

Study of Passive Cooling of a Thermal System Using ANSYS Fluent

By

Sahar Bayani Ghamsari

A final year project report submitted in partial fulfillment
of the requirements for the
Bachelor of Science Degree in Chemical Engineering
at Bahá'í Institute for Higher Education (BIHE)

Supervised by
Professor Farhang Sefidvash

March 2020

Abstract:

In the design of exothermic systems where heat is generated, the designer is concerned about whether the heat is adequately dissipated such that the equipment will not be overheated to damage the components of the system. Normally the heat is removed by forced convection. Since that involves active systems such as man-made pumps and control systems which have the possibility of failure, the inherent safety is sought after by utilizing the processes of natural convection and boiling that avoid the possibility of system failure being governed by the law of nature.

Here, as an example, the natural cooling of an exothermic helical tube is studied using the sophisticated ANSYS Fluent Computational Code. More specifically, the thermohydraulic behavior of water surrounding this helical system is studied by changing a wide range of design parameters of the helix. Note that in ANSYS Fluent there is no specific analytical solution for natural convection modeling but it emerges from direct solution of Navier-Stokes and energy differential equations.

With the big picture of inherent safety and passive cooling in mind, we simplified and narrowed down the objective of this undergraduate project by excluding boiling and only focused on the process of natural convection, while learning about the utilization of this complicated code and get know-how about the types of problems and phenomena that can handle.

With the aid of a wide range of analysis tools, such as contour plots, velocity vectors, and pathline plots, we were able to obtain optimum design parameters for maximum heat removal by natural convection. We also developed some comprehension about some system-related phenomena, such as chimney effect and thermal plume.

Finally, we employed the least square regression method (LSM) to fit our computational results with an analytical solution. As a result, we proposed a multivariable nonlinear equation to estimate the Nusselt number of our helical system.

Table of content:

1. Acknowledgments	5
2. Introduction.....	6
2.1. Convection heat transfer	6
2.1.1. Boundary layer of forced and natural convection	6
2.1.2. Applications of natural convection.....	7
2.1.3. Mathematical analysis of natural convection and the Grashof number	7
2.2. FBNR and inherent safety	9
2.3. Helical structure.....	11
2.3.1. Engineering considerations in the design of helical cooling systems	11
3. Review of the Previous Works	13
4. Methodology	17
4.1. Governing equations.....	17
4.1.1. Navier-Stocks equations.....	17
4.1.2. Energy equation.....	18
4.1.3. Turbulence modeling.....	19
4.2. Geometry preparation	21
4.3. Model discretization	22
4.4. Setting up the desired physics in ANSYS Fluent	25
4.4.1. Properties of materials added to the Fluent database	27
4.4.2. Cell zone and boundary conditions	29
4.5. Types of algorithms	30
4.6. ANSYS Fluent report definitions and convergence criteria	31
4.7. Verifying the accuracy of applied settings in simulation	32
4.8. A change in research objective	33
5. Results and Discussion.....	35
5.1. Transient results.....	35
5.2. The effect of tube thermal conductivity on heat flux	37
5.3. Effect of helix dimensions on heat flux.....	39
5.3.1. Effect of tube thickness and pitch	39
5.3.2. Effect of the inner diameter of the helix, D _{helix}	41
5.4. Velocity distribution	42

5.5. Effect of the thermal plume on heat transfer rate and surface Nusselt number.....	43
5.6. Horizontal helical tube.....	45
6. Conclusion	47
7. Recommendations and Suggestions for Future Work	48
8. Reference	49

1. Acknowledgments

My deep gratitude goes first to my supervisor, Professor Sefidvash, for his guidance and patience. My appreciation also extends to Soroush Ahmadi for mentoring the technical aspects of this project. I would also like to express my sincere appreciation for the Chemical Engineering Department of BIHE for providing me with the opportunity to study, and all the support they offered in my education process. Thanks also go to my friends for their genuine kindness and good humor. Last but not least, I would like to thank my family, especially my mother, who has been a constant source of love, support, inspiration and strength in all these years.

2. Introduction

2.1. Convection heat transfer:

Convection, one of the modes of heat transfer, occurs between a solid surface and a fluid phase adjacent to the solid surface. There are two types of convection, namely, natural convection and forced convection. Natural convection, also known as free convection, occurs due to fluid motion by natural means such as buoyancy. In forced convection, on the other hand, fluid is forced to move by external sources such as pump, fan, etc. A major difference between these two types of convection heat transfer is the fluid motion; higher fluid velocity in forced convection results in higher heat transfer rates, whereas in natural convection, velocities are noticeably lower and the heat transfer rates are consequently smaller. The advantage of using natural convection is that the fluid motion is generated naturally by density difference and buoyancy without the aid of any fluid mover [1].

2.1.1. Boundary layer of forced and natural convection:

If a vertical hot plate is located in a lower temperature fluid, a boundary layer profile will be observed near the surface. The boundary layer of natural and forced convection for this case are shown in Figure 1.a and 1.b. Temperature profile shows that the fluid temperature rises toward the plate. Since density and temperature are inversely proportional, the fluid next to the plate is lighter and starts to move vertically upward as a result of buoyancy forces.

In both forced and natural cases, the fluid velocity next to the plate is zero because of the no-slip condition. Nevertheless, the velocity profile increases monotonically in forced convection, but in the case of natural convection, the velocity is zero at the other end of the boundary layer due to the presence of stagnant fluid. The natural convection velocity profile is similar to that of an internal flow (Figure 1.c) with the difference that the velocity profile in an internal flow is symmetric, but for natural convection is not symmetric [1].

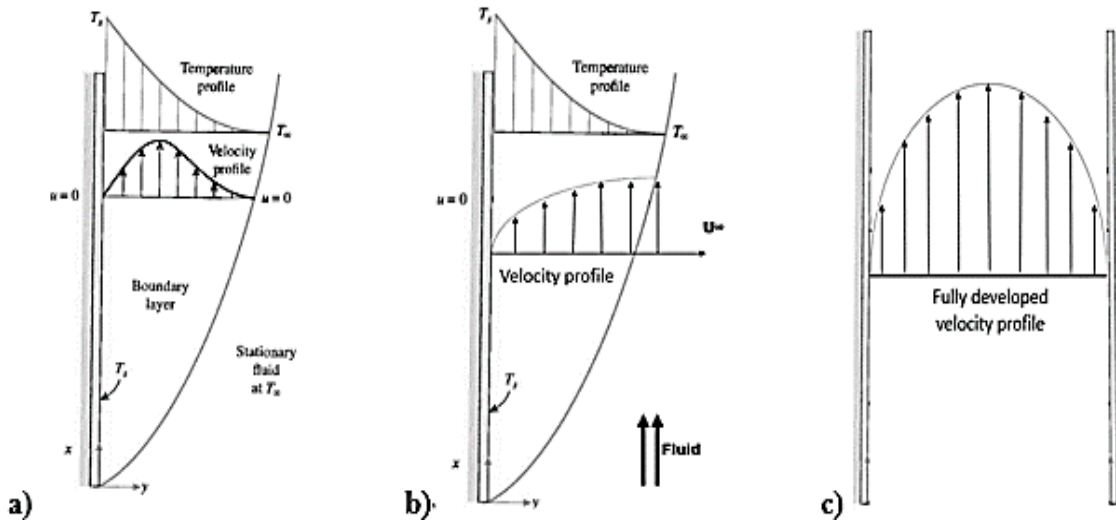


Figure 1. Boundary layer profiles. **a)** Velocity and temperature profiles for external natural convection flow [1]. **b)** Velocity and temperature profiles for external forced convection flow. **c)** Velocity profile for internal flow

2.1.2. Applications of natural convection:

Heat transfer by natural convection is of great importance in many industries and engineering applications. Cooling of electronic components, crystal growth, geothermal systems, heat exchangers, nuclear reactors, metallurgical processes, heat storage equipment, and waste heat recovery systems, just to name a few [2]. Natural convection has many advantages over the forced convection. It is reliable, cost- and energy-efficient, with no noise pollution. Although this method has many benefits, it should not be overlooked that the heat transfer coefficient is significantly smaller than the force convection. Thus, in order to fulfill an effective heat transfer rate, extensive design investigation is needed to obtain suitable shape and arrangement of the equipment to minimize the required surface area [3].

2.1.3. Mathematical analysis of natural convection and the Grashof number [1]:

Newton's second law of motion for a control volume can be written as equation 1.

$$F_x = \delta m \cdot a_x \quad (1)$$

Where δm is the fluid mass within the volume element and can be expressed as $\delta m = \rho(dx. dy. 1)$. Note that for simplicity, the equation is written for a 2D system ($dz=1$). a_x , the x-direction acceleration can be obtained by taking the total difference of velocity in x and y-direction and dividing it by dt:

$$a_x = \frac{du}{dt} = \frac{\partial u}{\partial x} \frac{dx}{dt} + \frac{\partial u}{\partial y} \frac{dy}{dt} = u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \quad (2)$$

Where $(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y})$ is a steady-state form of acceleration.

The next parameter in Newton's second law of motion is force. The forces acting on a differential volume element are pressure forces that vertically act on top and bottom of the volume element and shear stresses that act on side surfaces. Besides, gravity also acts on this element. So the net surface force acting in the x-direction can be written as:

$$\begin{aligned} F_x &= \left(\frac{\partial \tau}{\partial y} dy \right) (dx. 1) - \left(\frac{\partial P}{\partial x} dx \right) (dy. 1) - \rho g (dx. dy. 1) \\ &= \left(\mu \frac{\partial^2 u}{\partial y^2} - \frac{\partial P}{\partial x} - \rho g \right) (dx. dy. 1) \quad (3) \end{aligned}$$

The conservation of momentum equation in x-direction could be obtained from equations 1 to 3:

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = \mu \frac{\partial^2 u}{\partial y^2} - \frac{\partial P}{\partial x} - \rho g \quad (4)$$

The pressure gradient in the x-direction can be defined as equation 5 for quiescent fluid outside the boundary layer.

$$\frac{\partial P_\infty}{\partial x} = -\rho_\infty g \quad (5)$$

$$\rho \left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = \mu \frac{\partial^2 u}{\partial y^2} + (\rho_\infty - \rho) g \quad (6)$$

Non-dimensionalizing the simplified x-momentum equation results in equation 7.

$$u^* \frac{\partial u^*}{\partial x^*} + v^* \frac{\partial u^*}{\partial y^*} = \left[\frac{g \beta (T - T_\infty) L_c^3}{\nu^2} \right] \frac{T^*}{Re_L^2} + \frac{1}{Re_L} \frac{\partial^2 u^*}{\partial y^{*2}} \quad (7)$$

Based on the above equation, the Grashof number (Gr) is defined to represent the natural convection effects.

$$Gr_L = \left[\frac{g\beta(T-T_\infty)L_c^3}{\nu^2} \right] \quad (8)$$

This dimensionless parameter, named after Franz Grashof, is the ratio of buoyancy to the viscous forces acting on an element of fluid. Reynolds number (the ratio of inertia forces to the viscous forces) is extensively used in forced convection analysis. In the studies of natural convection, Gr number substitutes the role of the Re number [1].

2.2. FBNR and inherent safety:

The Fixed Bed Nuclear Reactor, FBNR, is a small nuclear reactor that works with Pressurized Water Reactor (PWR) technology. FBNR is modular in design and can be generalized to any size of reactor models. The schematic design of the FBNR can be seen in Figure 2.

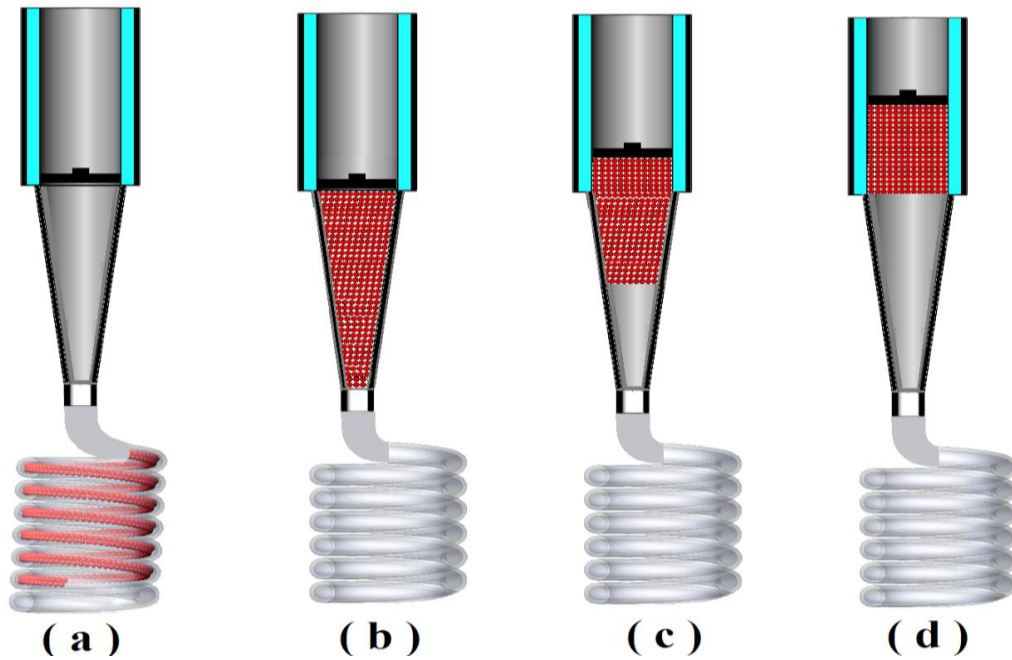


Figure 2. Operational states of FBNR. **a)** The fuel elements reside inside the fuel chamber under subcritical and passively cooled conditions. **b-d)** Fuel elements are pumped to the cone and cylindrical reactor core, respectively.

During the reactor operation, the spherical fuel elements are carried from the fuel chamber up into the core and held together by water flow, to form a suspended core. A pressurizer system is needed to keep the coolant fluid at constant pressure. Any signal due to any initiating accident will cut-off power

to the pump; the suspended core breaks down, and fuel elements fall back into the fuel chamber from the core by the force of gravity. At the moment of emergency, the fuel chamber gets filled up with water, and heat transfer occurs between fuel elements and water. Figure 3 shows the arrangement of the components of the passively cooled fuel chamber [4].

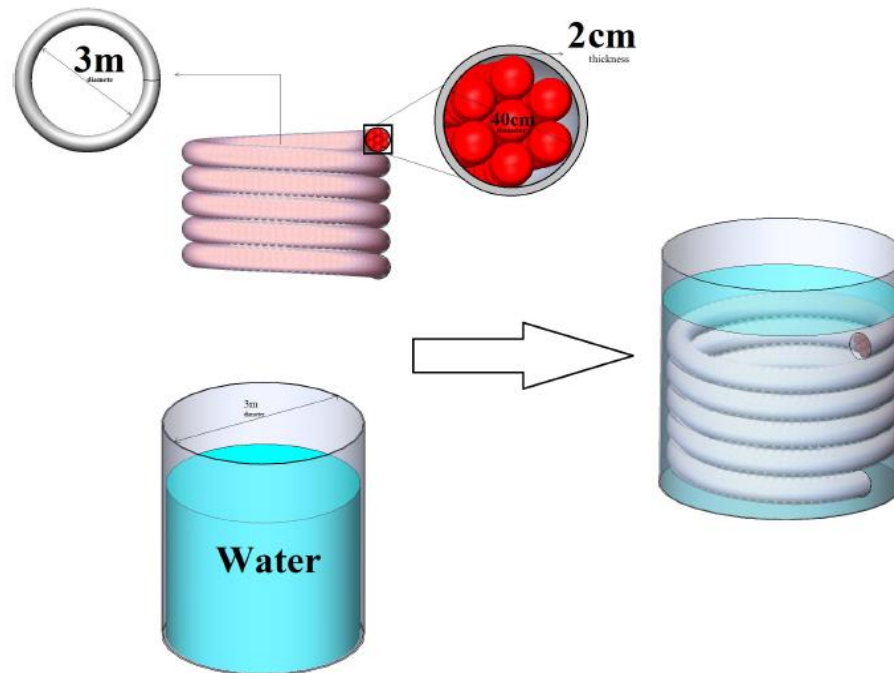


Figure 3. Schematic design of the passively cooled helical fuel chamber located in a water bath.

To ensure the reactor safety, the heat must be removed at an appropriate rate so that the increase in temperature will not damage the system. Active cooling systems commonly do the cooling. i.e. cooling through forced convection. However, in FBNR, the inherent safety or total safety is achieved by passive cooling through the process of natural convection, without any mechanical and human intervention or taking recourse to engineering devices. Since active cooling may fail, this novel approach is proposed to achieve inherent safety by taking advantage of infallible laws of nature [4]–[6]. One suggested design for the passively cooled fuel chamber of FBNR safety is a helical tube that will be discussed in detail.

2.3. Helical structure:

In any natural convection model, several important parameters directly control the rate of heat transfer. According to the convection equation, increasing the heat transfer surface or temperature difference increases the heat transfer rate. Therefore, the helical structure was chosen to increase the heat transfer surface area. Also, this structure is widely employed because of high heat transfer efficiency, convenient cleaning, easy processing and production and compactness. Helical structures have many applications in different industries, such as heating, ventilating, and air conditioning (HVAC) heat exchangers, condensers, and evaporators. However, relatively little work has been done on the natural convection heat transfer of these types of geometries [7]–[9].

2.3.1. Engineering considerations in the design of helical cooling systems:

One of the most important features of the helical structure, especially in the case of heat transfer, can be described in terms of a combination of two phenomena: Natural convection on the helix surface, as well as the additional fluid movements caused by the natural convection on lower turns, also known as thermal plumes. It is noteworthy that the thermal plume effect cannot only be described by the number of turns; the tube diameter and the pitch values also have a significant influence on the effectiveness of the thermal plume.

Another consideration in the design of helical systems is the occurrence of the chimney effect. According to Heo and Chung [10], the chimney effect is dominant for $1.5 < P/D < 5.0$ and increases with N . For other values of P/D ratio, the heat transfer behavior of the model changes. For example, when the P/D values are reduced, $P/D < 1.5$, the heat transfer rates are reduced due to the overlap of boundary layers.

In the design of helical heat removal equipment, the helix orientation is also of primary importance. It can be an either vertical, horizontal, or inclined helix, with different turning angles (θ). Several design parameters and their ratios play a significant role in heat transfer, including the diameter of the helical core, D_{core} , and the diameter of the tube, D_{pipe} , the total length, L , the helix height, h , pitch of the tube, P , the mean diameter of the helical structure, D_{helix} , and the number of turns, N . These parameters are illustrated in Figure 4.

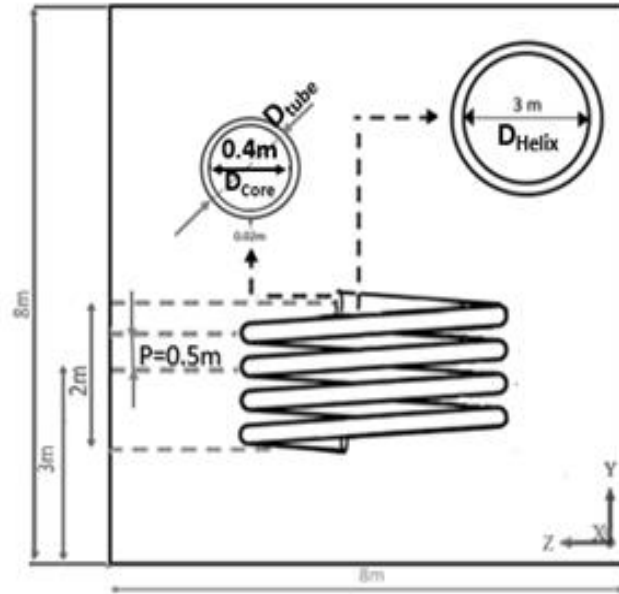


Figure 4. Schematic representation of the model and its physical dimensions

Natural convection heat transfer behavior on inclined cylinders is more complex than that of either horizontal or vertical cylinders due to the circumferential and axial development of the boundary layers. Based on various experiments, heat transfer in a horizontal cylinder is highest and decreases with an increasing slope of the cylinder compared to horizontal. This slope depends on parameters such as the pitch and diameter of the turns. One of the most important reasons that justify the low value of heat transfer in horizontal orientation is because the thermal plume effect is not expected [7].

Finally, preheating of the surrounding fluids on higher turns by lower turns is reducing the temperature gradient between solid and fluid and consequently as a diminishing effect on heat transfer rate. This is also directly related to P/D . For higher P/D ratios, the preheating effect decreases until it disappears at high critical P/D ratio ($P/D=5$) [7], [10].

3. Review of the Previous Works

Helical structures have been widely used in the design of heat exchangers. Heretofore, extensive research has been done on this structure and its heat transfer characteristics. In the majority of these studies, however, an external fluid mover, resulting in forced convection heat transfer, has been considered. Hence, there are a limited number of studies that focused on the natural convection on the outer surface of a helix [11], [12].

In 1994, an experimental study by Ali [13] dealt with the subject of natural convection on the outer surface of a helical coil that was submerged in water. According to this study, two correlations were presented for the Nusselt number based on Rayleigh number for two different diameters, equation 9 for $D = 0.012$ m and equation 10 for $D = 0.008$ m.

$$Nu_L = 0.685(Ra_L)^{0.295} \quad 3 * 10^{12} \leq Ra_L \leq 8 * 10^{14} \quad (9)$$

$$Nu_L = 0.00044(Ra_L)^{0.516} \quad 6 * 10^{11} \leq Ra_L \leq 1 * 10^{14} \quad (10)$$

The results of this study showed that higher tube diameters increased the heat transfer coefficient for the same Rayleigh number and tube length [13]. Furthermore, by using the height, H , as the characteristic length of the helix in Ra correlation, Ali proposed another correlation for Nusselt number, as shown in equation 11. In this case, both the pitch and tube diameter were considered.

$$Na_H = 0.257(Ra_H)^{0.323} \quad 6 * 10^8 \leq Ra_H \leq 3 * 10^{11} \quad (11) \quad [13]$$

Xin and Ebadian [14], in 1996, criticized that it is inexplicable to describe the different behavior of the helical structures with varying diameters of the tube in Ali's correlations. They determined the natural convection heat transfer coefficients on the outer surface, by using three different helicoidal tubes, both vertically and horizontally. In their experimental apparatus, a high DC current was passing through the tube wall, providing a constant heat flux boundary condition. In this case, the outer diameter of the tube was chosen as the characteristic length in the calculation of the Ra number. The average Nusselt number correlations, for vertical helical structures (equation 12), and for horizontal (equation 13), are shown below.

$$Nu_{o,d} = 0.290(Ra_{o,d})^{0.293} \quad \text{where } 4 * 10^3 \leq Ra_{o,d} \leq 1 * 10^5 \quad (12)$$

$$Na_{o,d} = 0.318(Ra_{o,d})^{0.293} \quad 5 * 10^3 \leq Ra_{o,d} \leq 1 * 10^5 \quad (13)$$

Comparing two above correlations for vertical and horizontal helical structures show that, in the laminar regime, the average heat transfer coefficient of the vertical coil is about 10% higher than for the horizontal one.

An assumption in Xin and Ebadian paper [14] that the middle turns of the coil represented the average behavior of the helix was criticized by Ali [15] in 1998. Moreover, he doubted the practicality of their correlation for the horizontal coil, since the end effects were not being taken into account. In his second study, Ali used four different horizontally oriented helical structures with the same tube diameter but with two different helical diameters, the number of turns, and heat flux values. Finally, he proposed several Nusselt number correlations for the different operating conditions. He concluded that equation 14 is valid for heat fluxes in the range of $1000 \leq q'' \leq 5000 \frac{W}{m^2}$.

$$Nu_x = 0.9125(Ra_x)^{0.301} \quad 2.5 * 10^3 \leq Ra_x \leq 5 * 10^6 \quad (14) \quad [15]$$

Another experimental study was done by Prabhanjan et al. (2004) on free convection over helically coiled tubes in water. They presented a number of Nusselt number correlations as a function of Rayleigh number based on the power-law equation (equation 15). In Rayleigh number calculations, different characteristic lengths were considered to obtain the most effective one. They concluded that the helical tube height is the best characteristic length to predict the Nusselt number.

$$Nu = a(Ra)^b \quad (15)$$

The values of a and b that are calculated in this experiment are given in Table 1 [12].

Table 1. Correlation results of Nusselt number. [12]

Characteristic length	a	b	Rayleigh range	Correlation coefficient
Tube length	0.0098	0.3972	$5 * 10^{14} - 3 * 10^{15}$	0.8684
Coil height	0.0749	0.3421	$9 * 10^9 - 4 * 10^{11}$	0.9306
Normalized length	2.0487	0.1768	$2 * 10^6 - 3 * 10^9$	0.9341

In another work (Izadpanah et al., 2018), seventeen helical coils are studied, and an empirical equation was presented for the Nusselt number as a function of Rayleigh number as well as coil parameters (H/D). As a part of the investigation, they also considered different characteristic lengths and found out that the coil length was the most suitable one for Rayleigh and Nusselt numbers. Equation 16 demonstrates the best correlation they found.

$$Nu_L = (Ra_L)^{0.3071} \left(\frac{H}{D}\right)^{-0.1097} \quad 5.01 * 10^{10} < Ra_L < 7 * 10^{11}$$

$$0.3167 < \frac{H}{D} < 1.9429 \quad (16) \quad [16]$$

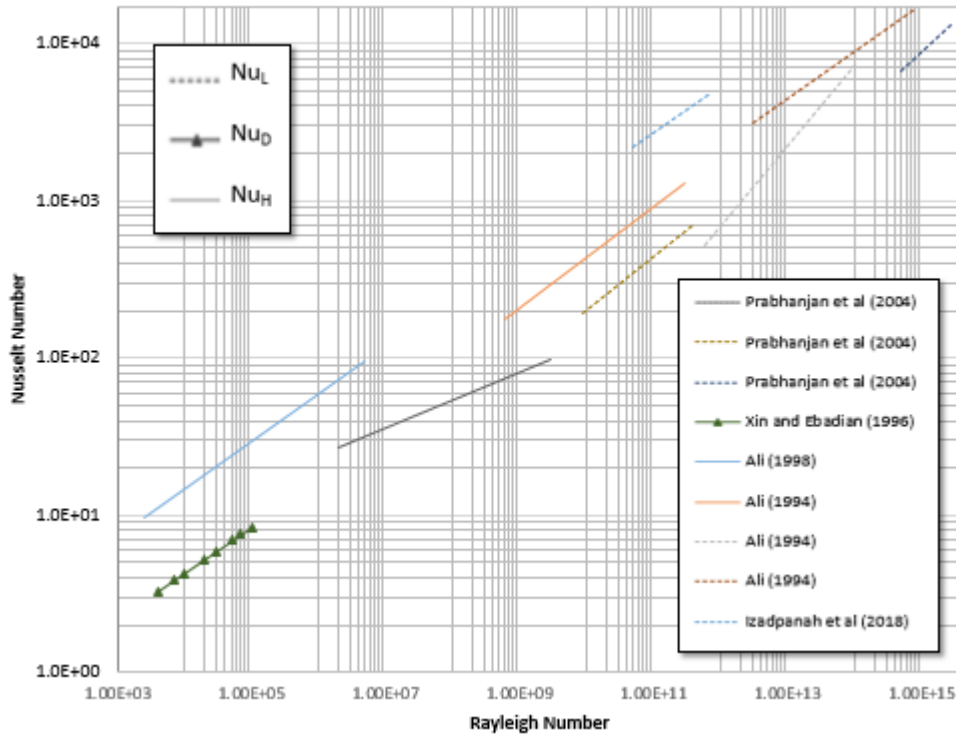


Figure 5. Nusselt Number equations from previous works

Beside experimental investigations and calculations, numerical and modeling techniques can also be used in analyzing the natural convection phenomena on helical structures. Nonetheless, only a few studies have been performed so far. In 2015, Neshat and Hossainpour [11] wrote their own CFD code in C language and simulated a helical coil, situated in a water bath, and studied the natural convection from the outer surface of them. This study aimed to model a transient natural convection system and

obtain the temperature of the bath and Nusselt number of the outer surface of a helical coil. For modeling the turbulence effects, SST $k - \omega$ turbulence model, a combined model of $k - \omega$ and $k - \varepsilon$, has been utilized. Statistical analyses were done on CFD results of about 27 different coils and a correlation formula was developed to calculate the fluid temperature. The advantage of this study is the developed correlation is valid not only for water but also for other fluids such as acetone, ethyl alcohol, etc [11].

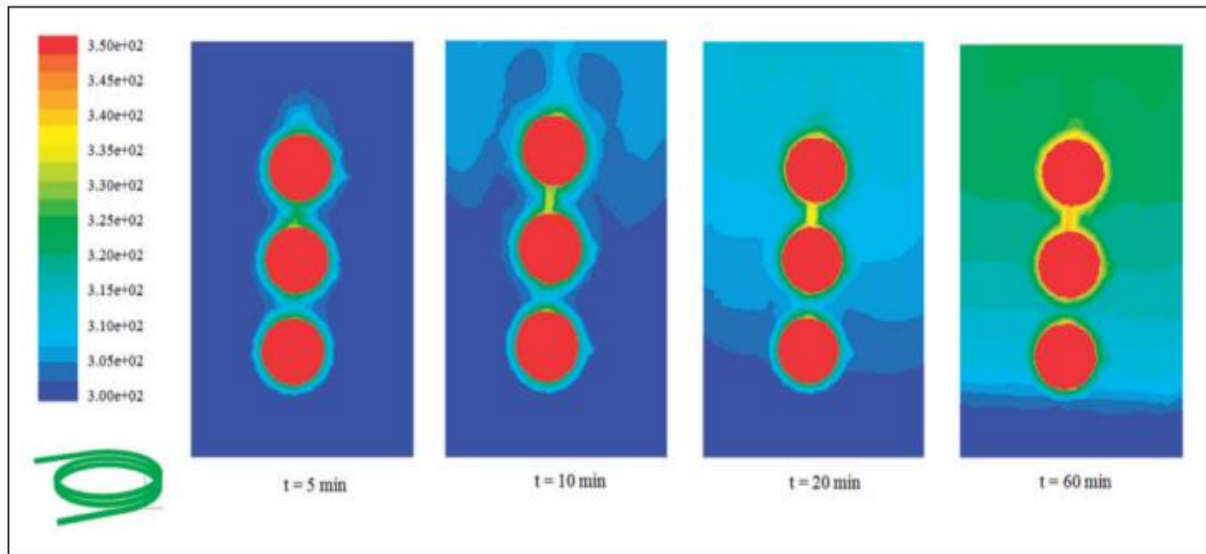


Figure 6. The time evolution of temperature distribution of water surrounding the helix [11].

On a broader view, others have studied different fluids (air [14], [15] and oil [17]) phase change phenomena [18], finned surfaces [19], and nano-fluids [20] but still our knowledge of this system is limited, and more detailed researches are required on the topic of outside natural convection heat transfer on the surfaces of the helical tube, involving different helical configurations and flow rates [12].

4. Methodology

In this study, we have used ANSYS Fluent code for our simulation purposes. ANSYS fluent is engineering simulation software, computer-aided engineering, CAE, and one of the most popular commercial CFD software packages. ANSYS Fluent has a vast number of capabilities that are developed by world-renowned experts and supported by extremely experienced engineers. ANSYS Fluent solves time-dependent Navier-Stokes equations using the finite volume methods (FVM) for the discretization of the computational domains [21].

In ANSYS Workbench, we started by selecting the type of our study “Fluid flow analysis (Fluent).” There are five main steps in system preparation and analysis of Fluent modeling, including geometry, mesh, setup, solution, and results. In the following sections, we will describe the details of our simulation setup.

4.1. Governing equations:

The governing equations of a mathematical model describe how the dependent variables change when one or more of the independent variables change. When heat transfer is also concerned, an additional equation of conservation of energy is also solved. Additional transport equations, with different degrees of complexity, are also solved when the flow is turbulent [22]. The explanation of these models are described below.

4.1.1. Navier-Stocks equations:

ANSYS FLUENT solves the following equations for mass balance, also known as the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (17) \quad [22]$$

This equation is the general form of the mass conservation equation that is valid for both incompressible and compressible flows. The source, S_m , is the mass added to the continuous phase from the dispersed second phase. In our modeling S_m is zero.

Conservation of momentum in an inertial reference frame is defined as,

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\tau}) + \rho \vec{g} + \vec{F} \quad (18)$$

Where p is the static pressure, $\bar{\tau}$ is the stress tensor, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces.[22]

For a three dimensional system we will have three independent momentum equations for x, y, z directions. The combination of continuity equation and these three momentum equations are commonly known as Navier-Stocks equations.

4.1.2. Energy equation:

In order to simulate a conjugated heat transfer problem, for example natural convection modeling, it is needed to consider the Energy equation. ANSYS Fluent solves the energy equation in the following form:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot (k_{eff} \nabla T - \sum_j h_j \vec{J}_j + (\bar{\tau}_{eff} \cdot \vec{v})) + S_h \quad (19)$$

where K_{eff} is the effective conductivity and $\vec{J}_j$ is the diffusion flux of species j. On the right-hand side of the equation, respectively, the first term represents conduction energy transfer, the second the species diffusion, and the third the viscous dissipation. Lastly, S_h includes the heat of chemical reaction [22].

4.1.3. Turbulence modeling:

Although turbulent flow is irregular and chaotic, they are governed by the Navier-Stokes equations [23]. This phenomenon is described by its characteristics as it is not possible to provide a complete definition of turbulent flows. The first characteristic of turbulent flow is that the fluid velocity field varies significantly and irregularly in both position and time. Due to this feature, there is no definite approach to solving turbulent problems, so it relies on statistical methods. The other feature is the ability to mix and increase rates of momentum much more effective than a comparable laminar flow. Turbulent flows occur at high Reynolds numbers.

Given that Re number is the ratio of inertia forces to viscous forces, the instabilities are due to the interaction of viscous terms and nonlinear inertia terms in the equations of motion. Since high levels of fluctuating vorticity characterize turbulent flows, the next important characteristic is rotational and three-dimensional vorticity fluctuations. Turbulent flows are always dissipative and continuum that is governed by the equations of fluid mechanics. At last, turbulence is a feature of fluid flows, not of fluids [24].

4.1.3.1. Turbulence models in ANSYS Fluent:

According to this fact that there is no universal, single acceptable turbulence model, the choice between these models depends on a number of considerations such as the physics of the flow, the level of accuracy required, the available computational resources, and the amount of time available for the simulation [25], [26]. ANSYS Fluent provides turbulence modeling capabilities in an unparalleled breadth of models. There are ten turbulence models available in the Fluent, described below.

For inviscid models, there are no viscous (friction) forces, so the Navier-Stokes equations reduces to the Euler equations, by removing the diffusion term [23]. The next simplest turbulence model is a one-equation model called Spalart-Allmaras that solves a modeled transport equation for the kinematic turbulent viscosity. Several versions of the popular $k-\varepsilon$ and $k-\omega$ models (2-equation turbulence models) are also available in Fluent. More advanced and complicated turbulence modeling such as Laminar-Turbulent Transition Models, transition $k-\omega$, transition SST, Reynolds stress model for highly anisotropic flows are also included in the Fluent. Finally, some state-of-the-art scale-

resolving turbulence models such as Detached Eddy Simulation (DES), Large Eddy Simulation (LES), and Scale- Adaptive Simulation (SAS) are also available in Fluent [27].

For all purposes, we suffice for two-equation turbulence models, namely k - ε RNG, which is discussed in more detail below.

4.1.3.2. The k - ε model:

The k - ε model is prevalent for external flow problems around complex geometries and also for industrial applications due to its reasonable convergence rate and relatively low memory requirements. However, it does not perform well in the areas of adverse pressure gradients, strong curvature to the flow, or jet flow. The basic idea of this model is to model turbulent kinetic energy, k , and dissipation rate of turbulent kinetic energy, ε , and then relate the missing information of length and time scales to these quantities to compute eddy viscosity as an algebraic expression of these two parameters. The k - ε model relies on the assumption that eddy viscosity can be connected to a time and length scale that indicates turbulence [28], [29].

The RNG k - ε turbulence model is a subcategory of k - ε model, derived from the Navier-Stokes equations, based on a mathematical technique called the renormalization group” (RNG).

The two additional partial differential equations for RNG k - ε models are shown below:

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) &= \frac{\partial}{\partial x_j} \left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \quad (20)\end{aligned}$$

Where parameters G_k and G_b represent the generation of turbulence kinetic energy, due to the mean velocity gradients for G_k and due to buoyancy for G_b . The contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate is determined by Y_M . The quantities α_k is the inverse effective Prandtl numbers for k and so α_ε is the inverse effective Prandtl numbers for ε . S_k and S_ε are user-defined terms [22].

4.2. Geometry preparation:

The geometry created using ANSYS DesignModeler is shown in Figure 7. The first step is to model a helical tube placed in a cube reservoir, as a water tank.

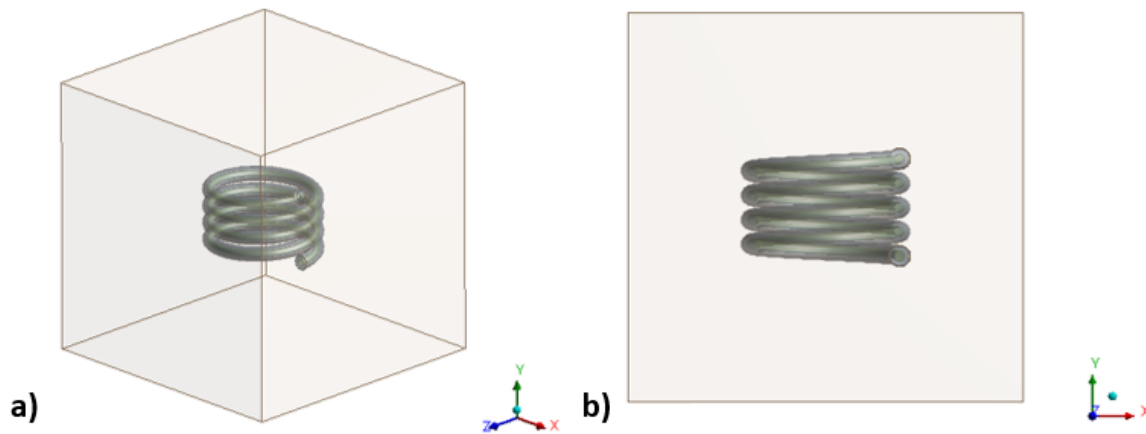


Figure 7. Different modeling orientations in DesignModeler

The dimensions of our main domain are $8*8*8 \text{ m}^3$ as shown in Figure 4. A considerable amount of this volume corresponds to the water around the helix. This had to be done to ensure the minimum boundary effects were imposed on the helix, and the numerical errors caused by the reverse flow on the outlet boundary condition are minimized.

Further, along the path of our simulations, we noticed that the volume of our computational domain was extremely large; even Though our model was divided to almost 7 million cells, the average volume of each cell was about 67 cm^3 . Consequently, we were not able to reach appropriate convergence for mass balance (the continuity equation). Unfortunately, our computing power was reached its limits, and we could not refine our mesh any further. Therefore, we decided to scale down our computational domain with a scale of 0.1.

With this modification, we were able to achieve better convergence: the convergence of the continuity residual that previously remained at about one has decreased to less than 10^{-2} under these conditions. The comparison between these two models is shown in Figure 8.

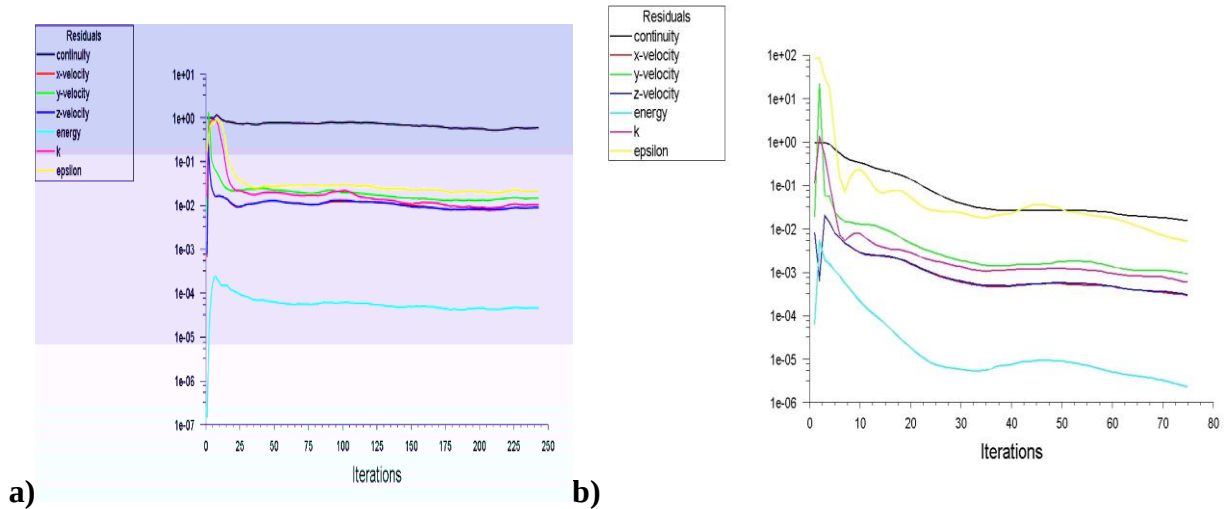


Figure 8. Residual plots. **a)** original model **b)** 0.1 scaled model

Note that size-independent parameters, such as heat transfer coefficient and heat flux, remain almost unchanged during this scale down process and can be utilized for analysis of the original model. For instance, the heat flux of the original model was $80.4 \frac{kW}{m^2}$ and the corresponding value for the scaled-down model was $79.3 \frac{kW}{m^2}$, with an acceptable relative error of 1.4%.

Therefore, our studies have been generally performed on the scale down model. After finding the optimum design of our helix, in order to obtain size-dependent parameters, such as the maximum core temperature, we switched back to the original model and ran a few essential simulations. With this hybrid approach, we were able to greatly reduce the computational cost of our study, without losing any meaningful precision.

4.3. Model discretization:

After generating the geometry in the DesignModeler, it is imported to the ANSYS Meshing to perform the model discretization. Various element sizes are used for different domains of the model, depending on their importance and sensitivity. Different meshing features such as body and edge sizing, inflation and sweep are used to improve the quality of our mesh. The tube outer wall surface, in contact with the cooling water, is a no-slip boundary. Therefore, the first layer of water that is touching the tube must be small enough to capture the proper velocity gradient for the fluid. Therefore, the inflation

feature is used to generate a very thin boundary layer adjacent to the wall. Figure 9 shows the final grid for scaled-down geometry, consisting of almost 6,000,000 elements.

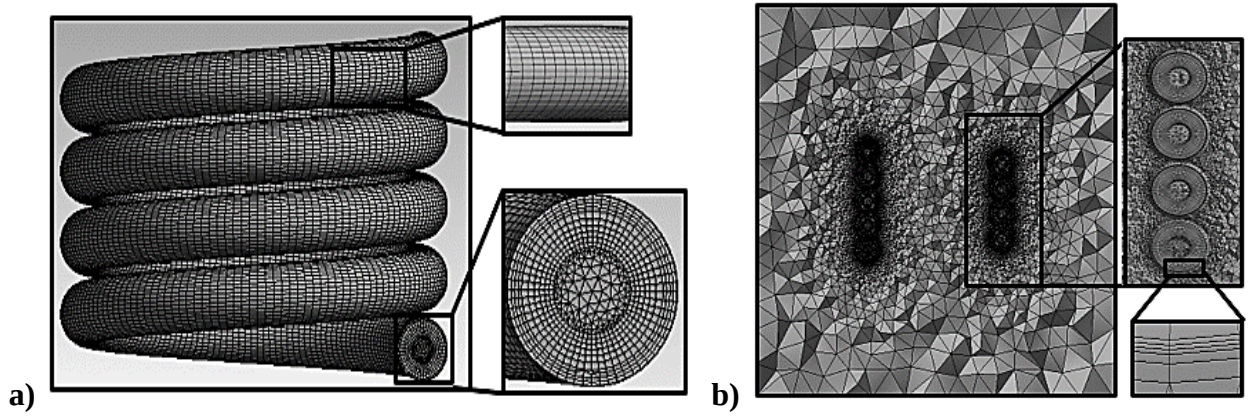


Figure 9. Discretized model **a)** The 3D solid helix domain **b)** The cut plot of the helix in water.

The boundary layer mesh is specifically important in our modeling, and this is remarkable since making very small changes in its size, many differences in the results. In order to accurately model our system, a fine near-wall mesh is required that is capable of resolving the viscous sublayer. One parameter that helps us determine the size of the boundary layer based on the model conditions is dimensionless wall distance that is also called y^+ , defined as [27]:

$$Y^+ \equiv \frac{u_* y}{\nu} \quad (21)$$

Where y is the distance to the wall and the local kinematic viscosity of the fluid is shown by ν , and u_* represents the friction velocity at the nearest wall and described by:

$$u_* = \sqrt{\frac{\tau_{wall}}{\rho}} \quad (22) \quad [27]$$

In our boundary layer meshing, we made the element size of the first layer (y) small enough ($\frac{1}{4}$ of the required y calculated from y^+) to capture the viscous sublayer.

After all of the considerations taken into account for meshing, we need to check the quality of the final mesh with the aid of ANSYS meshing reports. The quality of mesh plays an important role in the accuracy and stability of the numerical computation. Here, we examine the quality of our generated mesh. ANSYS Meshing offers various mesh metrics, including element quality, aspect ratio, skewness and orthogonal quality.

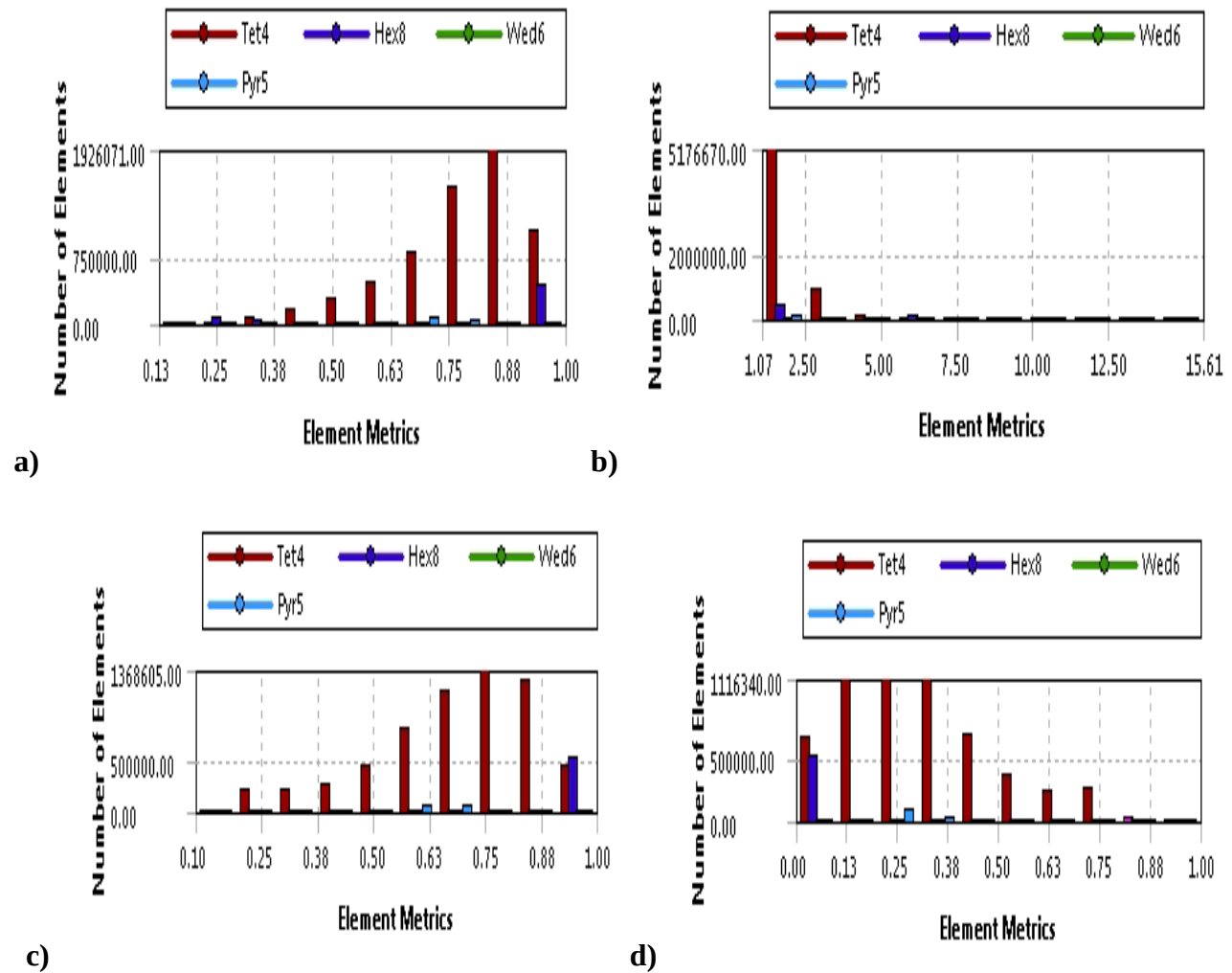


Figure 10. Different mesh metrics. **a)** Element quality **b)** Aspect ratio **c)** Orthogonal quality and **d)** Skewness

The element quality varies from 0 to 1, with one being the highest quality. By examining parts of the model that have poor element quality, shown in Figure 10.a, we found that they belong to the meshes around the tube, due to the inflation. However, the inflation was added intentionally to capture the velocity profile around the tube and cannot be considered as a flaw [22].

Aspect ratio, measure of the stretching of a cell, is another important indicator of the mesh quality. It is computed as the ratio of maximum value to the minimum value distances in a mesh. Distances were indicated are normal distances between the cell centroid and face centroids or nodes. Simply put, it is defined as the ratio of width to height. One is the ideal value for aspect ratio that represents perfectly cubic meshes. As one can see in Figure 10.b, the vast majority of our meshes possess reasonably small aspect ratios, indicating acceptable quality for this mesh metric [22].

The orthogonal quality also varies from 0 to 1, with one being the highest quality. If the minimum orthogonal quality is below 0.01, fluent will not accept the incoming mesh. For fast convergence, however, it is expected to have a significantly higher average value, at least 0.7. The minimum orthogonal quality for our modeling is 0.147, with an average of 0.774. These parameters are well within the accepted range for orthogonal quality [22].

Skewness also varies from 0 to 1, with zero being the highest quality. It is defined as the difference between the shape of the equally-angled triangular/tetrahedral mesh, and the real mesh. Skewness for both triangular or tetrahedral meshes should be kept below 0.95 with a significantly lower average value. Higher values for skewness may lead to convergence difficulties [30]. As can be seen in Figure 10.d, the skewness of our mesh ranged from 0.04 to 0.82, indicating acceptable skewness [22].

In Conclusion, the consideration of all four primary criteria for mesh quality has ensured that we have discretized our model sufficiently well.

4.4.Setting up the desired physics in ANSYS Fluent:

After generating the mesh and creating a named selection for all volume and surface bodies of the model, the mesh is updated and translated to Fluent. By clicking on set up, Fluent launcher opens in order to set some basic settings. The double-precision option allows for increasing the decimal place from 8 numbers to 16. We activated this option for a better convergence of the natural convection modeling. Thermal expansion coefficient, β , is usually a very small number but plays a significant role in natural convection modeling; using single precision may severely round up this parameter due to its small value and causes substantial errors in the results. At last, for parallel computing, the number of processors can be chosen to speed up the calculations. We used all 12 available cores of the CPU.

In the general section of the Fluent setup, the type of the solver is selected as “pressure based” because this type is generic. In contrast, the “density-based” type is typically used for sonic and supersonic velocities and highly compressible fluids.

Each system can be modeled either as a steady-state problem or a transient model. A major challenge in transient thermal design is the time scale. Natural convection processes occur typically in hour

scales. While the common commercial codes such as ANSYS Fluent tends to solve very small time steps due to the numerical stability requirement for natural convection. The issue of convergence in transient models is another challenge. According to Fadi [31], neither steady natural convection nor an unsteady forced convection solution seems to have problems with convergence, while unsteady natural convection solutions are subject to restriction for convergence. Therefore, we decided to solve the model in a time-independent and steady-state manner.

Modeling gravity is a must in natural convection modeling, since the fluid motion is created as a result of changes in density with temperature and, consequently, the buoyancy effect. Therefore, gravity is enabled and the gravitational acceleration of $-9.806 \frac{m}{s^2}$ is entered in the y-direction due to the orientation of the model in the task page.

The water in the bath is initially stationary, and there is no need for turbulence modeling. When the fluid starts to equilibrate with the helix, significant changes in density occur, velocity fluctuates, and the flow becomes exceptionally disordered. Here the fluid enters the turbulent regime. The point is fluid does not enter the turbulent zone suddenly. Instead, there is a transition state before it becomes fully turbulent.

The existence of these laminar, transition and turbulent flow regimes can be seen in different phenomena, from cigarette smoke to dye injection in a glass tube. Fluctuations in velocities enhance the rate of heat and momentum transfer between fluid particles and finally would increase the friction force on the surface and the convection heat transfer rate [1].

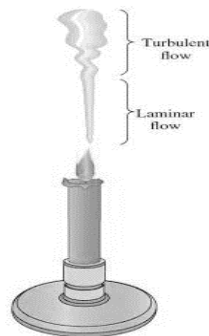


Figure 11. The laminar and turbulent flow regime of candle smoke [1].

Furthermore, our system of interest has the Rayleigh number of 10^{11} , within the known range of turbulent natural convection [22].

4.4.1. Properties of materials added to the Fluent database:

Water is available in the Fluent database, but its properties, including density, heat capacity, thermal conductivity, and viscosity are all constant values with no variation with temperature. Natural convection occurs only due to the density variations of fluid to make buoyancy-driven flows. As shown in Figure 12, if we assume the density is constant, heat can only be transferred symmetrically by the mode of conduction.

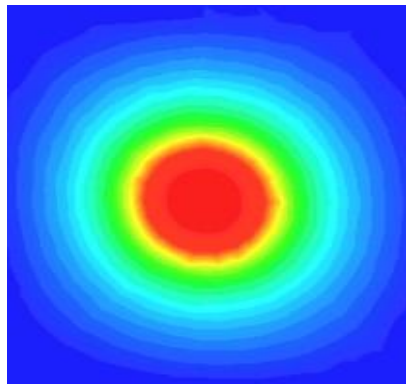


Figure 12. Convection flow of incompressible fluids

If the displacement in the fluid does not occur due to the constant density, the hot object is always exposed to the same high-temperature fluid and loses its heat transfer efficiency.

Hence, it is apparent that for natural convection modeling we have to consider the fluid properties, including density, as a function of temperature. To achieve this goal, piecewise-linear functions for all water properties for the range of temperatures from 10 to 120 have been added to the database. The imported data for water was extracted from [1] and can be found in Table 2 below.

Table 2. Variable thermal and physical properties of water [1]

Data point	Temperature (°C)	Density ($\frac{kg}{m^3}$)	Viscosity ($\frac{kg}{ms}$)	Specific heat ($\frac{J}{kg.K}$)	Thermal conductivity ($\frac{W}{mK}$)
1	10	999.7	$1.307 * 10^{-3}$	4194	0.580
2	20	998	$1.002 * 10^{-3}$	4182	0.598
3	30	996	$0.798 * 10^{-3}$	4178	0.615
4	40	992.1	$0.653 * 10^{-3}$	4179	0.631
5	50	988.1	$0.547 * 10^{-3}$	4181	0.644
6	60	983.3	$0.467 * 10^{-3}$	4185	0.654
7	70	977.5	$0.404 * 10^{-3}$	4190	0.663
8	80	971.8	$0.355 * 10^{-3}$	4197	0.670
9	90	965.3	$0.315 * 10^{-3}$	4206	0.675
10	100	957.9	$0.282 * 10^{-3}$	4217	0.679
11	110	950.6	$0.255 * 10^{-3}$	4229	0.682
12	120	943.4	$0.232 * 10^{-3}$	4244	0.683

For solid materials, such as the helix core and its surrounding tube, three parameters must be defined: density, thermal conductivity, and heat capacity. The tube is assumed to be stainless steel 304 that has known properties. The core, on the other hand, consists of the FBNR fuel element with limited availability of its properties (only its thermal conductivity was found in [5]). With a closer look at the conservation of energy equation, we noticed that the density and heat capacity of solids only appear in the time-dependent terms of the equation. Since we are only interested in the steady-state behavior of this system, we speculated that no difference would be seen in the final solution while changing solid densities and heat capacities. To check this hypothesis, we run the model many times, changing the unknown solid properties radically, but noticed that the final solution remained unchanged.

The summary of the solid properties can be found in Table 3.

Table 3. Summary of material properties of the helical core and tube assumed in numerical simulation

Material	Density ($\frac{kg}{m^3}$)	Specific heat ($\frac{J}{kg.K}$)	Thermal conductivity ($\frac{W}{mK}$)
Stainless steel	7500	500	16.3
Fuel element	-	-	12.5

4.4.2. Cell zone and boundary conditions:

In the cell zone condition, appropriate material (water, stainless steel, or FBNR fuel element) is assigned to different domains. Additionally, in the cell zone condition, it is possible to define frame and mesh motion, porous zone, fixed values, and source terms. In our simulation, we just deal with the source terms option to model heat generation.

The next step is defining boundary condition settings. Four sidewalls of the water bath are described as walls at 20C. The bottom side of the domain is defined as a velocity inlet boundary condition. To ensure that heat is transferred by natural convection, we choose a much smaller value of velocity (0.001 m/s) for this boundary compared with the velocity that is created due to natural convection. As shown in Figure 13, the fluid velocity caused by natural convection is in the range of 0.1 m/s, which is 100 times higher than the inlet velocity. The boundary condition for the upper side of the domain is of the pressure outlet type, which is set to atmospheric pressure (the relative pressure of 0 Pa).

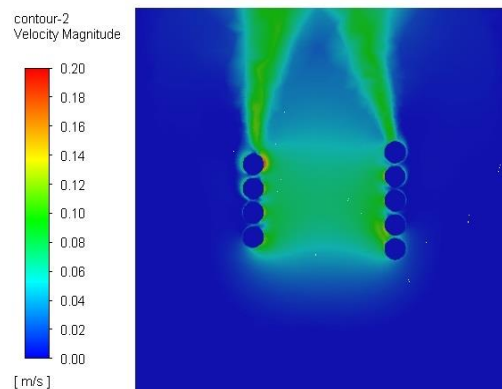


Figure 13. Velocity contour

The helical tube is divided into two contact surfaces, the inner contact surface is the helical core surface that is in contact with the inner surface of the tube, and the outer contact surface is the external tube surface that is in contact with water. Both the helical core wall and tube wall are defined as coupled boundary conditions. So, “wall shadows” boundary conditions are created automatically by ANSYS for tube wall and core wall. Given that, the wall surface of the helical tube and also the domain walls are set as no-slip boundaries.

4.5. Types of algorithms:

The pressure-based solver employs an algorithm, wherein a pressure equation is solved to achieve mass conservation of the velocity field. This pressure equation is derived from the Navier-Stokes equations in such a way that the corrected velocity field by the pressure satisfies the continuity equation. Due to the non-linear nature of the governing equations and the interdependency of variables, an iterative approach is needed, wherein the governing equations are solved repeatedly until the solution converges. [22]

There are four types of algorithms in ANSYS fluent to achieve pressure-velocity coupling: SIMPLE, SIMPLER, PISO, and Coupled. The default algorithm in ANSYS Fluent is set to the SIMPLE algorithm. This algorithm is generally used for steady-state calculations [22]. One of the approaches we chose to ensure the accuracy of settings and mesh quality was to change these algorithms and compare the results. Since changing the algorithm must not affect the final results, the observation of dissimilarities between the results of different algorithms, is an indication of an ill-conditioned setup and subsequent convergence problems. Our investigations showed that the setup is independent of the algorithm. Note that during the initial stages of our research, when the boundary layer was not set up carefully, considerable variation in results was observed for different algorithms; these inconsistencies were significantly reduced after proper modifications [20].

The results of two different algorithms, namely SIMPLE and Coupled, for a case study are shown in Table 4, indicating insignificant variations less than 3 percent, in parameters of interest.

Table 4. Results obtained from different methods.

Method	$h(\frac{W}{m^2K})$	$T_{max,core}$ (°C)	$T_{max,surface}$ (°C)
SIMPLE	1117.7	138.9	96.5
Coupled	1089.3	140.2	99.1
% Error	2.60%	0.93%	2.62%

4.6. ANSYS Fluent report definitions and convergence criteria:

The parameters needed to conclude and analyze the model have to be defined in the Report Definition section. In our studies, the numerical error of mass imbalance, the convective heat transfer coefficient on the helical tube surface, the surface Nusselt number, the maximum temperature of the helical core and the tube surface temperature are defined.

ANSYS Fluent uses the iterative method to numerically solve the governing differential equations for each cell in the system. Evidently, the initial guess of this iterative method (known as initialization) is far from the exact values. Therefore, it is crucial to check whether the solution is converging to a physically meaningful solution or not. By default, the residuals of variables such as mass, x-, y-, z- momentums, energy, and $k-\varepsilon$ are considered as convergence criteria. To be exact, Fluent will stop the simulation after each, and every one of the above-mentioned criteria reaches 0.001. The definition of the residuals in Fluent is shown below:

$$Res - m(1) = \frac{[abs(m(n) - m(n-1))]}{m(n)}$$

$$Res - m(2) = \frac{[abs(m(n) - m(n-2))]}{m(n)} \dots$$

$$Res - m(Np) = \frac{[abs(m(n) - m(n-Np))]}{m(n)}$$

Where: Res-m is the monitor residual [30].

Although this is a powerful and straightforward tool to check the convergence of simulations, it is not sufficient. We must define a number of system-dependent criteria, for instance, the heat transfer rate in our case, to ensure only insignificant changes in those parameters are observed at the end of the simulation.

The set of desired variables of q , h , and T for the helical tube surface, as well as mass imbalance and maximum temperature of the helical core, were decided to check the convergence of our solution.

4.7. Verifying the accuracy of applied settings in simulation:

Even if the solution converges, it may be far from reality. Thus, further investigations are needed to guarantee our modeling has been conducted well enough to represent a real physical problem. This can be done by two approaches, known as verification and validation.

In verification, we check the sanity of our results with common sense, engineering judgment, and simple approximate hand calculations. In the validation stage, however, we need extensive experimental evidence to back up our results.

For verification purposes, we started with hand calculations. Unfortunately, well-known and reliable relations for helical geometry were not available in engineering handbooks. Inevitably, a simpler geometry (cylinder) with well-established formulas were selected and modeled. The empirical and accurate Nusselt number relation for a horizontal cylinder is written below:

$$Nu_{cylinder} = \left\{ 0.6 + \frac{0.387 Ra_D^{\frac{1}{4}}}{\left(1 + \left(\frac{0.559}{r} \right)^{\frac{9}{16}} \right)^{\frac{8}{27}}} \right\}^2 \quad (23) \quad [1]$$

The Rayleigh number range at which this correlation can be used is less than or equal to 10^{12} . The diameter of the modeled cylinder was the same diameter as of the helix, 40 cm. The Rayleigh number relation for this geometry is given in equation 23. Based on the selected diameter, the Rayleigh number is within the permissible range of the above Nusselt number correlation.

By comparing a set of parameters, between the ANSYS Fluent modeling results and the hand calculations, we showed that reasonably acceptable accuracy was achieved. This set of parameters consists of heat transfer coefficient, total heat transfer rate, the maximum temperature of the center of the cylinder, and contact surface temperature between the cylinder and water. The quantitative summary of this comparison can be found in Table 5.

Table 5. Results obtained for sphere and cylinder models.

Geometry	$h(\frac{W.m^2}{K})$	% <i>Error</i>	$q (W)$	% <i>Error</i>
Cylinder	1.01e+03	30	1.23e+05	15.4

4.8. A change in research objective:

Before starting the result and discussion part of the thesis, we had to set up a goal for our case study. A thorough examination of our modelling setup pushed us to redefine our research objective, as explained in detail below.

Initially, this project aimed to design an appropriate helical fuel chamber for the FBNR reactor and investigate the conditions under which the sufficient heat removal occurred by passive cooling and the inherent safety of the system during the emergency shutdown was achieved. To model this condition, we had to estimate the amount of heat generated by decay after shutdown. It is well known that the radiation decreases exponentially with time, as shown below.

Table 6. Decay heat reduction [32].

time	power
0	7%
1 hour	1%
1 day	0.4%

In order to model the system at a steady-state condition, we considered the maximum amount of heat generation at the time of shutdown, which is 7% of the reactor operating condition. During the reactor operation, the heat generation by FBNR fuel elements is assumed to be 33 Wcm³ (70 MW total). The safety condition for this part is to keep the maximum temperature of the fuel elements under the value of the safety parameter that is 1200°C [32]. On the other hand, the helical tube packing, defined as the ratio of fuel elements to the total volume, is 60%, and the remaining 40% of the total volume of the helical tube is void space.

For our simulation purposes, we assumed that the internal volume of the tube is homogeneously filled, and individual fuel elements are not modeled. Note that 1,200,000 small spherical fuel elements are present in the fuel chamber, and modeling them individually is impractical. The rate of heat generation per unit volume of the fuel chamber is reduced to $1.4 \frac{W}{cm^3}$.

We performed a short simulation with $1.4 \frac{W}{cm^3}$ heat generation. By checking the temperature of the outer wall of the helix, we found out that the temperature of the water had risen above its boiling point. Therefore, it became clear that some changes in the modeling process were needed. The advanced modeling of such a system required the consideration of the phase change (boiling) that was out of the scope of an undergraduate thesis. Hence, we modified the thesis statement to “how much heat can be extracted from the helix, with the constraint that the wall temperature must not exceed 100C?”. This modification helped us to only focus on the natural convection process and avoid the complexity of a multiphase simulation.

In order to find the proper heat generation rate that keeps the surface temperature below 100 degrees, was changed from constant q boundary condition (the second type of boundary condition, called Neumann boundary conditions that specify the normal derivative of the function on a surface) to constant temperature boundary condition (the first type, called Dirichlet boundary conditions that specify a constant value of the function on a surface) [1] [33].

5. Results and Discussion

5.1. Transient results:

Steady-state modeling takes considerably less computational time than the transient method. So, all the reported results of our study in the upcoming sections are obtained from a converged steady-state solution. One practical approach to verify the accuracy of the model and ensure that the physics are modeled correctly is to compare the steady-state results with transient one for a single case study. Provided that the results of the transient modeling approach the steady-state results at long simulation times, we can be more confident about the accuracy of the steady-state results.

A major challenge in transient thermal design is the time scale. Natural convection processes occur typically in hour scales. While the standard commercial codes such as ANSYS Fluent tends to solve very small time steps due to the numerical stability requirement for natural convection. The issue of convergence in transient models is another challenge. According to Fadi [31], neither steady natural convection nor an unsteady forced convection solution seems to have problems with convergence, while unsteady natural convection solutions are subject to restriction for convergence [31], [34], [35]. To achieve numerical stability for our simulation, we started with small time steps (1 second) during the initial iterations and increased that to 10 seconds after reaching some stability and consistency.

The results of transient modeling are presented below. Figure 14 shows the velocity contours at different times. At the beginning and after the initializing the model, velocity all over the domains is equal to the input velocity, 0.001 m/s, except near the walls, which is zero due to the no-slip condition. Over time, the changes in velocity and turbulence in the model start to increase the velocity around the helix and finally reaches steady-state condition at 5000 seconds (80 mins). We can see the velocity contour at 80 mins is very similar to that of the steady-state case (Figure 13). Note that the computational time required for the transient modeling was ten times longer than the steady-state modeling. Still, the same results were achieved, legitimizing our decision to continue with steady-state modeling for the rest of the analyses.

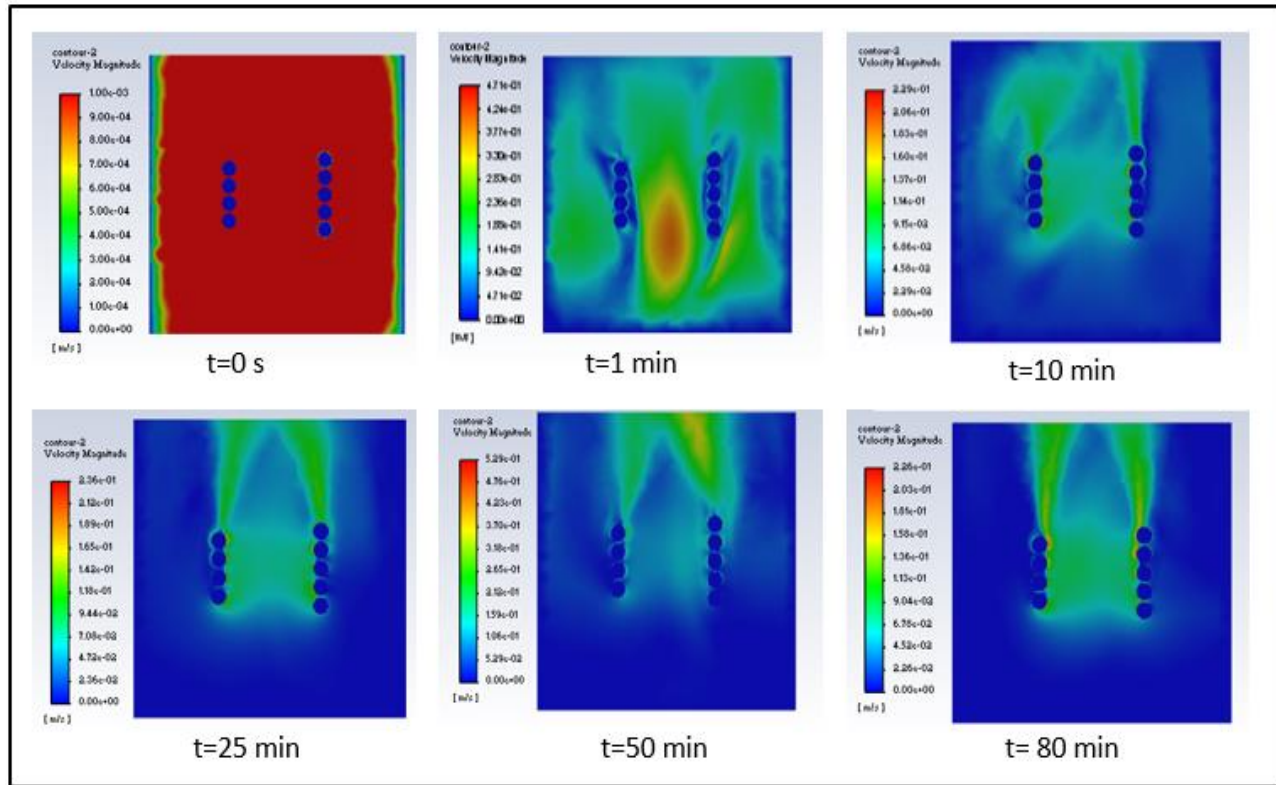


Figure 14. The time evolution of velocity distributions caused by natural convection around the helix.

For a quantitative comparison between the transient and steady-state modeling, two similar models with constant wall temperatures, 100C, are solved, and the outcome is presented in Table 7. Note that both convection heat transfer coefficient (h) and the total heat flux (q) differ only by less than 3 percent between the two models.

Table 7. The comparison of heat transfer coefficient (h) and total heat flux (q) in transient and steady-state models.

Model	$h \left(\frac{W}{m^2 \cdot K} \right)$	$q \left(\frac{W}{m^2} \right)$
Transient	1.168e+03	99270.7
Steady state	1.197e+03	101771.8
% Error	2.4%	2.5%

5.2.The effect of tube thermal conductivity on heat flux:

Before starting the analysis of natural convection analysis, it is worth studying the conduction through the tube. In our model, we have three domains, heat generation in the core, heat conduction through the tube wall, and finally, natural convection on the outside. The resistance caused by the wall conduction can hinder the heat transfer by natural convection. In other words, if the thermal conductivity of the tube wall is considerably small, it will be a liability of the whole system. Since our objective is to study the maximum amount of heat that can be removed by the natural convection, it is essential to eliminate the resistance of the wall as much as possible.

To have a quantitative understanding of the effect of tube conduction on the total heat flux, nine different thermal conductivities have been selected, ranging from 1 to $100 \frac{W}{m.K}$. After simulating each case, the total amount of heat flux has been extracted and shown in Figure 15.

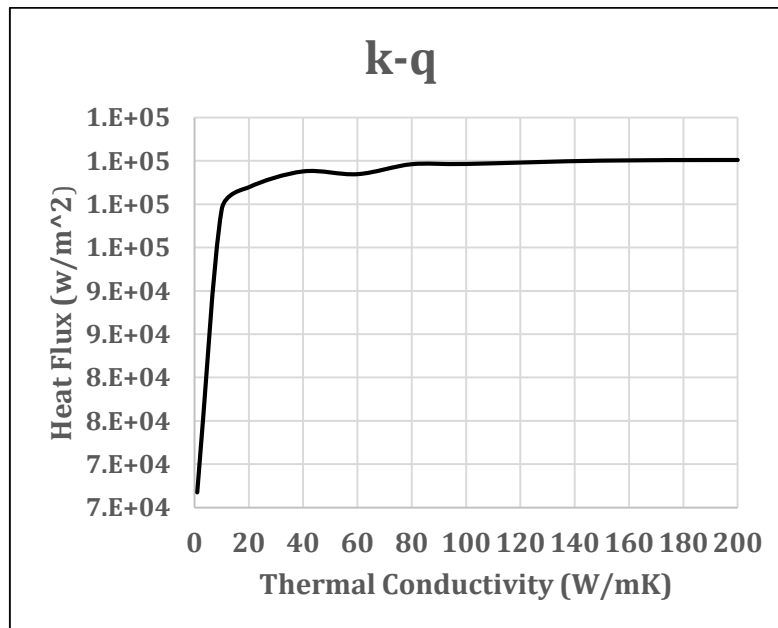


Figure 15. Effect of thermal conductivity on the total heat flux.

As illustrated in Figure 15, the hindrance effect of thermal conductivity of the tube on the total heat flux is significant for lower thermal conductivities. As the thermal conductivity increases gradually, the total amount of heat flux also increases, and finally levels off for values higher than $10 \frac{W}{m.K}$.

The concept of thermal resistance can justify this observation. Thermal resistance depends on the geometry and thermal properties of a medium. Ignoring the curvature of the tube, the resistance of the tube can be simplified and approximated as a cylinder. The equation for thermal resistance (R) of a cylinder can be expressed as equation 24.

$$R_{cyl} = \frac{\ln\left(\frac{r_2}{r_1}\right)}{2\pi Lk} \quad (24) \quad [1]$$

Therefore, it is evident that the thermal resistance (R) is inversely proportional to the thermal conductivity of the medium (k). At higher values of k, the denominator enlarges, and consequently, the thermal resistance decreases. At some point, the thermal resistance approaches zero and, therefore, the effect of thermal conductivity becomes insignificant.

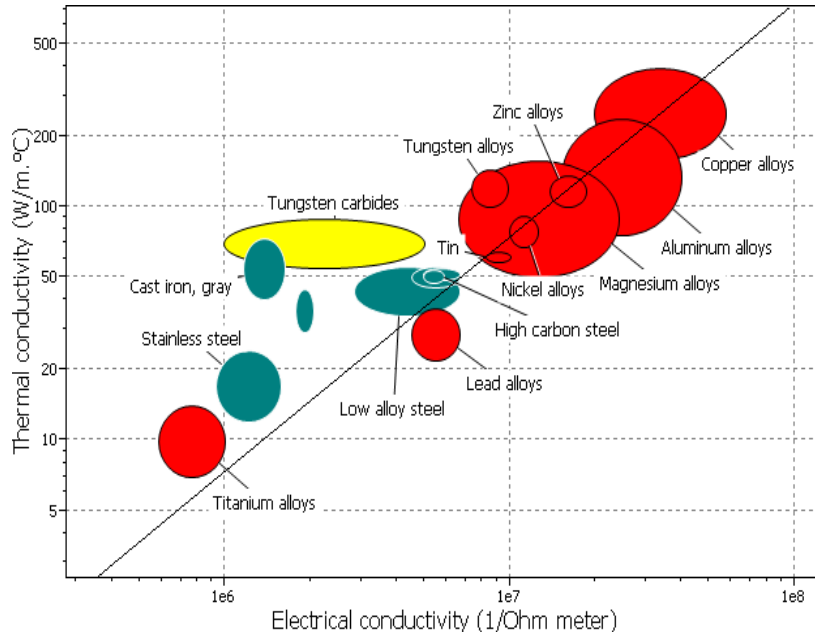


Figure 16. Thermal conductivity of different metals and alloys [36].

As shown in Figure 16, almost all metal alloys, except some titanium alloys, possess thermal conductivities above $10 \frac{W}{m.K}$. Since in our model the thermal conductivity of stainless steel ($16.3 \frac{W}{m.K}$) is used, we can be confident that the conduction heat transfer has no significant effect on the total thermal resistance of our model.

5.3.Effect of helix dimensions on heat flux:

Except for the size of the water bath surrounding the helix, the impact of other dimensions, including the thickness of the tube, the pitch and the mean diameter of the helical structure, on the heat flux are examined here.

5.3.1. Effect of tube thickness and pitch:

As far as the effect of tube thickness and its pitch on the natural convection is concerned, they directly change the spacing between two consecutive turns of a helix, causing dramatic changes in the flow field around the helix. This effect can be seen in Figure 17, where any change P or T will directly change the S . Therefore, it is evident that the independent investigation of the effect of pitch and the tube thickness on heat transfer is not feasible. To analyze the impact of these two parameters on the heat flux, 13 different models with various thicknesses (0.02 to 0.04 m) and pitches (0.5 to 0.9 m) were studied. The results of this analysis are shown in Figure 18.

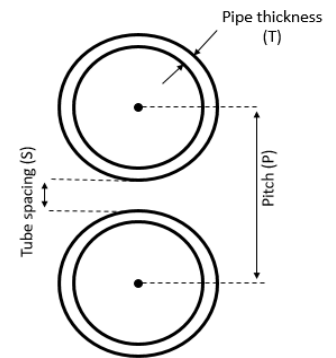


Figure 17. Schematic representation of P , T , and S .

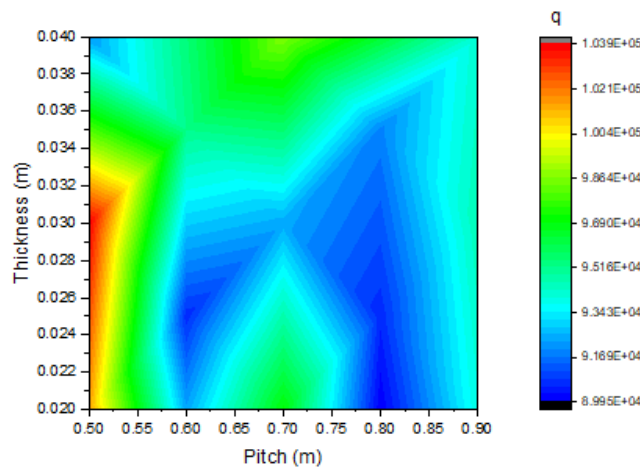


Figure 18. Effects of pitch and thickness on the heat transfer rate.

As shown in the contour above, the most amount of heat was removed from the helix in which the wall thickness was 0.03 m, and the pitch was at its lowest (0.5 m).

By checking the velocity vectors of these models, we noticed that by increasing the pitch, the chimney effect decreased, and the flow in the inner loop of the helix was able to escape and cool down more quickly. This observation can explain why the lowest pitch has the highest heat flux. Through a literature survey, we noticed that Shouwen et al. [8] also noted that with the reduction of the pitch, the heat transfer effect is enhanced. They related this trend to the enhancement in the turbulent effect caused by the spiral heating tube rotating structure.

Regarding the thickness, however, we tried to explain why 0.03 m led to the highest heat removal, but we could not find a reasonable explanation. One possible explanation was related to the concept of the critical radius of insulation. In cylindrical and spherical solids, the thermal resistance contains two competing factors; as the radius of the object increases, the conduction resistance also increases. However, the increase in radius provides a higher surface area for the convection and subsequently decreases the resistance. If we plot the heat transfer as a function of the outer radius of the solid, shown in Figure 19, we obtain a bell-shaped graph with a maximum Q . The radius where the maximum heat is observed is known as the critical radius and can be calculated from equation 25.

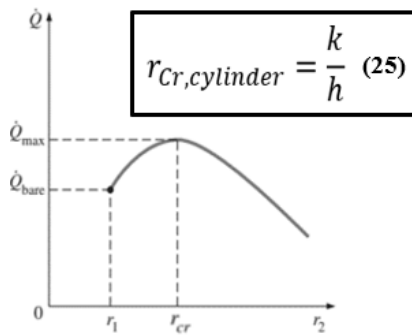


Figure 19. The variation of heat transfer rate with the outer radius of insulation [1]

The concept of the critical radius is mostly used for insulator analysis, but in our case, the ratio of k/h might be comparable with the critical radius. By obtaining the heat transfer coefficient data from ANSYS Fluent, we calculated the critical radius, which was 1.25 cm and is shown below.

$$r_{Cr,cylinder} = \frac{16.3 \left(\frac{W}{mK} \right)}{1300 \left(\frac{W}{m^2K} \right)} = 0.0125 \text{ m}$$

The calculated critical radius was very smaller than the outer radius of the tube (22 cm) and could not be used to explain our observed trend in the effect of wall thickness.

5.3.2. Effect of the inner diameter of the helix, D_{helix} :

The upward heat flow in the space confined by the helix wall can be affected directly by the $\frac{D_{Helix}}{D_{tube}}$ ratio; if the helix mean diameter (D_{helix}) is comparable with the velocity and thermal boundary layer thickness on the helix surface; we may observe some interference, and accordingly, a decrease in the heat transfer efficiency [3]. The higher the ratio of $\frac{D_{Helix}}{D_{tube}}$, the more effective heat transfer we have.

From Figure 20, It is clear that in our model, the effect of the inner diameter of the helix on heat transfers and temperature distribution is not comparable with the large D_{helix} . The blue part in the middle of the helical structure indicates that the temperature at this position is low, which means there is no trace of the spiral side to the other. As mentioned in the above paragraph, if this distance were less and affected the boundary layer and its thickness, it would also reduce the heat transfer. But in this case, with this substantial value of the inner diameter of the helix, we may overlook the effect of the inner diameter of the helix on the heat transfer rate.

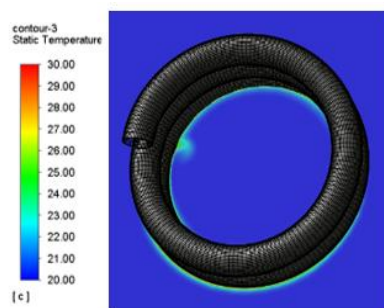


Figure 20. Temperature contour in the XZ-plane

5.4.Velocity distribution:

In natural convection phenomena, all three classes of fluid motion, namely laminar, transition, and turbulent, are expected. As a result, complicated velocity patterns can subsequently lead to interesting and unpredictable patterns. In Figure 21, the velocity distribution of our model is plotted. Figure 21.a illustrates a cut plot from the top view of the helix. In the beginning, we were expecting a symmetrical velocity distribution over each circular cut of the helix. But, as one can see, the velocity of water, trapped in the inner loop of the helix, is much higher than the fluid velocity around it. However, the fluid out of the helical loop is in contact with a large reservoir of water that can dissipate the absorbed heat and remain cool. As a result, we see a significantly asymmetric pattern in velocity. Further investigation showed that this phenomenon is also observed in a recent publication in 2020 by Brodnianská et al. [3], for helical structures, known as the chimney effect.

Moreover, from pathlines shown in Figure 21.b, we can see that the up going heated water continues to heat up while being in contact with the upper turns of the helix and increases its velocity exponentially. This assisting flow phenomenon is also known as the thermal plume, which is enhanced considerably due to the helical structure of the heated system.

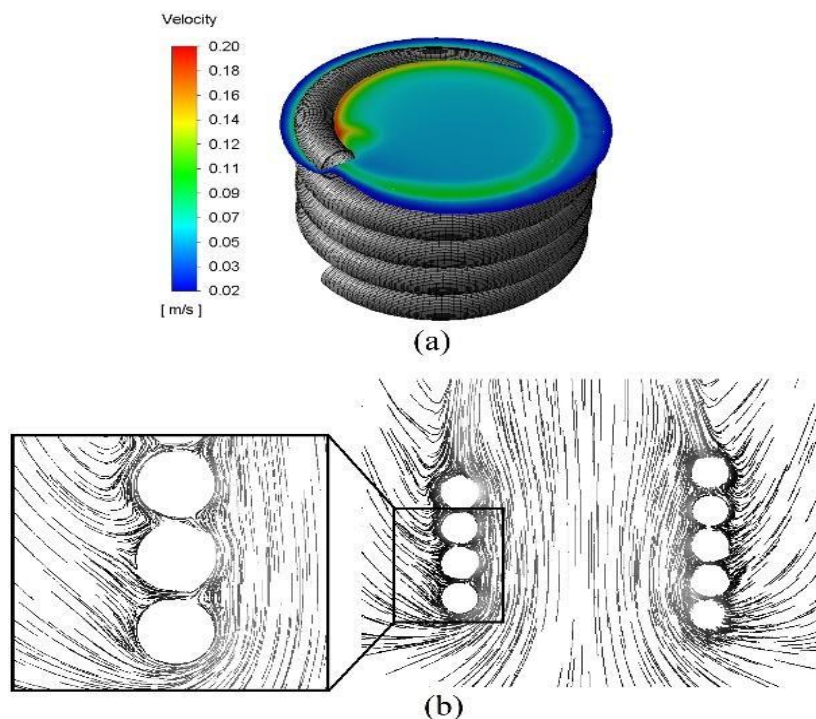


Figure 21. a) Velocity contour. b) Velocity pathline of the model

5.5. Effect of the thermal plume on heat transfer rate and surface Nusselt number:

As mentioned in the introduction section, based on the ratio of P/D , lower turns of the helix may affect the heat flux on the higher turns. Based on the dimension of our model given in Figure 4, the P/D ratio is equal to 1.13. Since this value is less than the critical value of 1.5 [10], the heat transfers of the upper turns decreases. We defined two lines, and then plotted the temperature distribution over these lines in Figure 22. As can be seen, the maximum temperature of the helical tube in the first and last turns, which is more in contact with colder fluid than the middle turns, are very close to each other, but the maximum temperature of the last turn is a little higher than the first one. This higher temperature indicates a decrease in heat transfer. On the other hand, the same result can be seen by comparing the tubes in the middle of the coil. Their maximum temperatures were also very close to each other, but with higher values of maximum temperature for upper turns.

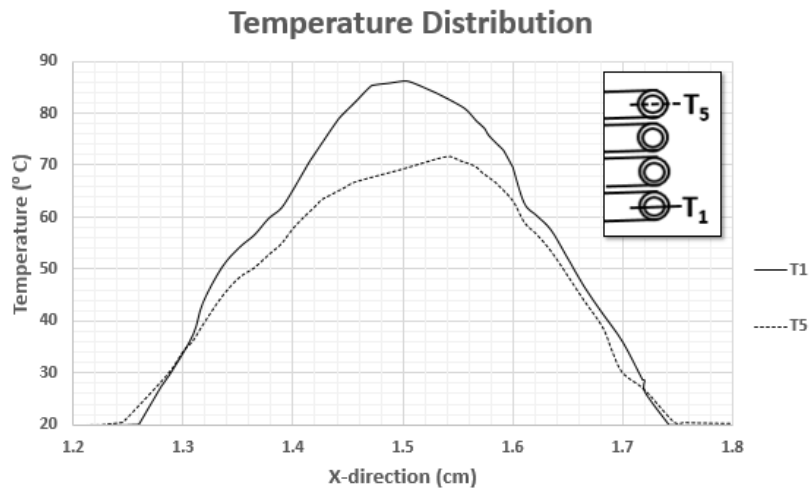


Figure 22. Temperature distribution of 2 different turns of the helix

To investigate how the plume effect is affecting the Nusselt number of higher turns, we modeled a number of helices, with a different number of turns, including 1, 2, 4, 6, and 7 turns. The average Nusselt number for each of these models were computed and shown in Table 8.

According to what we discussed earlier, an increase in the number of turns can entail a propagation of velocity on the upper turns, hence, increasing the Nusselt number. This trend can be seen in Table 8 as well.

Table 8. The Nusselt number in 3 helical tubes with different numbers of turns.

Number of turns	1	2	4	6	7
Nusselt number	1550.2	1595.0	1667.5	1768.8	1789.7

To visualize the velocity effects, and also, the increase in Nusselt numbers of the upper turns, Figure 23 and 24 are plotted. The direct impact of velocity on the Nusselt number is evident from Figure 23. Moreover, as shown in Figure 24 in the spacing between turns velocity is notably small due to the formation of eddy currents. In these areas, we have the lowest surface Nusselt number. Conversely, on the interior wall of the helix, the highest surface Nusselt number value in each turn is observed that is consistent with the high-velocity sections in the velocity contour of Figure 21.a.

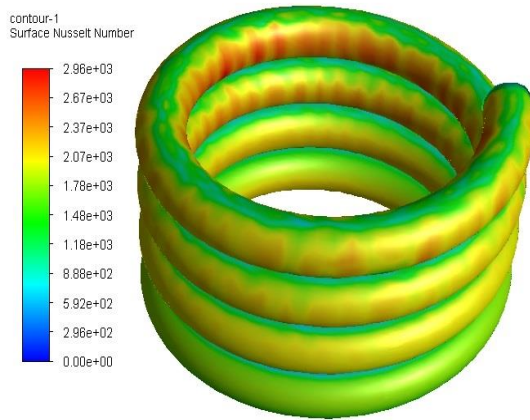


Figure 23. Surface Nusselt number contour for helical tube

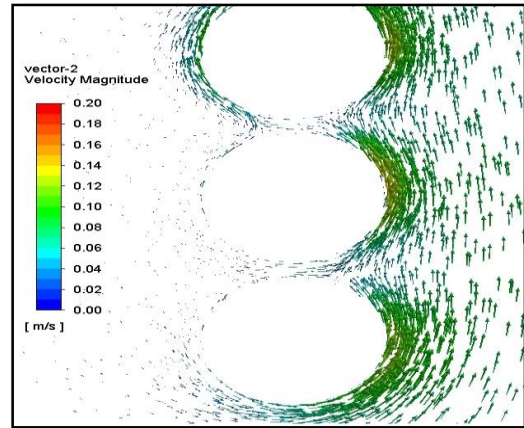


Figure 24. Velocity vector between two turns

For our vertical model, we conducted 18 variations of possible design parameters by changing the pitch (P), tube diameter (D), the total length of the helix (L), the total height of the helix (H), and number of turns (N). Finally, using nonlinear curve fitting and least square regression method (LSM), we are proposing a relation for estimating Nusselt number. The comparison between the estimated Nusselt number by our proposed equation and the CFD results are depicted in Figure 25.

$$Nu = (15.578 \ln(Ra_D Ra_L Ra_H) + 411.158) \times N^{0.0413} \quad (26)$$

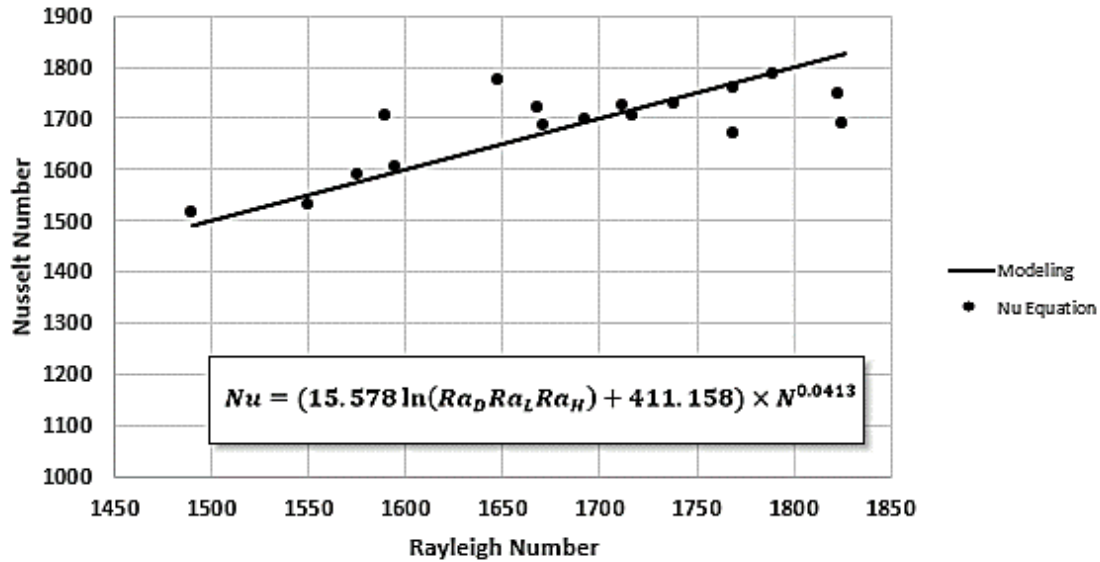


Figure 25. Results of the estimated Nusselt number equation

Where Ra_D is the Rayleigh number in which characteristic length is the tube diameter, Ra_L , and Ra_H are Rayleigh numbers calculated based on length, and height of the helical tube, respectively. Also, N represents the number of helical turns.

5.6. Horizontal helical tube:

In the introduction and methodology section, the effect of the angle of inclination of the helix was discussed, and the heat transfers in horizontal and vertical helical structures were compared. Here, we examine the results of the simulation of a horizontal helix. In Table 9, the heat transfer rate and surface Nusselt number of the vertical and horizontal helical tube in constant temperature boundary conditions are compared.

Table 9. The heat transfer rate and surface Nusselt number of the vertical and horizontal helical tube.

Orientation	$Q \left(\frac{W}{m^2} \right)$	Surface Nusselt number
Vertical	118911.8	1992.54
Horizontal	97392.9	1931.98
% Difference	18.1%	3.0%

As discussed earlier, the increase in Nusselt number for upper turns is strongly influenced by the increase in velocity by lower turns due to the thermal plume effect. In a horizontal helix, nevertheless, this effect is diminished. As opposed to the vertical configuration, in which the velocity decreased in the spacing between helical turns, in the horizontal orientation, the fluid flow increases in the spacing between the turns.

In constant temperature boundary condition, the temperature of fluid adjacent to the helical tube in both vertical and horizontal models have the same value. So there is no difference in thermal properties, such as thermal conductivity and Pr number, of the water in these two cases. Consequently, the difference in Nusselt number values refers to different velocity effects.

Based on the amount of heat flux, it can be concluded that the vertical configuration is more efficient for heat removal by almost 18%.

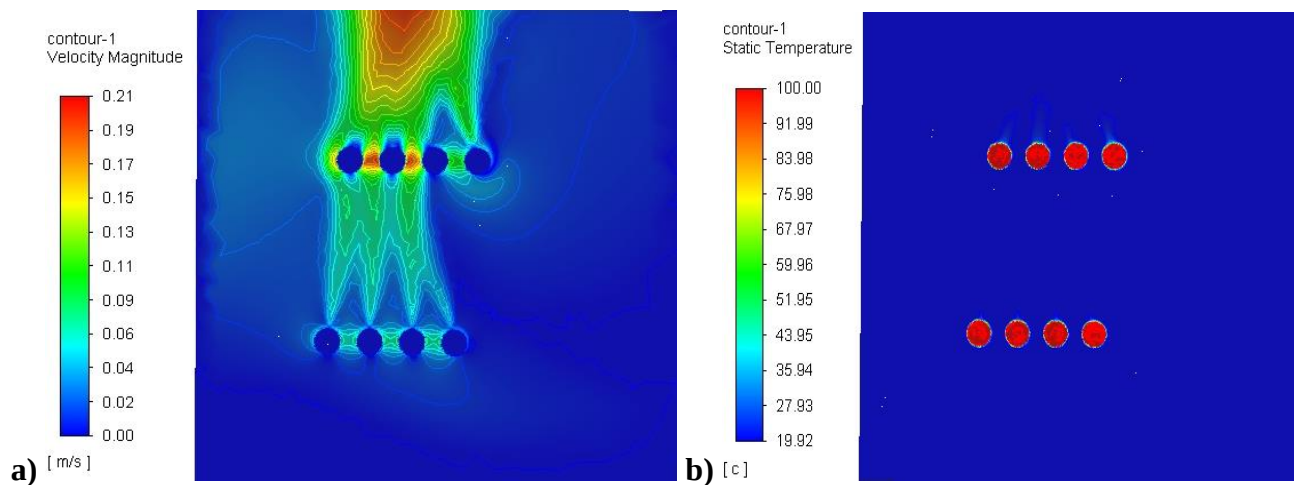


Figure 26. Contours of horizontal helical tube modeling. **a)** Velocity contour. **b)** Temperature contour.

6. Conclusion

In many industrial applications, there is a need to remove undesired excessive generated heat. Natural convection, although weaker than forced convection, can be advantageous due to its simplicity and lack of any fluid mover that eliminates any chance of malfunctioning. The main drawback of these passively cooled systems is their low heat transfer coefficient that requires extensive engineering investigations to optimize their performance to achieve maximum heat removal. Helical geometries are of particular interest in this field due to their compactness and high accessible surface area.

In this project, we employed a computational fluid dynamics (CFD) package, ANSYS Fluent, to model and to study the thermohydraulic behavior of a passively cooled helical system with various design parameters. A wide range of analysis tools, such as contour plots, velocity vectors, and pathline plots, were utilized to develop an understanding of the natural convection phenomenon. For instance, the observation of interesting effects, namely chimney effect and thermal plume, helped us to explain the mechanism of natural convection on a helical surface and make informed decisions about optimum design parameters.

Finally, using the least square regression method, we proposed a multivariable nonlinear equation to estimate the Nusselt number of our helical system, based on the obtained computational results.

7. Recommendations and Suggestions for Future Work

For future students who are interested in related topics to this project, we proposed two research objective:

As mentioned in the introduction, the FBNR reactor has a conceptual design idea to achieve inherent safety by passively cooling down the fuel chamber by immersing it in a water bath. In such highly exothermic systems, the boiling heat transfer will dominate the natural convection, and considerably higher amounts of heat can be dissipated. To properly model this phenomenon, advanced state-of-the-art phase change models have to be implemented in ANSYS Fluent. In the future, one can employ our discretized geometry and only focus on modeling of the boiling event.

One can also expand this simulation and solve the time-dependent model in which the water bath has a fixed amount (no inlet/outlet boundary condition), and the generated heat in the fuel chamber drops over time by the exponential decay relation. This study can determine the required water bath volume to ensure the inherent safety of FBNR.

8. Reference:

- [1] Y. A. Cengel and A. J. Ghajar, *Heat and mass transfer : fundamentals & applications*. 2015.
- [2] A. Ghaffari, T. Javed, I. Mustafa, and F. Labropulu, "Modeling and simulation of natural convection flow along a rough surface of sinusoidal nature with variable heat flux: Using keller box scheme," *Therm. Sci.*, vol. 2018, no. 6, pp. 3391–3400, 2018, doi: 10.2298/TSCI170929106G.
- [3] Z. Brodnianská and S. Kotšmíd, "Numerical Study of Heated Tube Arrays in the Laminar Free Convection Heat Transfer," 2020, doi: 10.3390/en13040973.
- [4] S. Şahin and F. Sefidvash, "The fixed bed nuclear reactor concept," *Energy Convers. Manag.*, vol. 49, no. 7, pp. 1902–1909, 2008, doi: 10.1016/j.enconman.2007.12.017.
- [5] F. Sefidvash, "Sustainability and the Fixed Bed Nuclear Reactor (FBNR)," *Sustainability*, vol. 4, no. 8, pp. 1683–1710, 2012, doi: 10.3390/su4081683.
- [6] F. Sefidvash, *Water Cooled FBNR Nuclear Reactor*. International Atomic Energy Agency (IAEA): IAEA, 2011.
- [7] J. Y. Moon, J. H. Heo, and B. J. Chung, "Natural convection experiments on the outer surface of an inclined helical coil," *Heat Mass Transf. und Stoffuebertragung*, vol. 51, no. 9, pp. 1229–1236, 2015, doi: 10.1007/s00231-015-1495-5.
- [8] S. Pang, S. Zhang, M. Wang, and B. Liu, "CFD-Based Numerical Simulation for Helical Heating Pipes," *Comput. science Eng.*, no. Ictia, pp. 17–23, 2018.
- [9] M. S. Mitchell, G. M. Hansen, J. D. Spitler, M. S. Mitchell, G. M. Hansen, and J. D. S. Experimental, "Experimental development of natural convection heat transfer correlations for spiral-helical surface water heat exchangers (1385-RP) Experimental development of natural convection heat transfer correlations for spiral-helical surface water heat," *Sci. Technol. Built Environ.*, vol. 0, no. 0, pp. 1–12, 2018, doi: 10.1080/23744731.2017.1365764.
- [10] J. Heo and B. Chung, "International Journal of Heat and Mass Transfer Influence of helical tube dimensions on open channel natural convection heat transfer," *Int. J. Heat Mass Transf.*, vol. 55, no. 11–12, pp. 2829–2834, 2012, doi: 10.1016/j.ijheatmasstransfer.2012.02.043.
- [11] E. Neshat and S. Hossainpour, "Numerical investigation of unsteady natural convection heat transfer from the outer surface of helical coils," *Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng.*, vol. 231, no. 3, pp. 383–391, 2017, doi: 10.1177/0954408915598130.
- [12] D. G. Prabhanjan, T. J. Rennie, and G. S. V. Raghavan, "Natural convection heat transfer from helical coiled tubes," *Int. J. Therm. Sci.*, vol. 43, no. 4, pp. 359–365, 2004, doi: 10.1016/j.ijthermalsci.2003.08.005.
- [13] M. E. Ali, "Experimental investigation of natural convection from vertical helical coiled tubes," *Int. J. Heat Mass Transf.*, vol. 37, no. 4, pp. 665–671, 1994, doi: 10.1016/0017-9310(94)90138-4.
- [14] R. C. Xin and M. A. Ebadian, "Natural convection heat transfer from helicoidal pipes," *J. Thermophys. Heat Transf.*, vol. 10, no. 2, pp. 297–302, Apr. 1996, doi: 10.2514/3.787.
- [15] M. E. Ali, "Laminar natural convection from constant heat flux helical coiled tubes," *Int. J. Heat Mass Transf.*, vol. 41, no. 14, pp. 2175–2182, 1998, doi: 10.1016/S0017-9310(97)00322-0.
- [16] E. Izadpanah, A. Zarei, S. Akhavan, and M. Babaie Rabiee, "An experimental investigation of natural convection heat transfer from a helically coiled heat exchanger," *Int. J. Refrig.*, vol. 93, pp. 38–46, 2018, doi: 10.1016/j.ijrefrig.2018.06.008.
- [17] M. Ali, "Natural Convection Heat Transfer from Vertical Helical Coils in Oil," *Heat Transf. Eng.*, vol. 27, pp.

- 79–85, Jan. 2006, doi: 10.1080/01457630500458617.
- [18] E. M. Languri, H. B. Rokni, J. Alvarado, B. Takabi, and M. Kong, “Heat transfer analysis of microencapsulated phase change material slurry flow in heated helical coils: A numerical and analytical study,” *Int. J. Heat Mass Transf.*, vol. 118, no. December 2017, pp. 872–878, 2018, doi: 10.1016/j.ijheatmasstransfer.2017.10.130.
- [19] A. Alimoradi, M. Olfati, and M. Maghareh, “Numerical investigation of heat transfer intensification in shell and helically coiled finned tube heat exchangers and design optimization,” *Chem. Eng. Process. Process Intensif.*, vol. 121, Aug. 2017, doi: 10.1016/j.cep.2017.08.005.
- [20] F. Bagherzadeh, M. Saffar-Avval, M. Seyfi, and A. Abbassi, “Numerical investigation of nanofluid heat transfer in helically coiled tubes using the four-equation model,” *Adv. Powder Technol.*, vol. 28, Nov. 2016, doi: 10.1016/j.appt.2016.09.011.
- [21] D. Maity, P. Siddheshwar, and S. Saha, *Advances in Fluid Mechanics and Solid Mechanics Proceedings of the 63rd Congress of ISTAM 2018: Proceedings of the 63rd Congress of ISTAM 2018*. 2020.
- [22] “ANSYS Fluent Theory Guide,” vol. 15317, no. November, pp. 724–746, 2013.
- [23] L. Davidson, “Fluid mechanics , turbulent flow and turbulence modeling,” p. 520.
- [24] I. Most, G. Stream, N. Committee, F. D. Films, and I. One, “The Flow of Natural Gas,” *Sci. Am.*, vol. 4, no. 103supp, pp. 1644–1644, 1877, doi: 10.1038/scientificamerican12221877-1644esupp.
- [25] C. Lenin, Edwin and Arrieta, “Computational Fluid Dynamic Simulation of Vertical Axis Hydrokinetic Turbines,” *Intech*, vol. i, no. tourism, p. 13, 2012, doi: 10.1016/j.colsurfa.2011.12.014.
- [26] Z. Zhai, *Computational Fluid Dynamics for Built and Natural Environments*, 1st ed. 20. Springer, Singapore, 2020.
- [27] M. Saniei nejad, *Introduction to the concepts of turbulent flows and their modeling*, Third eddi. Tehran: Danesh negar, 2006.
- [28] J. D. Murphy, *Turbulence modeling*. Washington, D.C: Ames Research Center, 1984.
- [29] J. Sodja, “Turbulence models in CFD,” *Univ. Ljubljana*, no. March, pp. 1–18, 2007.
- [30] T. D. Canonsburg, “ANSYS Fluent User ’ s Guide,” vol. 15317, no. November, pp. 724–746, 2013.
- [31] M. Fadl, L. He, P. Stein, and G. Marinescu, “Assessment of Unsteadiness Modeling for Transient Natural Convection,” *J. Eng. Gas Turbines Power*, vol. 140, no. 1, 2018, doi: 10.1115/1.4037721.
- [32] F. Sefidvash, “Version 14/07/2009,” in *FBNR Workshop*, 2009, p. 335.
- [33] S. Fatemi, *Mathematical modeling in chemical engineerings*, 5th editio. University of Tehran Press, 2013.
- [34] O. Marzouk, A. H. Nayfeh, I. Akhtar, and H. N. Arafat, “Modeling steady-state and transient forces on a cylinder,” *JVC/Journal Vib. Control*, vol. 13, no. 7, pp. 1065–1091, 2007, doi: 10.1177/1077546307078737.
- [35] H. Mistry, Ganapathi-subbu, S. Dey, P. Bishnoi, and J. L. Castillo, “Modeling of transient natural convection heat transfer in electric ovens,” *Appl. Therm. Eng.*, vol. 26, no. 17–18, pp. 2448–2456, 2006, doi: 10.1016/j.applthermaleng.2006.02.007.
- [36] “Introduction to thermal and electrical conductivity.” [Online]. Available: https://www.doitpoms.ac.uk/tlplib/thermal_electrical/printall.php%0A.