KSÜ Mühendislik Bilimleri Dergisi, 28(3), 2025 KSU J Eng Sci, 28(3), 2025
Araştırma Makalesi Research Article

Kahramanmaras Sutcu Imam University
Journal of Engineering Sciences

Geliş Tarihi : 26.03.2025 Received Date : 26.03.2025
Kabul Tarihi :14.04.2025 Accepted Date : 14.04.2025

MULTI-VARIABLE ANALYSIS OF THERMAL AND FLOW CHARACTERISTICS IN CYLINDRICAL CHANNELS USING RESPONSE SURFACE METHODOLOGY

YANIT YÜZEY METODOLOJİSİ KULLANILARAK SİLİNDİRİK KANALLARDA ISIL VE AKIŞ ÖZELLİKLERİNİN ÇOK DEĞİŞKENLİ ANALİZİ

Fuat TAN [1*] (ORCID: 0000-0002-4194-5591)
Oğuz Veli SATI [1] (ORCID: 0009-0002-6656-1823)

[1]Balıkesir University, Department of Mechanical Engineering, Balıkesir, Türkiye

*Sorumlu Yazar / Corresponding Author: Fuat TAN, fuattan@balikesir.edu.tr

ABSTRACT

The present study deals with the flow dynamics and heat transfer parameters under laminar flow conditions in the cylindrical tube geometry. The fluid velocity distribution, temperature profile, and exit parameters were analyzed considering the variables of tube length (80–120 mm), entry radius (12–20 mm), and entry temperature (333–413 K). With the help of ANSYS Fluent, Response Surface Method (RSM) and Box-Behnken experimental design (BBD), 46 different scenarios have been analyzed. The results indicate that narrow inlet diameters not only affected the boundary layer formation by increasing the velocity gradients but also linearly increased the exit temperature as the temperature increases. Moreover, as the pipe length increases, it is found that the flow development region elongates and the temperature differences fall. Flow regimes were evaluated through Reynolds and Nusselt numbers, and model accuracy was proven with over 95% regressive coefficient. This study contributes to the determination of the optimal flow and heat transfer conditions in cylindrical pipes. It enriches the literature with good quantitative data about the relationship between geometric and thermal variables and flow behaviors, and provides useful tips for heating units and industrial processes that function with laminar flow delivery.

Keywords: CFD, experimental design, pressure, heat transfer, reynolds

ÖZET

Bu çalışma, silindirik boru geometrisinde laminer akış koşulları altında akış dinamiği ve ısı transfer parametrelerini incelemektedir. Boru uzunluğu (80–120 mm), giriş yarıçapı (12–20 mm) ve giriş sıcaklığı (333–413 K) değişkenleri dikkate alınarak, akışkan hız dağılımı, sıcaklık profili ve çıkış parametreleri analiz edilmiştir. ANSYS Fluent, Yanıt Yüzey Yöntemi (RSM) ve Box-Behnken deney tasarımı (BBD) kullanılarak 46 senaryo değerlendirilmiştir. Sonuçlar, dar giriş çaplarının hız gradyanlarını artırarak sınır tabaka oluşumunu etkilediğini ve sıcaklık arttıkça çıkış sıcaklığının doğrusal olarak yükseldiğini göstermektedir. Ayrıca, boru uzunluğunun artmasıyla akış gelişim bölgesi uzamakta ve sıcaklık farkları azalmaktadır. Çalışmada akış rejimleri Reynolds ve Nusselt sayıları ile değerlendirilmiş, model doğruluğu %95'in üzerinde regresyon katsayısı ile kanıtlanmıştır. Yapılan bu çalışma ile silindirik borularda optimal akış ve ısı transfer koşullarının belirlenmesine katkı sağlanmaktadır. Geometrik ve termal değişkenler ile akış davranışları arasındaki ilişkiye dair sağlam nicel verilerle literatürü zenginleştirmekte ve laminer akışla çalışan ısıtma üniteleri ile endüstriyel süreçler için faydalı ipuçları sağlamaktadır.

Anahtar Kelimeler: CFD, deneysel tasarım, basınç, ısı transferi, reynolds sayısı

ToCite: TAN, F., & SATI, O. V., (2025). NUMERICAL INVESTIGATION OF THERMAL AND STRUCTURAL BEHAVIOR IN INJECTION-MOLDED FEMUR IMPLANTS. *Kahramanmaras Sutcu Imam University Journal of Engineering Sciences,* 28(3), 1309-1319.

INTRODUCTION

Cylindrical pipes in fluid mechanics and heat transfer applications are widely employed in industrial and engineering processes. In these systems, the fluid's velocity distribution and temperature variation change significantly depending on the pipe geometry and inlet conditions. Parametric analyses are good ways of checking the impacts of this study in order to find out the best possible design criteria.

D. Sumner studied how transverse and longitudinal spacing between cylinders, along with Reynolds number, affect flow patterns, aerodynamic forces, vortex formation, and other related parameters (Sumner,2010). Commodore Nagesh et al. analyzed the stress distribution across the thickness of different cylindrical models in ANSYS. They tested the accuracy of the numerical model for a composite tube under internal pressure. The proposed boundary conditions were validated, showing that a single circumferential element was sufficient. This approach significantly improves computational efficiency in structural analysis. The study highlights the importance of effective boundary conditions in enhancing the accuracy and efficiency of parametric analyses (Nagesh et al.,2023). Li et al. studied the effects of different loading conditions on maximum stresses in a cylinder using the finite element method (Li et al.,2012).

Panagiotis E. Chatzistergos et al. compared experimental and numerical results to demonstrate the importance of parametric studies. Their findings confirmed that this approach significantly improves accuracy (Chatzistergos et al.,2010). Halime Çelik and Nezaket Parlak investigated single-phase laminar water flow and heat transfer in a microchannel with a rectangular cross-section. The channel contained six fins, and parameters such as fin length, width, and angle were analyzed. Computational Fluid Dynamics (CFD) software was used to determine the optimal geometry. The analysis was conducted for Reynolds numbers between 70-210, and the results were compared. This study highlights the importance of geometric optimization for single-phase flow and heat transfer in microchannels (Parlak and Celik,2018).

Jing, D et al. analyzed the effects of FVG on mixing efficiency and pressure loss at different Reynolds numbers (Jing et al.,2023). Bisagni, C. et al. conducted a fluid mechanics analysis to study the behavior of a cylindrical pipe system under dynamic loading. They used parametric analysis to examine how temperature differences and air velocity inside the pipe affect system efficiency and pressure losses (Bisagni et al.,2020). Bhowmick, S., et al. investigated the effects of hot and cold air inlet positions and geometric parameters on the dynamic behavior of the internal fluid and pressure losses in a cylindrical pipe using parametric analysis (Bhowmick et al.,2018). Qin, B et al. analyzed the effects of different flow conditions, pipe geometry, and hot-cold air inlets. They used parametric analysis to study thermal and fluid dynamic behavior (Qin et al.,2020).

Kadry, A. A., et al. developed a new mathematical model to accurately predict the capacity of multi-planar pipe connections. The model was validated using data from ANSYS-based parametric analyses. It provides separate equations for local and global beam failures and shows 92.5% accuracy when compared to experimental results. This demonstrates its potential as a reliable and efficient tool for industrial applications (Kadry et al.,2022). Celik, E. et al. performed numerical simulations using Fluent CFD software and examined performance under various boundary conditions in detail (Celik et al.,2020). Mangrulkar, C. K., et al. conducted parametric analyses to optimize heat transfer by studying dynamic processes such as vortex formation and flow separation during the flow of hot fluids over cylindrical surfaces (Mangrulkar et al.,2019).

Li, M. Z., et al. and Kishor S. Rambhad et al. used a parametric analysis approach to study the effects of different pipe diameters, inlet conditions, and flow velocities. Their research provides a strong foundation for understanding the dynamic behavior of pipe systems with varying temperature and velocity inlets (Li et al.,2020- Rambhad et al.,2021). Tan, X., et al. analyzed the effects of a multi-inlet design on thermal performance and flow characteristics in a cross-flow system. Numerical results showed that optimizing flow velocity and temperature distribution can enhance heat transfer while minimizing pressure drop (Tan et al.,2021).

Erdinç, M. T., et al. used various software tools to solve heat transfer and fluid mechanics problems, with ANSYS Fluent being one of the most reliable. However, it could not directly calculate dimensionless parameters like local and average Nusselt numbers, requiring additional post-processing, which was time-consuming when parameters changed. ANSYS Workbench allowed the calculation of Nusselt number, entropy generation, and the angles between velocity vectors and temperature gradients. This study analyzed heat transfer using an example model with irregularly arranged plates, and the results were presented (Erdinç et al.,2015).

Jing, D. et al. conducted a numerical study on the effects of a flexible vortex generator (FVG) on fluid mixing at different Reynolds numbers. They analyzed how the horizontal and branching lengths of a Y-shaped elastic filament influence mixing efficiency and pressure loss. Results showed that at Re=50, a filament with large horizontal and branching lengths significantly improved mixing. However, at Re=10 and 90, it was ineffective and even reduced mixing while increasing pressure loss. These findings highlight the importance of optimizing branching length to maximize fluid mixing efficiency (Jing et al.,2023).

Doba, F. and Oğulata, R. T., tested a cross-flow plate heat exchanger in a laboratory to evaluate its efficiency. Experiments measured temperature, air velocity, and pressure losses, while entropy generation was analyzed based on the second law. The study examined how entropy generation changes with parameters such as flow path length, mass flow rate, heat transfer area, and volume. Results were presented with graphs. This research contributes to designing more efficient heat exchangers, reducing energy losses, and improving thermal system performance (Doba and Ogulata,2000). Zena K. Kadhim et al. compared the use of plain and finned tubes in cross-flow heat exchangers through CFD analysis. Experiments measured temperature differences and heat transfer coefficients at specific air velocities and water flow rates. Results showed that finned tubes provided higher heat transfer performance than plain tubes. This study highlights the advantages of finned tubes in improving heat exchanger efficiency and energy savings (Kadhim et al.,2016).

Previous studies have looked into different aspects of the fluid's movement and thermal exchange in cylindrical structures. However, the majority of them focused on just one variable, like the velocity of the fluid at the inlet or the diameter of the pipe, or used experimental devices with limited variable control. Chatzistergos, P. E. et al. (2010) and Qin et al. (2020) focused on structural performance or dynamic behavior under specific loadings, and then only Kadry, A. A., et al. (2022) and Jing, D. et al. (2023) were mainly engaged in the output of particles by geometric optimization or mixing efficiency. Nevertheless, the literature lacks a systematic numerical investigation that applies a statistically validated design of experiments when considering the concurrent influences of geometric and thermal factors such as inlet diameter, pipe length, and temperature. Unlike existing research, it provides a statistically validated model for predicting flow and heat transfer responses under laminar conditions and delivers quantitative insights into how key input variables interact to affect performance metrics such as outlet velocity, temperature, Reynolds, and Nusselt numbers.

In this study, the flow characteristics and heat transfer processes under various pipe lengths, inlet diameters, and inlet temperatures have been analyzed. The solutions of continuity, momentum, and energy equations were obtained using Computational Fluid Dynamics (CFD) method, and the parametric design was carried out using Response Surface Methodology (RSM). The objective of the study is to set the speed distribution and temperature profiles for the cylindrical tube systems, identifying fluid dynamics to be optimized.

MATERIALS AND METHODS

Numerical Model

Computational Fluid Dynamics (CFD) is a useful method for analyzing the effects of selected parameters on a physical model in a virtual environment. By defining parameters as variables, experimental design can be applied through simulations. In this study, the selected parameters are the main inlet pipe diameter, cold fluid inlet velocity and temperature, and total pipe length. The physical model includes six opposing circular inlets, each with a 4 mm diameter. The main inlet pipe has a diameter range of 12–20 mm and a total length of 80–120 mm. The experimental design was conducted using the Box-Behnken method.

The commercial ANSYS Fluent 2022 R2 software was used to solve the physical model. To model the flow behavior, the continuity, momentum, and energy equations were applied for a two-dimensional, laminar, incompressible, and developed flow. The continuity equation for two-dimensional incompressible flow is:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

It is used as follows:
x = Dimension in the flow direction
y = Dimension perpendicular to the flow direction
u = Velocity component in the x-direction
v = Velocity component in the y-direction

Similarly, for a two-dimensional, steady, incompressible laminar flow, the momentum equations for the x and y directions are given below:

The x-direction momentum equation is as follows.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial P}{\partial x}+v\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

The y-direction momentum equation is as follows.

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial P}{\partial y}+v\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

It is used as follows. Here,

P = Pressure (Pa)
ρ = Fluid density (A/mm^2)
v = Kinematic viscosity ($m^2 \cdot s^{-1}$)

The energy equation for two-dimensional, fully developed, incompressible, laminar flow is:

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=a\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

$a=\frac{k}{\rho}\cdot c_p$ Thermal diffusivity coefficient,
k = Thermal conductivity coefficient, (W / m·K)
c_p = Specific heat capacity at constant pressure, (kJ / kg·K)

In this study, the given equations were solved using ANSYS Fluent, considering the necessary boundary conditions. The velocity and temperature distributions in the channel were determined.

Geometry

The physical model includes six cold fluid channels connected to an axial main inlet. The dimensions and 3D view of the model are shown in Figures 1 and 2. Aluminum was chosen as the material for the pipe system.

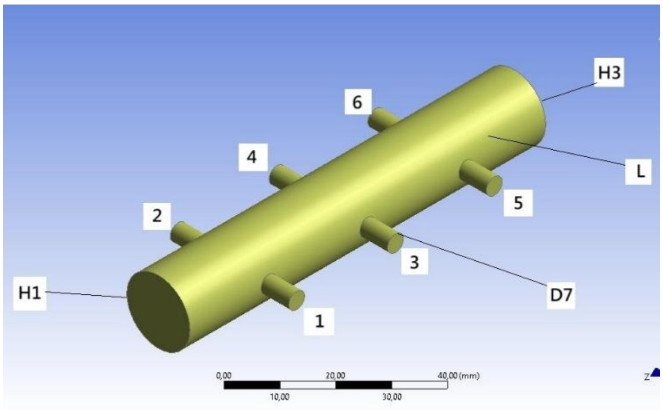


Figure 1. 3D Model

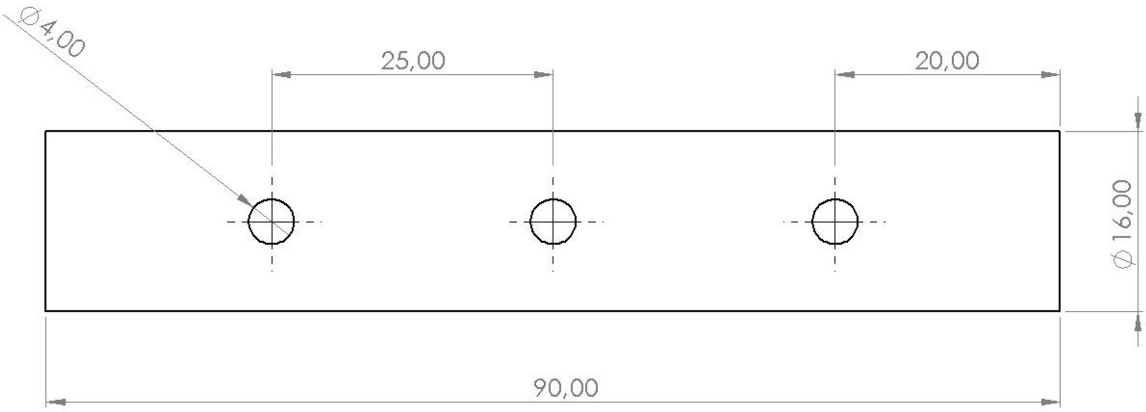


Figure 2. Model Dimensions

The table below presents the parameter limits and variables of the physical model. The fixed values of H3 and D7, along with the cold fluid inlet velocity and temperature, are also specified in the table. The specified boundary values for the primary geometric and thermal parameters employed in the simulations are displayed in Table 1. The stepwise change of the pipe length, inlet radius, inlet velocity, and inlet temperature was carried out in such a way that the resultant flow and thermal behavior of the system could be studied.

Table 1. Parametric Boundary Values

Parameters	Boundaries	Unit
L	80-120	mm
H1	12-20	mm
H1 Velocity	0.002-0.004	m/s
H1 Temperature	333-413	K

Table 2 contains fixed parameters over all the simulation scenarios, including cold inlet channels dimensions, velocity, and temperature ratios. The determination of these parameters guarantees a steady and reproducible boundary condition for every simulation scene.

Table 2. Parameter Table

Parameter	Value	Unit
H3	16	mm
D7	4	mm
cold_inlet_1_Velocity	0.005	m/s
cold_inlet_2_Velocity	0.005	m/s
cold_inlet_3_Velocity	0.006	m/s
cold_inlet_4_Velocity	0.006	m/s
cold_inlet_5_Velocity	0.007	m/s
cold_inlet_6_Velocity	0.007	m/s
cold_inlet_1_Temperature	313	K
cold_inlet_2_Temperature	313	K
cold_inlet_3_Temperature	303	K
cold_inlet_4_Temperature	303	K
cold_inlet_5_Temperature	293	K
cold_inlet_6_Temperature	293	K

Mesh

Although the flow inside the volume is laminar, mesh quality is crucial and directly affects the results. Therefore, improvements were made to enhance mesh quality. The mesh structure of the model is shown in Figure 3. An element size of 0.0015 mm was used. The final model consists of 13,193 mesh elements and 63,142 nodes.

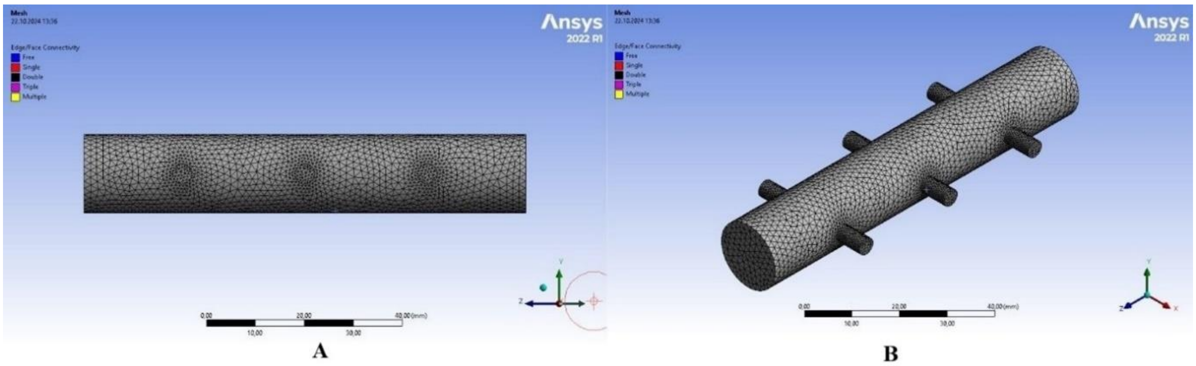


Figure 3. Mesh View (**A.** Sıde Vıew, **B.** Isometric View)

Boundary Conditions and Settings

In the analysis, the fluid entered the system with a temperature of 373 K and a velocity of 0.003 m/s. The fluid used was pure water, and its material properties were kept constant. The thermophysical properties at T = 298 K are given below. Here, μ is dynamic viscosity, ρ is density, Cp is specific heat capacity, and k is thermal conductivity.

Table 3. Water Physical Properties

μ (Pa.s)	8,8325x10^{-4}
k (W/mK)	0,601
C_p (j/kgK)	4175,78
P (kh/m^3)	998,2

The flow regime in the control volume is laminar, and the results indicate steady-state flow. Water was used as the working fluid, and the contact surface was frictionless aluminum. The boundary conditions of the physical model are shown in Figure 4. The model includes six cold fluid channels connected to the main flow (hot_fluid_inlet). The main flow temperature ranges from 333 K to 413 K, while the velocity varies between 0.002 m/s and 0.004 m/s.

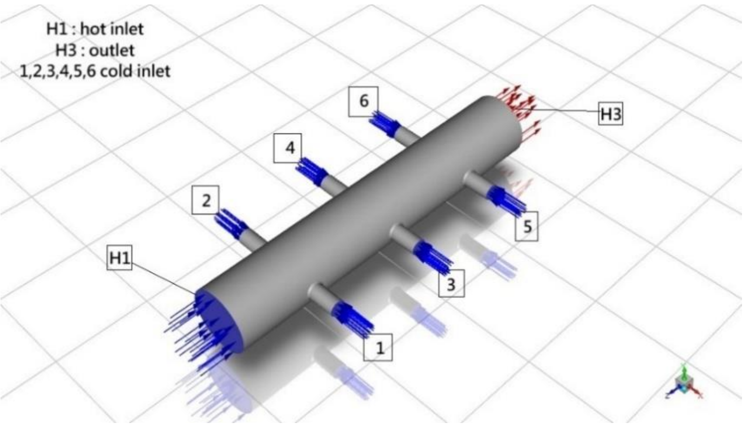


Figure 4. Physical Model

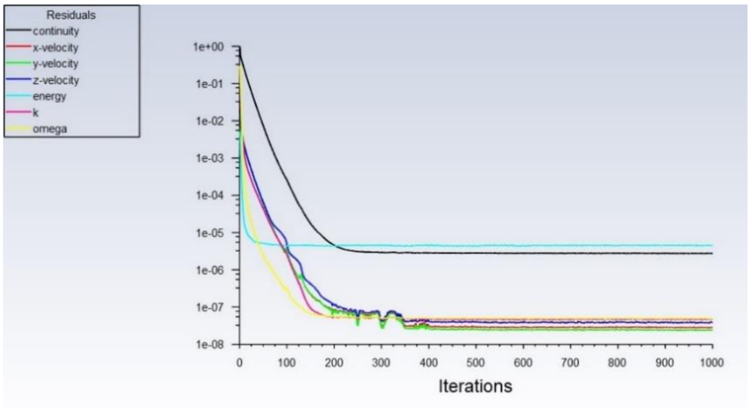


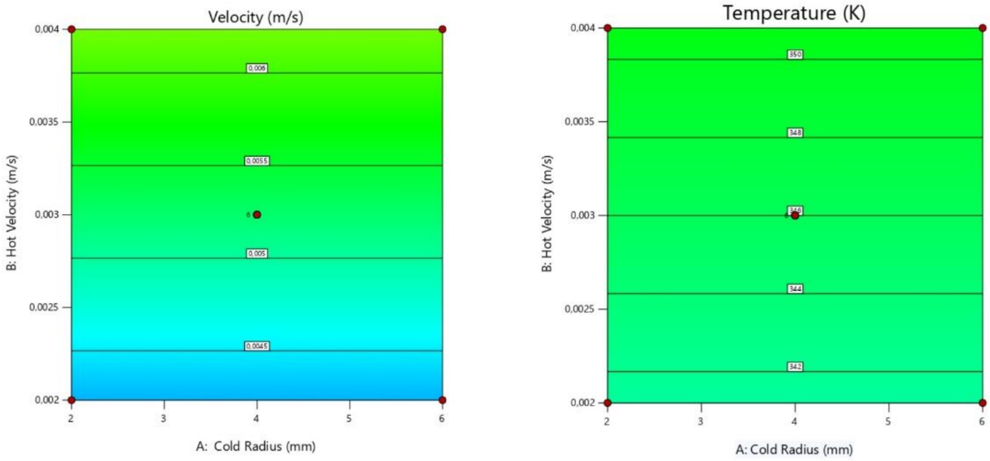
Figure 5. Residual Graph

As seen in Figure 5, 1000 iterations were performed. The solution converged after 400 iterations, with an error rate lower than 10^{-5}.

Experimental Design

Parametric analysis evaluates data based on related parameters and calculates results accordingly, reflecting them in a parameter table. To establish the mathematical relationship between parameters, an experimental design approach was used. A total of 46 analyses were conducted using the RSM Box-Behnken method. The performance of the Box–Behnken Design (BBD) plan used in this study contributed to the achievement of the objectives since it successfully reduced the number of simulations while maintaining the same accuracy and reliability of the model were maintained. The BBD does not include extreme combinations of parameter levels, which makes it a fast and safe option in both virtual and physical experiments. This situation is more typical of the one occurring in this work, where it is the quadratic effects of several variables in the range of practicality that are the main concerns. Tunçel et al. (2024) displayed the same advantages in their research as they were able to use BBD to increase the strength of PETG samples by reducing the number of trials while still getting significant statistical results in the end.

The ANOVA table generated by the method showed a regression coefficient above 0.95, indicating high reliability. Although no specific experiments were conducted to validate this study, the accuracy of the simulation results was supported by numerical verification. Additionally, the residuals were brought to a low point of less than 10^{-5} and the coefficient of determination ($R^2 > 0.95$) derived from the response surface model is a reflection of the statistical reliability of the predictions. The ANOVA table is shown in Table 4. The response surface model images for velocity and temperature results are presented in Figure 6. The parameter selected for the present project, the pipe length (80–120 mm), inlet radius (12–20 mm), and inlet temperature (333–413 K), was chosen based on the data and informed practices available in the earlier literature. For example, Parlak and Çelik (2018) have used similar inlet specifications in their microchannel heat transfer studies, and also, Sumner (2010) and Jing, D. et al. (2023) looked into comparable conditions for thermal loads in cylindrical geometries. These periods would solicit both low and moderate high thermal gradients, thus the dependable assessment of flow development and heat transfer behavior could be carried out by using the nearby applications.



a **b**

Figure 6. **a)** Velocity and **b)** Temperature Response Surface Model Results

Table 4. Anova Table

Std. Dev.	7,316E	R^2	1,000
Mean	0,0054	**Adjusted** R^2	1,000
C.V. %	0,0135	**Predicted** R^2	1,000
		Adec PrecisionR^2	9228,3535

Input parameters;

D7 cold radius (mm) = Inlet diameter of small pipes

hot_velocity (m/s) = Fluid inlet velocity

hot_temperature (K) = Fluid inlet temperature

H1 inlet radius (mm) = Inlet channel radius
V2 pipe length (mm) = Geometry length

Output parameters;
outlet_velocity (m/s) = Fluid outlet velocity
outlet_temperature (K) = Fluid outlet temperature

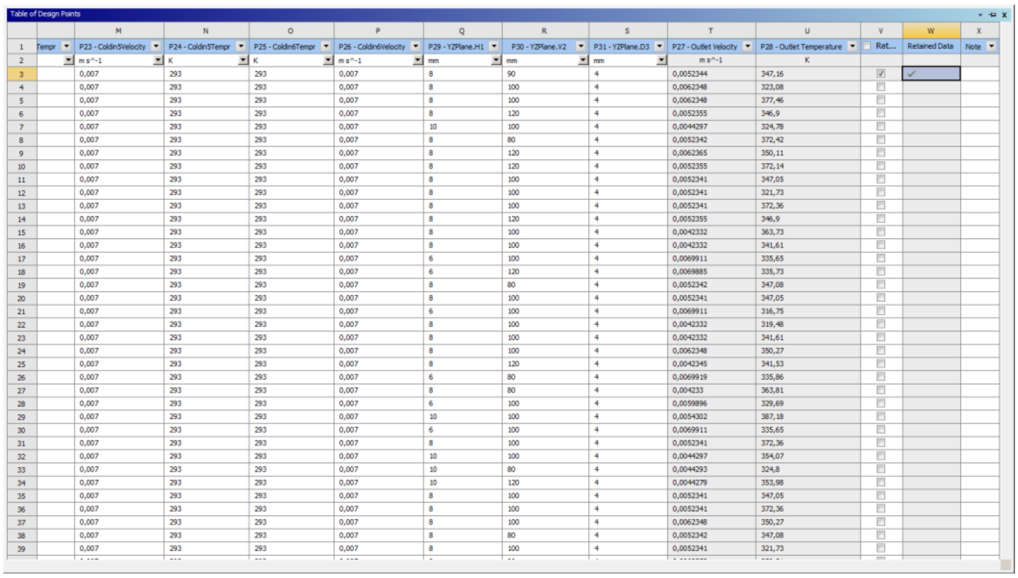


Table of Design Points

		M	N	O	P	Q	R	S	T	U	V	W	X
1	Tempr	P23 - ColdIn5Velocity	P24 - ColdIn5Tempr	P25 - ColdIn6Tempr	P26 - ColdIn6Velocity	P29 - YZPlane.H1	P30 - YZPlane.V2	P31 - YZPlane.D3	P27 - Outlet Velocity	P28 - Outlet Temperature	☐ Ret...	Retained Data	Note
2		m s^-1	K	K	m s^-1	mm	mm	mm	m s^-1	K			
3		0,007	293	293	0,007	8	90	4	0,0052344	347,16	☑	✓	
4		0,007	293	293	0,007	8	100	4	0,0062348	323,08	☐		
5		0,007	293	293	0,007	8	100	4	0,0062348	377,46	☐		
6		0,007	293	293	0,007	8	120	4	0,0052355	346,9	☐		
7		0,007	293	293	0,007	10	100	4	0,0044297	324,78	☐		
8		0,007	293	293	0,007	8	80	4	0,0052342	372,42	☐		
9		0,007	293	293	0,007	8	120	4	0,0062365	350,11	☐		
10		0,007	293	293	0,007	8	120	4	0,0052355	372,14	☐		
11		0,007	293	293	0,007	8	100	4	0,0052341	347,05	☐		
12		0,007	293	293	0,007	8	100	4	0,0052341	321,73	☐		
13		0,007	293	293	0,007	8	100	4	0,0052341	372,36	☐		
14		0,007	293	293	0,007	8	120	4	0,0052355	346,9	☐		
15		0,007	293	293	0,007	8	100	4	0,0042332	363,73	☐		
16		0,007	293	293	0,007	8	100	4	0,0042332	341,61	☐		
17		0,007	293	293	0,007	6	100	4	0,0069911	335,65	☐		
18		0,007	293	293	0,007	6	120	4	0,0069885	335,73	☐		
19		0,007	293	293	0,007	8	80	4	0,0052342	347,08	☐		
20		0,007	293	293	0,007	8	100	4	0,0052341	347,05	☐		
21		0,007	293	293	0,007	6	100	4	0,0069911	316,75	☐		
22		0,007	293	293	0,007	8	100	4	0,0042332	319,48	☐		
23		0,007	293	293	0,007	8	100	4	0,0042332	341,61	☐		
24		0,007	293	293	0,007	8	100	4	0,0062348	350,27	☐		
25		0,007	293	293	0,007	8	120	4	0,0042345	341,53	☐		
26		0,007	293	293	0,007	6	80	4	0,0069919	335,86	☐		
27		0,007	293	293	0,007	8	80	4	0,004233	363,81	☐		
28		0,007	293	293	0,007	6	100	4	0,0059896	329,69	☐		
29		0,007	293	293	0,007	10	100	4	0,0054302	387,18	☐		
30		0,007	293	293	0,007	6	100	4	0,0069911	335,65	☐		
31		0,007	293	293	0,007	8	100	4	0,0052341	372,36	☐		
32		0,007	293	293	0,007	10	100	4	0,0044297	354,07	☐		
33		0,007	293	293	0,007	10	80	4	0,0044293	324,8	☐		
34		0,007	293	293	0,007	10	120	4	0,0044279	353,98	☐		
35		0,007	293	293	0,007	8	100	4	0,0052341	347,05	☐		
36		0,007	293	293	0,007	8	100	4	0,0052341	372,36	☐		
37		0,007	293	293	0,007	8	100	4	0,0062348	350,27	☐		
38		0,007	293	293	0,007	8	80	4	0,0052342	347,08	☐		
39		0,007	293	293	0,007	8	100	4	0,0052341	321,73	☐		

Figure 7. Parametric Result Table

In the images above, the outlet velocity and outlet temperature, which correspond to the given input parameters, were calculated using the parametric analysis method.

RESULTS

The research aims at investigating the effects of different inlet parameters, flow dynamics, and heat transfer on cylindrical pipes by employing a parametric analysis method. The numerical effects of the diameters of the input, the length of the pipe, and the input temperature on the output velocity, temperature distribution, and flow characteristic are found through simulations implemented with ANSYS Fluent software.

Based on the results obtained, the output speed has increased as the input diameter has reduced. For example, if the input is a 12mm diameter, the average output velocity will be calculated to be 0.0068 m/s. In contrast, the outlet velocity of 0.0042 m/s is already less when the input diameter is 20mm. The reason for this is the acceleration of the fluid in the narrowed portion and, as a result, the increase in the speed of the flow. In wider entrances, it has been observed that the fluid is moving with a lower velocity.

The effect of the inlet temperature was examined, and it was found that the temperature difference showed a linear change depending on the inlet temperature. For the input temperature 333 K, the output temperature was calculated as an average of 315 K, while the exit temperature for the input temperature 413 K was raised to the 398 K level exactly. This situation shows that the heat transfer occurring inside the pipe increases directly along with the inlet temperature, and the output temperature also increases similarly.

The increase in outlet velocity resulting from the decrease in inlet diameter is a consequence of the principle of conservation of mass, in which the diminishment of cross-sectional area elicits an escalation of flow under constant volumetric conditions. This observation complies with the classical laminar flow behavior seen in narrowing geometries. In the same way, the steady growth in outlet temperature as the inlet temperature rises signifies an appropriate surge in heat energy, which fuels convective heat transfer along the pipe walls. Therefore, in the absence of phase change or turbulence, convective heat transfer along the pipe walls is directly influenced. The temperature difference became less pronounced as the length of the pipe went up, which indicated a thermally developed flow regime. It was more or less the same for theoretical predictions of laminar regimes. Inlet radius and inlet temperature were the factors that primarily influenced the outlet velocity and the Nusselt number, indicating their pivotal roles in thermal system design.

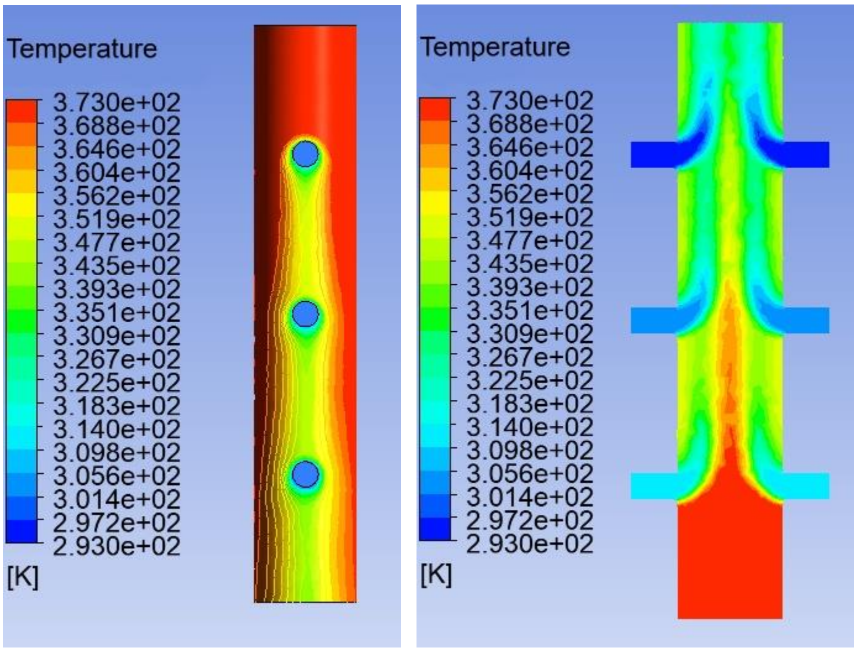


Figure 8. System Temperature Results

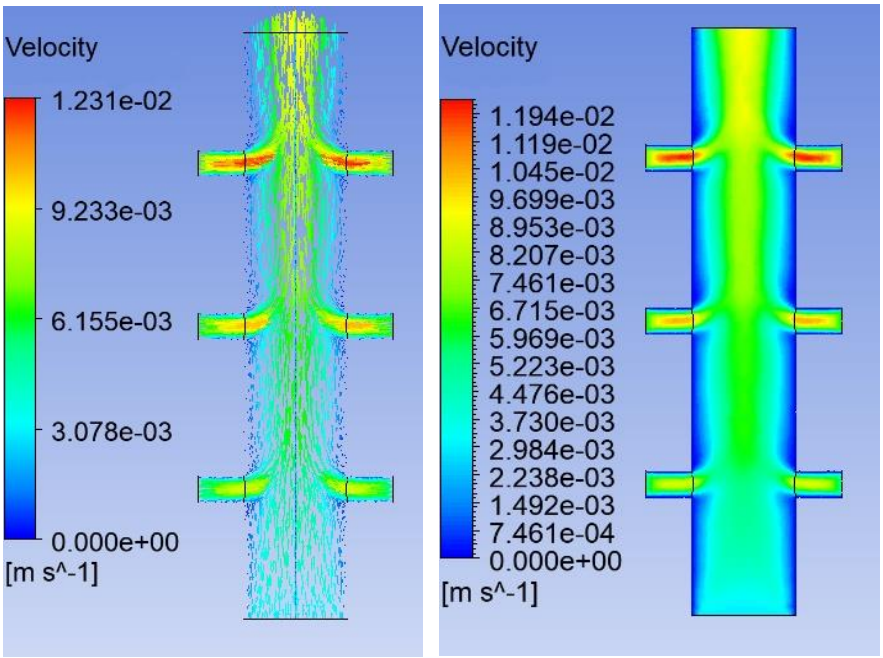


Figure 9. System Velocity Results

The impact that the length of the pipe had on the fluid behavior was also studied. In the 80 mm length pipes, the temperature difference was calculated to be 22 K, whereas in the 120 mm length pipes, this difference decreased to 14 K. The reflection from the pipe boundary to the flow direction was proportional to the entrance length, and hence, the higher entrance length led to a larger acceptance region of the pipe surface boundary and, in effect, made the flow a fully developed one. The vortexing flow boundary moves with the time-dependent velocity of the suspended particles, then it is not dependent on the particle radii.

According to the Reynolds number, the value changed between 120 and 320 depending on the diameter and rate of velocity. The Reynolds number is determined to be higher due to the increase in velocity in small-diameter pipes. These values confirm that the system under consideration is operating in the laminar flow region. In the laminar regime, flow continuity and heat transfer occur following the flow of the fluid layer touching the surface due to the dominance of viscous forces.

After a variety of parameters have been tested for it, the Nusselt number has fluctuated between 18.4 and 32.7. Increasing the Nusselt number was found in high inlet velocities, and this means that as a result of the speed of the flow increasing, the convective heat transfer coefficient also goes up. Nusselt number was also found to be bigger through narrow inlet diameters because the reason that velocity is higher.

Results are largely consistent with the approach taken in other studies in the area. It is known from the literature that small inlet diameters promote velocity gradients, resulting in intensified flow; the temperature difference depends on the pipe length, and the flow regime is directly linked to Reynolds and Nusselt numbers. Therefore, the present work agrees with the previous experimental and numerical studies made in the literature.

A regression coefficient above 0.95 is one of the solid proofs of the accuracy and trustworthiness of the obtained model. These analyses provide major data about optimizing pipe systems. For the studies to be conducted in the future, the turbulent flow regimes used for the analysis should be further detailed with different materials and expanded parameter ranges.

In a practical sense, the findings can guide the initial design of systems such as heat exchangers, microchannel cooling devices, and those in which laminar flow predominates. The parametric framework and statistical model introduced can be modified to perform the optimization tasks, thus allowing engineers to assess the thermal performance accurately and in a short time without having to create full-scale physical prototypes.

REFERENCES

Abdul Hassan, A. Y. (2016). *CFD study for cross flow heat exchanger with integral finned tube* [Master's thesis, University of Wasit].

Bhowmick, S., Kushan, D. S., & Sanyal, S. (2018). Parametric study of interaction effect between closely-spaced nozzles in a thin cylindrical pressure vessel. *International Journal of Pressure Vessels and Piping, 165*, 34-42. https://doi.org/10.1016/j.ijpvp.2018.06.002

Bisagni, C., Zaczynska, M., & Abramovich, H. (2020). Parametric studies on the dynamic buckling phenomenon of a composite cylindrical shell under impulsive axial compression. *Journal of Sound and Vibration, 482*, 115462. https://doi.org/10.1016/j.jsv.2020.115462

Celik, E., Karagoz, I., & Küpeli, S. (2020). Serpantin akış kanallı bir PEM yakıt pilinin üç boyutlu modellenmesi ve parametrik analizi. *Euroasia Journal of Mathematics, Engineering, Natural & Medical Sciences, 7*(8), 94-107.

Chatzistergos, P. E., Magnissalis, E. A., & Kourkoulis, S. K. (2010). A parametric study of cylindrical pedicle screw design implications on the pullout performance using an experimentally validated finite-element model. *Medical Engineering & Physics, 32*(2), 145-154. https://doi.org/10.1016/j.medengphy.2009.11.006

Doba, F., Oğulata, R. T., & Yilmaz, T. (2000). Irreversibility analysis of cross flow heat exchangers. *Energy Conversion and Management, 41*(15), 1585-1599. https://doi.org/10.1016/S0196-8904(99)00166-3

Erdinç, M. T., Yılmaz, T., Cihan, E., & Ünal, Ş. (2015). Isı Transferi Problemlerinin Ansys Workbench İle Hızlı Analizi Ve Kaydırılmış Levhada Örnek Uygulama. *Ulıbtk*, *15*(20), 1424-1230.

Jing, D., & Liao, W. (2023). Parametric study on the mixing and pressure loss of channel flow behind cylinder connected with Y-shaped elastic flag. *Chemical Engineering and Processing: Process Intensification, 184*, 109270. https://doi.org/10.1016/j.cep.2023.109270

Kadhim, Z. K., Kassim, M. S., & Hassan, A. Y. A. (2016). CFD study for cross flow heat exchanger with integral finned tube. *International Journal of Scientific and Research Publications*, *6*(6), 668-677.

Kadry, A. A., Ebid, A. M., Abdel-salaam, A. M., El-Ganzoury, E. N., & Haggag, S. A. (2022). Parametric study of Unstiffened multi-planar tubular KK-Joints. *Results in Engineering*, *14*, 100400. https://doi.org/10.1016/j.rineng.2022.100400

Li, M. Z., He, Y. P., Liu, Y. D., & Huang, C. (2020). Analysis of transport properties with varying parameters of slurry in horizontal pipeline using ANSYS fluent. *Particulate Science and Technology*, *38*(6), 726-739. https://doi.org/10.1080/02726351.2019.1621412

Li, Y., Wang, C., & Wang, R. (2012). The thermal stress analysis and structure optimum of neck tube with vertical cryogenic insulated cylinders based on ANSYS. *Nuclear Engineering and Design, 252*, 144-152. https://doi.org/10.1016/j.nucengdes.2012.06.015

Mangrulkar, C. K., Dhoble, A. S., Chamoli, S., Gupta, A., & Gawande, V. B. (2019). Recent advancement in heat transfer and fluid flow characteristics in cross flow heat exchangers. *Renewable and Sustainable Energy Reviews*, *113*, 109220. https://doi.org/10.1016/j.rser.2019.06.027

Nagesh, C., & Gupta, N. K. (2023). Structural response validation of composite cylindrical pressure vessels using FEA. *Materials Today: Proceedings*, *87*, 91-98. https://doi.org/10.1016/j.matpr.2022.12.248

Parlak, N., & Çelik, H. (2018). Kanatçık geometrisinin ısı geçişine etkisinin parametrik incelenmesi. *International Journal of Multidisciplinary Studies and Innovative Technologies, 2*(2), 25-29.

Qin, B., Wang, Q., Zhong, R., Zhao, X., & Shuai, C. (2020). A three-dimensional solution for free vibration of FGP-GPLRC cylindrical shells resting on elastic foundations: a comparative and parametric study. *International Journal of Mechanical Sciences*, *187*, 105896. https://doi.org/10.1016/j.ijmecsci.2020.105896

Rambhad, K. S., Kalbande, V. P., Kumbhalkar, M. A., Khond, V. W., & Jibhakate, R. A. (2021). Heat transfer and fluid flow analysis for turbulent flow in circular pipe with vortex generator. *SN Applied Sciences*, *3*(7), 709. https://doi.org/10.1007/s42452-021-04686-2

Sumner, D. (2010). Two circular cylinders in cross-flow: A review. *Journal of Fluids and Structures, 26*(6), 849-899. https://doi.org/10.1016/j.jfluidstructs.2010.07.001

Tan, X., Lyu, P., Fan, Y., Rao, J., & Ouyang, K. (2021). Numerical investigation of the direct liquid cooling of a fast-charging lithium-ion battery pack in hydrofluoroether. *Applied Thermal Engineering*, *196*, 117279. https://doi.org/10.1016/j.applthermaleng.2021.117279

Tunçel, O., Kahya, Ç., Tüfekçi, K. (2024). Optimization of Flexural Performance of PETG Samples Produced by Fused Filament Fabrication with Response Surface Method. *Polymers,* 16(14), 2020. https://doi.org/10.3390/polym16142020