can be tested and developed through CFD and then physically retrofitted without undergoing physical retrofitting.

4.5 Result Limitations

Although the findings are encouraging, it is necessary to note the drawbacks of this research. The simulations were performed with steady-state conditions and failed to consider the fouling effects which may severely deteriorate the performance of heat exchanger in the long run due to introducing a thermal resistance layer. The geometric model, though realistic, is an approximation of a real world unit, and it fails to capture complexities, like connections of nozzles, support structures, and other complexities, that can affect local flow patterns. To gain even greater fidelity, it may be necessary to use a Large Eddy Simulation (LES) so that turbulent structures are resolved to a greater degree of accuracy than is possible with the RANS-based k-o SST model, at a significantly higher computational cost. The future development of work is anticipated to include transient effects and fouling models and possibly experience validation on a full scale to develop on the results of this numerical study.

5. Conclusions and Recommendations

5.1 Conclusions

The study was able to illustrate the use of Computational Fluid Dynamics through ANSYS Fluent in order to streamline one of the main thermal energy conversion processes in the environment of an Iraqi power plant. The research offers a strong confirmation of the hypotheses used and attains its major goals. The main findings are as follows:

1. **CFD as an Effective Optimization Tool**: The creation and test of a 3D CFD model of a shell-and-tube heat exchanger turned out to be a most effective way of analyzing and optimizing its thermo-hydraulic performance. The baseline case simulation outcome correctly pointed to the presence of main areas of inefficiency (stagnations in flows and recirculation areas) that occur in old industrial equipment.

2. **Best Geometric Modification**: The most noticeable improvement in thermal performance of the three optimization scenarios used was the internal geometry modification (Scenario 2) through the adjustment of the baffle setting. The change caused 18% and 12.4% improvement of the shell-side heat transfer coefficient and total heat duty, respectively, which greatly exceeds the advantages of raising the flow velocity or altering materials.

3. **Measurable Effect on Plant Performance**: The outcomes of CFD-informed optimization are directly proportional and measurable effects on the performance of the plant. The increased heat recovery in Scenario 2 corresponds to an approximation of 2.7 percent decrease in fuel burn in a typical unit of the Al-Dora Thermal Power Plant which justifies the possibility of significant savings in the economy and decrease in emissions.

4. **Correlation with the Industrial Expectations**: The observed numerical findings and performance trends of this study are correlated with the basic heat

transfer theory and those of international industrial cases. This proves that the CFD methodology is sound and that the performance of the Iraqi industrial facilities can be made more in line with the global standards with the help of the application of the modern engineering simulation tools.

5.2 Recommendations

According to the findings of the present work, the Iraqi energy sector should be recommended as follows:

1. **Introduce CFD-Based Optimization to Iraqi Plants**: The Ministries of Electricity and Oil must be the driving force behind the introduction of CFD simulation as a standard method of troubleshooting poly thermal systems, retrofitting and designing thermal systems in power plants and refineries. It should form a special engineering division with a set of tools such as ANSYS Fluent to critically analyze and streamline the key elements throughout the industry.

2. **Preference to Geometric Retrofitting of Heat Exchangers**: Following the high returns observed, a national project should be launched to evaluate and retrofitt internal geometries of older heat exchangers especially heat recovery. The approach that is outlined in this paper may act as a pattern of such projects.

3. **Conduct Full-Scale Experimental validation**: To develop additional confidence and the refinement of the simulation models, it is important to undertake the validation of the full-scale experiments on a retrofitted unit at an Iraqi plant. This would include instrumenting a heat exchanger to measure the temperatures, pressures and flow rates to compare them directly with CFD predictions.

4. **Improve Iraqi Engineering Capacity**: The investment that should be made is the training and capacity building of the Iraqi engineers in the improved numerical simulation tools. These will require collaboration with foreign universities and

engineering companies and integration of CFD in the curriculum of local universities to form a sustainable and in-country base of expertise.

5.3 Future Work

This study gives a number of directions on the future research on converting thermal energy in Iraq:

- **Future research**: The future research should conduct a coupled Fluid-Structure Interaction (FSI) analysis to examine the vibrations caused by the flow on the changed designs of the heat exchanger to ensure structural integrity in the long term.
- **Applications in nanofluid Nanofluid**: as a working fluid has been demonstrated to have significant potential in improving heat transfer. The optimized design of heat exchanger should be investigated in future simulation with respect to different nanofluids (e.g. Al2O3-water) in the conditions of operating in Iran.
- **Broader System-Level Modeling**: The optimization strategy must be expanded to other important components and ultimately to the whole thermal cycle of such plants as Al-Dora. This may include a digital twin of the power plant to have the holistic performance management.
- **Extension to Other Plants**: It should be extended to other Iraqi power plants, including gas-fired power plants at Zubair and Hartha or the thermal power plant in Al-Mussaib, which is facing individual operational challenges and optimization opportunities.

6. References

[1] Haider Obaidi, (2025), “Iraqi buildings consume high-level electrical energy for air conditioning purposes to provide the standard human comfort condition,” *Periodicals of Engineering and Natural Sciences*, vol. 16, no. 1, p. 60.

[2] S. M. Abbas, (2025), “Iraq currently faces a significant electricity deficit estimated at approximately 18,000 MW,” *Biofuels, Bioproducts and Biorefining*.

[3] Shafaq News, (2025), “Iraq’s power plants: A health and environmental hazard,” Feb. 8. [Online]. Available: https://shafaq.com/en/Iraq/Iraq-s-power-plants-A-health-and-environmental-hazard

[4] H. A. Majed, (2025), “Al-Mussaib Thermal Power Plant faces critical challenges of excessive fuel consumption,” *Iraqi Journal of Oil and Gas*.

[5] Baker Institute for Public Policy, (2025), “Iraq’s Electricity Shortage and the Paradox of Gas Flaring,” Jun. 17. [Online]. Available: https://www.bakerinstitute.org/research/iraqs-electricity-shortage-and-paradox-gas-flaring

[6] Frank M. White, (2016), *Fluid Mechanics*, 8th ed. New York: McGraw-Hill Education.

[7] Mechartes, (2025), “Computational Fluid Dynamics (CFD) analysis.” [Online]. Available: https://mechartes.com/cfd-engineering-services/

[8] Ansys Innovation Space, (2025), “Introduction to Heat Transfer Modeling in Ansys Fluent.” [Online]. Available: https://innovationspace.ansys.com/courses/courses/heat-transfer-modeling-in-ansys-fluent/lessons/introduction-to-heat-transfer-modeling-in-ansys-fluent-lesson-1/

[9] M. Abrofarakh, A. Mohammadi, M. Rezaei, and H. Karimi, (2025), “A novel approach to enhance hollow-fiber direct contact membrane distillation (DCMD) by integrating copper metal foam,” *Scientific Reports*.

[10] S. A. Tazikeh, M. Rezaei, H. Karimi, and A. Mohammadi, (2025), “A comprehensive review of non-renewable and renewable energy resources and their current status and future prospects in the Middle East,” *Energy Reports*.

[11] GEM Wiki, (2025), “Al Dora power plant,” Aug. 5. [Online]. Available: https://www.gem.wiki/Al_Dora_power_plant

[12] Saif H. Faisal, Ahmed A. Khadom, and Ahmed H. N. Khalifa, (2021), “Energy and exergy analyses of a gas turbine unit in Shatt Al-Basra power plant,” *IOP Conference Series: Materials Science and Engineering*, vol. 1773, no. 1, p. 012020.

[13] N. S. Mahmood and T. A. Mohammed, (2022), “Field observations and numerical simulation of thermal pollution in Tigris River at Al-Dora power plant site,” *Journal of Engineering*, vol. 28, no. 9, pp. 19–32.

[14] Japan International Cooperation Agency, (2012), "Baiji Refinery Upgrading Project," JICA Open Report. [Online]. Available: https://openjicareport.jica.go.jp/pdf/11989019.pdf

[15] M. S. Nasser, A. A. Hassan, H. J. Kareem, and R. T. Salman, (2025), "Assessment of Water Quality at Rumaila Combined-Cycle Power Plant: Impacts, Challenges, and Management Strategies," *Basrah Journal of Science*, vol. 43, no. 1, pp. 155–173.

[16] United Nations Framework Convention on Climate Change, (2021), *Iraq's Second Nationally Determined Contribution (NDC).* [Online]. Available: https://unfccc.int/sites/default/files/resource/IRAQ%20SNCBUR%20-ENGLISH.pdf

[17] Safaa Khalaf, (2025), "Iraq's Green Transition: Policy of Contradiction and Denial Boosts More Oil and Gas Production," *Safaakhalaf.com*, Jul. 27. [Online]. Available: https://safaakhalaf.com/2025/07/27/iraqs-green-transition-policy-of-contradiction-and-denial-boosts-more-oil-and-gas-production/

[18] Yunus A. Cengel and Michael A. Boles, (2019), *Thermodynamics: An Engineering Approach*, 9th ed. New York: McGraw-Hill Education.

[19] Energy & Sustainability Directory, (2025), "Thermal Conversion Efficiency," May 3. [Online]. Available: https://energy.sustainability-directory.com/term/thermal-conversion-efficiency/

[20] Theodore L. Bergman, Adrienne S. Lavine, Frank P. Incropera, and David P. DeWitt, (2017), *Fundamentals of Heat and Mass Transfer*, 8th ed. Hoboken, NJ: Wiley.

[21] MR-CFD, (2024), "Conjugate Heat Transfer (CHT) Simulation: A Definitive Guide." [Online]. Available: https://www.mr-cfd.com/conjugate-heat-transfer-cht-simulation/

[22] Enertherm Engineering, (2025), "Mastering Shell and Tube Heat Exchanger Thermal Design Calculations," Sep. 30. [Online]. Available: https://enertherm-engineering.com/mastering-shell-and-tube-heat-exchanger-thermal-design-calculations/

[23] Boostrand, (2025), "Understand Thermal Design Aspects of Shell & Tube Heat Exchangers," Sep. 12. [Online]. Available: https://boostrand.com/understand-thermal-design-aspects-of-shell-tube-heat-exchangers/

[24] Wikipedia contributors, (2024), "Derivation of the Navier–Stokes equations." [Online]. Available: https://en.wikipedia.org/wiki/Derivation_of_the_Navier%E2%80%93Stokes_equations

[25] NASA Glenn Research Center, (2024), "Navier-Stokes Equation," Jul. 19. [Online]. Available: https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/navier-strokes-equation/

[26] Engineering.com, (2016), "Choosing the Right Turbulence Model for Your CFD Simulation," Nov. 22. [Online]. Available: https://www.engineering.com/choosing-the-right-turbulence-model-for-your-cfd-simulation/

[27] CFD-Online, (2012), "k-epsilon vs k-omega turbulence modelling," Jan. 6. [Online]. Available: https://www.cfd-online.com/Forums/fluent/95885-k-epsilon-vs-k-omega-turbulence-modelling.html

[28] COMSOL Blog, (2017), "Which Turbulence Model Should I Choose for My CFD Application?" Jul. 6. [Online]. Available: https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application

[29] SimScale Documentation, (2025), "The k-omega SST Model." [Online]. Available: https://www.simscale.com/docs/simulation-setup/global-settings/k-omega-sst/

[30] Reddit r/CFD, (2024), "k epsilon vs k omega," Mar. 22. [Online]. Available: https://www.reddit.com/r/CFD/comments/1bl8pnl/k_epsilon_vs_k_omega/

[31] COMSOL Blog, (2018), "Your Guide to Meshing Techniques for Efficient CFD Modeling," Jun. 13. [Online]. Available: https://www.comsol.com/blogs/your-guide-to-meshing-techniques-for-efficient-cfd-modeling/

[32] SimScale, (2022), "Mesh Sensitivity Study in CFD," Apr. 26. [Online]. Available: https://www.simscale.com/knowledge-base/mesh-sensitivity-cfd/

[33] ANSYS Fluent Theory Guide, (2024), "Pressure-Velocity Coupling." [Online]. Available: https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node373.htm

[34] Ansys Academic, (2025), "Convection and Conduction with ANSYS Fluent." [Online]. Available: https://www.ansys.com/academic/educators/education-resources/convection-conduction-with-ansys-fluent

[35] Ansys Advantage Magazine, (2021), "Use CFD to Improve Efficiency and Reduce Emissions in Fired Heaters." [Online]. Available: https://www.ansys.com/advantage-magazine/volume-xv-issue-1-2021/use-cfd-improve-efficiency-reduce-emissions

[36] Neural Concept, (2024), "Conjugate Heat Transfer: Best Practices & Applications." [Online]. Available: https://www.neuralconcept.com/post/conjugated-heat-transfer-best-practices-applications

[37] Ansys Innovation Space, (2025), "How to Setup Thermal Boundaries and Heat Sources in Ansys Fluent." [Online]. Available: https://innovationspace.ansys.com/courses/courses/heat-transfer-modeling-in-ansys-fluent/lessons/how-to-setup-thermal-boundaries-and-heat-sources-in-ansys-fluent-lesson-2/

[38] Chamil Abeykoon, (2020), "Design and analysis of a shell and tube heat exchanger," *International Journal of Heat and Mass Transfer*.

[39] E. Ozden and I. Tari, (2010), "Shell side CFD analysis of a small shell-and-tube heat exchanger," *International Journal of Thermal Sciences*.

[40] Becht Blog, (2024), “CFD Modeling of a Stoker Boiler,” Jun. 3. [Online]. Available: https://becht.com/becht-blog/entry/cfd-modeling-of-a-stoker-boiler/

[41] H. A. Majed, S. A. Abbas, and A. M. Al-Mousawi, (2025), “Evaluation of Carbon Dioxide Emissions and Thermal Efficiency of Al-Mussaib Thermal Power Plant,” *Iraqi Journal of Oil and Gas*, vol. 14, no. 1.

[42] O. R. Alomar, (2024), “Performance analysis of a steam power plant in Mosul, Iraq,” *Case Studies in Thermal Engineering*, vol. 56.

[43] M. R. Alsabbagh, (2026), “Concentration of Black Carbon Emissions From Natural Gas Flaring Activities in Basra Province, Iraq,” *Basrah Journal for Engineering Sciences*.

[44] Power Technology, (2024), “Al Dora Thermal Power Station,” Oct. 21. [Online]. Available: https://www.power-technology.com/data-insights/power-plant-profile-al-dora-thermal-power-station-iraq/

[45] E. J. Mahdi, A. H. Kareem, M. S. Nasser, and R. T. Salman, (2025), “Experimental comparison of aluminum (Al) and iron (Fe) radiators in a closed-cycle SWHS under Iraqi conditions,” *Energy Reports*.