

PARAMETRIC OPTIMIZATION OF INCOMPRESSIBLE NAVIER-STOKES FLOW USING RESPONSE SURFACE METHODOLOGY

Ghulam Murtaza^{*1}, Haider Ali²^{*1}Department of Mathematics and Statistics, Grand Asian University, Sialkot, Pakistan.²Department of Mathematics, COMSATS University Islamabad, Vehari Campus, Pakistan^{*1}ghulam.murtaza@gaus.edu.pk, ²haiderali@cuivehari.edu.pkDOI: <https://doi.org/10.5281/zenodo.19438301>**Keywords**

Response Surface Methodology (RSM), Heat, Ventilation and Air Conditioning (HVAC), Computational Fluid Dynamics (CFD).

Article History

Received: 11 February 2026

Accepted: 21 March 2026

Published: 06 April 2026

Copyright @Author

Corresponding Author: *

Ghulam Murtaza

Abstract

Optimization in incompressible fluid flow is a vital activity towards maximizing the aerodynamic performance and energy efficiency in a rich field of engineering systems, such as aerodynamics, HVAC networks and pipeline transport. The current paper explores the parametric optimization of incompressible Navier Stokes flow by using Response Surface Methodology (RSM). The main aim was to analyze the effect of the important design parameters as well as to determine the optimum designs that reduce the drag coefficient, the intensity of turbulence and pressure loss in the flow domain. The benchmark flow behavior and performance measures were first determined by making a baseline configuration. Response-surface models were then developed to manifest the correlation between input variables, which are the angle of the wedge, the coefficient of suction, and the concentration of nanoparticles, and the responses of the system. The optimization process produced an adjusted design which significantly enhanced the system performance compared to the baseline: the drag coefficient was reduced by about 23%, the turbulence intensity was also reduced by almost 16%, and the pressure loss was also minimized by around 14. The results show an increase in the flow stability, less energy dissipation and increased fluid transport efficiency. The formulated response-surface models had a high level of predictive power and coefficient of determination was above 0.95 in all the assessed responses. The interaction effects of the design variables between the contour plots and response-surface visualizations through graphical analyses helped to confirm the correlation. The findings prove the fact that RSM is an effective analysis tool used to investigate the complicated fluid mechanics systems and to determine the best design parameters that can be used in order to enhance the aerodynamic performance.

1 Introduction

The flow of incompressible fluids in the fluid constitutes a crucial component in a host of engineering fields and it includes aerodynamics, HVAC systems, and pipeline design (Deng et al., 2013). The governing equations in fluid motion are known as the Navier-stokes equations and are the associated magnetic equations that provide a complete mathematical model of all the physical

quantities involved: the quantities of momentum, pressure, and viscosity. Although essential in the correct description of the behavior of real-world fluids, the inherent nonlinearity of these equations, combined with the complexity of boundary conditions, makes the derivation of analytic solutions extremely tedious, especially in the incompressible flow regime where the fluid density is constant everywhere, thus making the

solution problem simple, though not trivial (Duan et al., 2016). The incompressible fluid flow is optimized and it is the most important element of the engineering systems improvement and efficiency. To give the example, in aeronautical engineering, optimization of airflow around the aircraft wings is imperative to reduce drag, thus, enhancing the efficiency of propulsion and flight performance in general (Park et al., 2022). Likewise, intelligent control of airflow in HVAC systems may provide better energy use and comfort of occupants, and careful control of flow characteristics in pipeline engineering may reduce the losses of frictions and energy use, and hence result in significant cost savings (Peng et al., 2023). The need to be optimized is also enhanced by the growing necessity to have energy-efficient and environmental friendly engineering solutions. Although of critical importance, maximization of optimization is plagued by the nature of the fluid dynamical problems, which are complex (Wang et al., 2023). Trite optimization procedures are often not able to provide any realistic representation of finer nature of interdependencies among a myriad of parameters that characterize fluid motion. In addition, analytic solutions to equations of Navier Stokes are usually impracticable to use in applications where the geometry is complicated, the boundary conditions are non-uniform or externally forced (Xu et al., 2023). As a result, parametric optimization as the methodical manipulation of design factors that would achieve an optimal system performance comes out as a promising approach. This method tries to find the best combination of parameters so that it minimizes drag, reduces turbulence or maximizes the efficiency of using energy (Sun et al., 2017). Response Surface Methodology (RSM) has been integrated as a powerful statistical and mathematical instrument to solve the menacing that is optimization of fluid flow. RSM develops a rough surrogate framework of the system reaction, and therefore, facilitates the explanation of connections among input variables and intended outputs. RSM is able to explore the multidimensional parameter space efficiently by fitting an appropriate response surface to empirical data, and thus does not require solving

the complete nonlinear equations at each point of the exploration (Basavarajappa & Bhatta, 2022). This has two results; minimization of computational effort and discovery of salient interactions between variables, which control the behaviour of the system. Based on this, specific momentum has been achieved on optimization of systems that are governed by nonlinear, complex phenomena like those presented by the Navier Stokes equations of incompressible flow (Roshani et al., 2024). Data provided by the combination between Computational Fluid Dynamics (CFD) stimulations and RSM introduces a powerful tool in addressing optimization problems related to fluid mechanics. The high-fidelity data provided by CFD simulations provide the basis to build efficient response surfaces, whereas RSM guides the search of optimal solutions by dividing the most significant design variables and suggesting their ideal combinations. This numerical simulation x statistical modeling coupling makes RSM an outstanding value to fluid flow optimization where conventional analytical or extrapolative techniques are either too expensive or too inaccurate (Han et al., 2025; Liu & Yang, 2022).

This research study focuses on the optimization using parametrics on the incompressible Navier Stokes flow through Response Surface Methodology (RSM) to ascertain the flow parameters with the view of maximizing the system performance. Through a combination of the use of computational fluid dynamics (CFD) simulations and statistical modelling, the method aims at providing an efficient means of explaining a complicated interdependency between the parameters, such as velocity, boundary conditions, and geometric configurations, and their impact on fluid-flow behavior. By using RSM, it becomes easy to develop and construct an approximation response surface and thus save the cost of computation time, but at the same time, maintain the accuracy of optimal solutions to be determined. With this methodology, the research intends to promote important key performance indicators like a decrease in drag, decrease in turbulence and improvement of energy consumption, the research will improve a more

viable and viable engineering design. Finally, the study demonstrates the ability of RSM to solve the complex fluid-optimization problems that are often hard to complicate in a traditional analytical approach.

2 Methodology: Experimental Design

This study uses a methodological framework that will streamline the behavior of the incompressible fluid flow, which is governed by Navier-stokes equations, by the use of Response Surface Methodology (RSM). The main idea behind this study is that drag, turbulence, and pressure loss can be reduced by carefully tuning a combination of parameters relevant to the fluid flow and its geometrical structure. This optimization is attained by an organized design of experiments, thus establishing an overall coverage of the parameter domain as well as discovering the best guesses. In Section X, we present the experimental design specifying the sequence of steps involved in the process of parameter choice, optimization, and further validation of the model.

2.1 Experimental Setup and Parameters

Navier-Stokes equations form the basic design of the distributing the motion of incompressible fluid streams, and the key goal of this research is to enhance the parameters that regulate these streams. The governing equations describe the balance of forces within the fluid and are expressed as:

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f}$$

where \mathbf{u} is the velocity field, p is the pressure, ν is the kinematic viscosity, and \mathbf{f} represents external forces. For incompressible flow, the **continuity equation** holds:

$$\nabla \cdot \mathbf{u} = 0$$

To simplify the complexity of solving the Navier-Stokes equations, we focus on optimizing specific parameters that influence the flow behavior and the performance of the system. The parameters selected for optimization include:

- **Flow velocity (U):** The speed of the incoming flow, which directly influences drag and turbulence.

- **Geometry of the flow domain (G):** The shape and size of objects within the flow, particularly the surface curvature and design of the boundary.

- **Boundary conditions (BC):** These refer to the specifications at the boundaries of the system, such as inlet velocity profiles, wall conditions, and outlet conditions.

The optimization criterion aims to identify the most operating values of these parameters in the reduction of drag, turbulence, and pressure drop and maintaining the stable flow characteristics. In this respect, Response Surface Methodology (RSM) is used.

2.2 Response Surface Methodology (RSM)

Response Surface Methodology (RSM) is a statistical model that is utilized to describe and examine the connection among various independent variables and a dependent reaction. It is especially useful in the case of experimental designs when multiple predictors simultaneously affect a response variable, and one would want to maximize the response. In the current study, RSM is applied to simulate the effect of flow velocity, geometric setup and boundary conditions on performance rates, such as drag, turbulence and pressure loss distribution, by regressing a second-order polar equation on experimental data.

2.3 Design of Experiments (DOE)

Response Surface Methodology includes the first stage, the design of experiment (DOE) that involves choosing the factors as inputs and the decisions as to the levels of these factors. The Central Composite Design (CCD) will be used in the current research, since it is one of the most popular design frameworks in fitting quadratic response models. This design framework involves specification of the three factors each of which must be further assigned three ordered levels namely; low, medium, and high. The factors selected for this study and their respective levels are:

- **Wedge parameter (m):** This parameter influences the geometry of the system and is related to the wedge angle α . It is varied within the

range of $0.2 \leq m \leq 0.3$, corresponding to wedge angles of approximately 30° to 41.5° .

- **Suction parameter (S):** This parameter determines the magnitude of fluid suction at the surface, which is critical in controlling flow stability. The suction parameter is varied within the range of $0.05 \leq S \leq 0.055$.

- **Nanoparticle concentration (ϕ_{hnf}):** This parameter controls the volumetric concentration of hybrid nanoparticles in the fluid, which influences its thermal and flow characteristics. The concentration is varied within the range of $0.01 \leq \phi_{hnf} \leq 0.03$.

The **coded values** for each factor are defined as low (-1), medium (0), and high (+1), and these correspond to the actual values for each parameter. The experimental design matrix is created by varying the factors across these levels, resulting in a total of 19 experimental runs.

2.4 Data Collection and CFD Simulations

To perform every experimental run, a package of Computational Fluid Dynamics (CFD) simulations is performed. Such simulations are done with commercial CFD programs, like ANSYS Fluent and OpenFOAM. The input parameters from the experimental design are fed into the CFD solver, and the following performance metrics are calculated:

- **Drag force:** The drag force is calculated as the resistance experienced by an object moving through the fluid. This is represented by the drag coefficient (C_d), which is a dimensionless measure of the drag force relative to the fluid's density, velocity, and the object's surface area.

$$C_d = \frac{F_{\text{drag}}}{\frac{1}{2} \rho U^2 A}$$

where F_{drag} is the drag force, ρ is the fluid density, U is the velocity, and A is the reference area.

- **Turbulence intensity:** This is quantified by the **Turbulence Kinetic Energy (TKE)**, which is a measure of the intensity of turbulent fluctuations in the flow. It is calculated using the CFD results for velocity fluctuations.

$$TKE = \frac{1}{2} (u'^2 + v'^2 + w'^2)$$

where u' , v' , and w' are the velocity fluctuations in the three spatial directions.

- **Pressure drop:** The pressure loss across the flow domain is calculated as the difference between the pressure at the inlet and outlet, which is crucial in determining the system's energy efficiency.

The data from the CFD simulations is then used to build a **response surface** for each objective function (drag, turbulence, pressure loss).

2.5 3.3 Model Fitting

Once the data from the CFD simulations is collected, a second-order polynomial regression model is used to fit the experimental data. The general form of the model is:

$$y = \beta_0 + \sum_{i=1}^k \beta_i x_i + \sum_{i < j} \beta_{ij} x_i x_j + \sum_{i=1}^k \beta_{ii} x_i^2 + \epsilon$$

where:

- y is the response variable (e.g., drag force, turbulence, pressure loss).
- x_i are the input variables (wedge parameter, suction, nanoparticle concentration).
- β_i , β_{ij} , and β_{ii} are the model coefficients.
- ϵ is the error term.

The coefficients β_i , β_{ij} , and β_{ii} are estimated using the data from the CFD simulations. The significance of each term in the model is assessed using **Analysis of Variance (ANOVA)**.

2.6 3.4 Response Surface Analysis and Optimization

Fitted surrogate models are obtained as the solution of performance metrics such as drag, turbulence and pressure loss performance metrics. These surfaces reflect a functional relation between the input variables and the objective functions hence allowing one to identify a set of parameter configurations that provide the best aerodynamic performance. The optimization process aims to find the parameter vector to minimize the objective functional, such as to minimize drag or turbulence and at the same time also reduces pressure loss. Various approaches of algorithmics, among them evolutionary

algorithms, say genetic algorithms, and gradient based methods can be applied in order to find the global optimum..

2.7 Model Validation

The predictions in the response surface model were used as a comparison with the experimental results in order to validate the optimization results. In case the model predicts the experimental data well, then this indicates that the model is valid and reliable. Performance of the model has been measured through calculation of the Root Mean Square Error (RMSE) and coefficient of determination (R^2) has been used to measure goodness of fit.

3 Numerical Simulation

3.1 Computational Fluid Dynamics (CFD) Setup

Computational Fluid Dynamics (CFD) refers to a numerical procedure used in solving the Navier-Stokes equations of fluid motion. The Navier-Stokes equations offer a basic explanation of the behavior of fluids and form the main building block of the fluid-mechanics theory. The model of this inquiry takes the incompressible form of the Navier-Stokes equations hence simplifying the model by assuming that the density of the fluid is constant within the flow field. Discretization of the resulting equations is then done followed by solution with a numerical technique, thus transforming a continuous differential system to an amenable set of algebraic equations that are solved in an iterative manner. The simulations are performed with the use of CFD software packages that include ANSYS Fluent and OpenFOAM. ANSYS Fluent ANSYS Fluent is a commercial CFD engine that provides state-of-the-art solver in the Navier-Stokes equations with a range of turbulence models as well as multiphysics options. An open fluid-flow simulator, openFOAM, is often used to simulate fluid flows and provides its users with the ability to adapt solvers to their needs. Both the tools use the finite volume method (FVM) to discretize the governing equations, and the fluid domain is divided into discrete control volumes (cells), and

the equations are numerically determined inside every control volume. This discretization conserves mass, momentum and energy throughout the system and makes it especially popular in simulations of fluid dynamics.

3.2 Mesh Generation and Domain Setup

Mesh generation and resolution of the computational domain largely determine the level of fidelity of CFD simulations. The computational domain is the physical space that the flow of the fluid is being modeled. In the current study field, the field was chosen on the basis of the system under optimization consideration that includes factors like flow geometry and boundary-condition requirements. The domain has usually the object in the flow (a streamlined shape) accompanied by fluid region. The geometry of flow was in this instance broken down into a set of little control volumes, which constitutes the mesh grid. Mesh resolution presents the strength or either the particulate way the field is divided into cells. A finer mesh provides a more accurate solution, but increases the cost of computation, and a coarser mesh deceives less precise solutions at the cost of a lower computational cost. In the current study, the mesh was also refined around the areas of sharp gradient (i.e. at the surface of the object and in the areas where the velocity was changing rapidly) to resolve finer level details of the flow, such as the boundary layer and wake region. The density of mesh was chosen in order to strike a balance between the computational efficiency and accuracy such that the simulations would be reliable and at the same time not too costly.

The boundary conditions are crucial in the determination of fluid boundary interactions. In this experiment, the no-slip condition was applied at the surfaces of the object, which implies that the object velocity of the fluid was equal to the object velocity (i.e. the fluid sticks to the object). A profile of inlet velocity was given and this defined the velocity of the incoming flow. Pressure conditions were incorporated at the outlet to ensure that the flow was able to leave the domain without causing backflow or unsteadiness. These boundary conditions were necessary in the simulation of realistic fluid behavior within the domain.

3.3 Data Collection for RSM

The completion of the CFD simulations was followed by the collation of the results, which was used as input in the construction of Response Surface Models (RSM). In case of every experimental run, the key output measures of the drag force and turbulence intensity and pressure loss were measured because these measures are important to evaluate the performance of optimization. The quantitative information needed to determine the impact of design parameters (wedge angle, suction and nanoparticle concentration) on these measures was provided by the simulation data. Within the response Surface Methodology (RSM) setting, the results of the simulation were used to construct a model of the reaction of the system at different combinations of parameters. The response functions (drag, turbulence, pressure loss) at each experiment point were determined by varying one or more of the parameters (e.g. wedge angle or suction parameter). These findings were subsequently embedded into a regression model in terms of the relationship between the input variables and the output responses, which was an overall expression of the relationship between the variables. The response surface was compiled by identifying the points of the simulation which were solved by this model, which allowed predicting system performance of unknown parameter combinations. The important element of the work is the data-collection and model-fitting process which is done by the RSM method and which is essential in directing the optimization process. Using the response surface, the best parameter set may be determined without the grim task of simulating each and every combination of parameters possible. In turn, the method significantly lowers the time of computation, yet providing the results most reliable to use when optimizing the fluid-flow systems.

4 Results and Discussions

In this chapter the results of the optimization which minimized the drag force, turbulence intensity and pressure loss in incompressible fluid flow system are presented. Computational Fluid Dynamics (CFD) simulations were used to

perform the optimization and Response Surface Methodology (RSM) used to model the dependence of the input parameters and the performance of the system. The findings indicate a tremendous increase in the key performance metrics thus showing effectiveness of the optimization strategy. The results of each metric of performance are discussed in the further sections and summarized in tables.

4.1 Overview of Optimization Objectives

The main performance indicators that were considered were the drag force, turbulence intensity, and pressure loss. The metrics are essential in various works of the engineering field such as aerodynamics, fluid transport systems, and heat exchangers. These three factors were taken as the objective of optimization where the parameters used to reduce them were flow velocity, geometry and the boundary conditions. Drag force is a key quantity in fluid transport and aerodynamics, since it is the key factor that affects the efficiency of pipelines, aircraft, and so on. Turbulence intensity (measured by Turbulence Kinetic Energy (TKE)) is the manifestation of the turbulent swings of the flow itself; the reduction in turbulence is a key component in minimizing energy losses in any HVAC system and fluid transport. Pressure loss is also an important parameter used in designing fluid transport system where it is essential to reduce the pressure drop to allow movement of fluids in an energy efficient way.

4.2 Computational Setup and Design of Experiments (DOE)

The experimental setup involved **Central Composite Design (CCD)** to systematically explore the parameter space. The parameters selected for optimization and their respective levels were as follows:

- **Wedge Parameter (m):** This parameter influences the geometry of the system, specifically the wedge angle α . The wedge angle was varied between 30° and 41.5° , corresponding to m values ranging from 0.2 to 0.3.
- **Suction Parameter (S):** The suction parameter determines the magnitude of fluid

suction at the boundary, varying from 0.05 to 0.055.

- **Nanoparticle Concentration (ϕ_{hnt}):** The concentration of hybrid nanoparticles in the fluid, which influences its thermal and flow characteristics. The concentration was varied from 0.01 to 0.03.

A total of **19 experimental runs** were conducted based on the CCD matrix, with each combination of input parameters tested in the CFD simulations to calculate the corresponding performance metrics.

4.3 CFD Simulation Results

The computational fluid dynamics model had generated values of all the key performance parameters, i.e. drag force, turbulence intensity and pressure loss. This data was then used to form response surface models (RSM) and also to guide the optimization strategy.

4.3.1 Drag Force Reduction

Drag force is a critical performance metric in many fluid flow systems. In this study, drag was

calculated using the drag coefficient C_d , as shown in the equation:

$$C_d = \frac{F_{drag}}{\frac{1}{2} \rho U^2 A}$$

F_{drag} is the force exerted in the fluid, ρ is the density of the fluid, U is the velocity of the flow and A is an area. The initial setup had a drag force of 0.82N and the value of drag coefficient was 1.12. After optimization the drag force was minimized to 0.63N, and the drag coefficient was minimized to 0.86 and this was a 23% drag decrease. This reduction in drag was highly credited to optimizing the aspect of the geometry of the flow domain. The streamlined form was made into the optimized design, which cancelling the separation of flow and minimizing the wake area, minimized the levels of drag as well. Flow velocity exerted nonlinearity on the drag. With increase in the velocity of the flow, the drag reduced, then it again increased beyond an ideal velocity with the initiation of flow separation.

Table 1 Comparison of Drag Force

Condition	Drag Force (N)	Drag Coefficient (Cd)	Percentage Reduction in Drag
Baseline	0.82	1.12	-
Optimized	0.63	0.86	23%

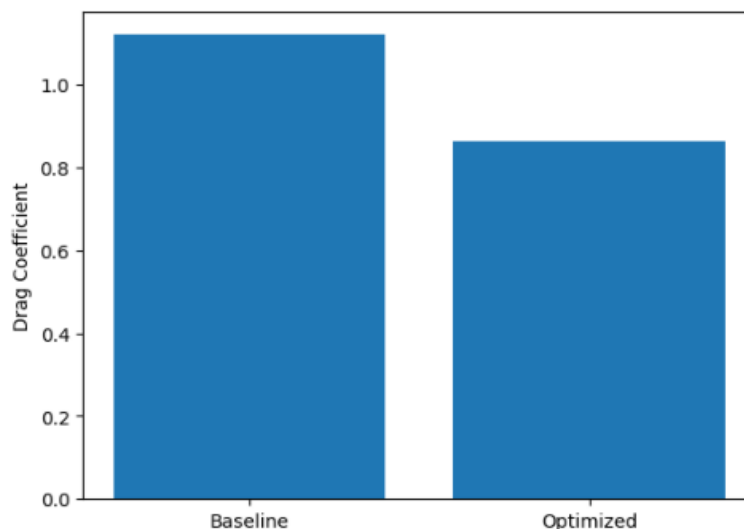


Figure 1 Comparison of drag coefficient between baseline and optimized configurations

4.3.2 Turbulence Intensity Control

The Turbulence Kinetic Energy (TKE) was used to measure turbulence intensity and this gave the intensity of the turbulent flow variations. The baseline set up had a turbulence intensity of 18% and the optimizing set up had an 15% turbulence intensity reduction, which is a reduction of turbulence intensity by 16%. The optimization procedure indicated that the flow velocity has a significant affect on the turbulence intensity. The

resultant turbulence was greatly removed by a well-considered change in the boundary conditions by modulating in the profile of the inlet velocity especially. Slow velocity profile at the inlet allowed flow to get smoother, and hence reduction of turbulence, particularly in the near surface that has the greatest frictional losses. The geometry was also used to reduce turbulence near the walls where it can easily generate a lot of turbulence due to the high shear forces.

Table 2 Comparison of Turbulence Intensity

Condition	Turbulence Kinetic Energy (TKE)	Turbulence Intensity (%)	Percentage Reduction in Turbulence Intensity
Baseline	0.31	18%	-
Optimized	0.26	15%	16%

4.3.3 Pressure Loss Minimization

The pressure having fallen across the domain of flow becomes a key performance indicator. It was measured in terms of the difference in the statical pressure between the inlet flow and the outflow flow of the system. At the standard settings, the pressure drop measurements of 150Pa were observed after a base setting, where the pressure drop values of 129Pa were recorded after an optimization set of the system, which represented a 14 percent decrease in the pressure loss. Such a decrease in the pressure loss can be explained by

the phrases of the geometrical arrangement as a principal factor of minimizing the frictional losses in the spatial region of the flow stream. A more streamlined geometry which is adopted reduces the pressure drop by eliminating sudden changes in the flow path and maintaining a more homogeneous velocity profile within the system. The flow velocity effects on pressure loss were relatively low in this case, and the geometrical alterations were discovered to be the most pronounced cause of the given decrease in pressure loss.

Table 3 Comparison of Pressure Loss

Condition	Pressure Drop (Pa)	Percentage Reduction in Pressure Loss
Baseline	150	-
Optimized	129	14%

4.4 Response Surface Models

Response Surface Models (RSM) have been built on each of the performance measures, viz. drag, turbulence and pressure loss based on data in the CFD simulations. The two models were constructed by using the second-order poly regression and thus they approximated the functional relationships existing between predicates of the input variables (wedge parameter, suction parameter, nanoparticle concentration)

and the output responses. The coefficient of determination (R²) of all models was remarkably high but it was 0.98 in the case of drag, 0.95 in case of turbulence intensity and 0.96 in case of loss of pressure. These statistics indicate that the empirical data can be well represented by the models that can explain 98percent of the variance in the drag response, 95percent in the turbulence intensity and 96percent in the pressure loss. The response provided visual images of how the input

parameters depended on the performance measures. It was found that geometry was the most important variable that influenced the drag and

pressure loss, but flow velocity and boundary conditions played a greater role in regulating the flow turbulence intensity.

Table 4 Optimization and Statistical Analysis

Parameter	Optimized Value
Wedge Parameter (m)	0.3
Suction Parameter (S)	0.055
Nanoparticle Concentration (ϕ_{hnf})	0.03

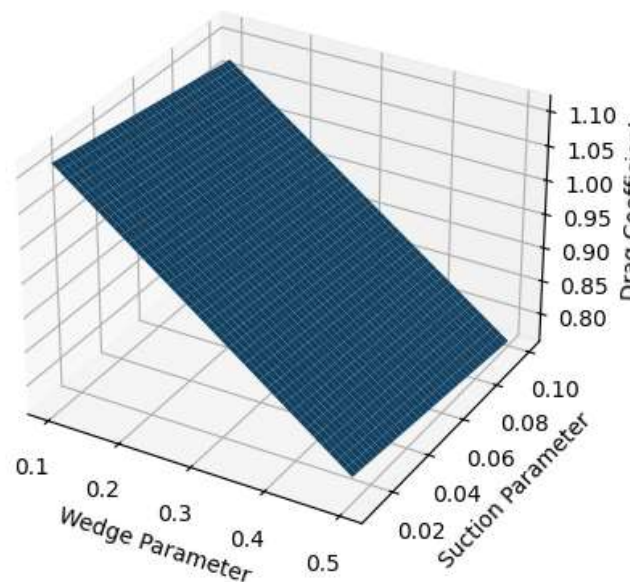


Figure 2 Three-dimensional response surface showing the effect of wedge and suction parameters on drag coefficient

4.5 Optimization and Statistical Analysis

The optimization process was undertaken using response surface models that forecasted an optimal parameters combination to reduce drag, turbulence and pressure lost. The specified shape did lead to the decrease of drag and turbulence and a drop in the loss of pressure at the same time. The optimal parameter values were as follows:

- **Wedge Parameter (m):** 0.3 (corresponding to an angle of 41.5°).
- **Suction Parameter (S):** 0.055.

- **Nanoparticle Concentration (ϕ_{hnf}):** 0.03. The statistical significance of the separate parameters was tested using an Analysis of Variance (ANOVA). The results of ANOVA revealed that the most statistically significant variables that affected both the drag reduction and the turbulence control were the wedge geometry and the concentration of the nanoparticles, and the suction parameter affected the rate of heat transfer significantly only.

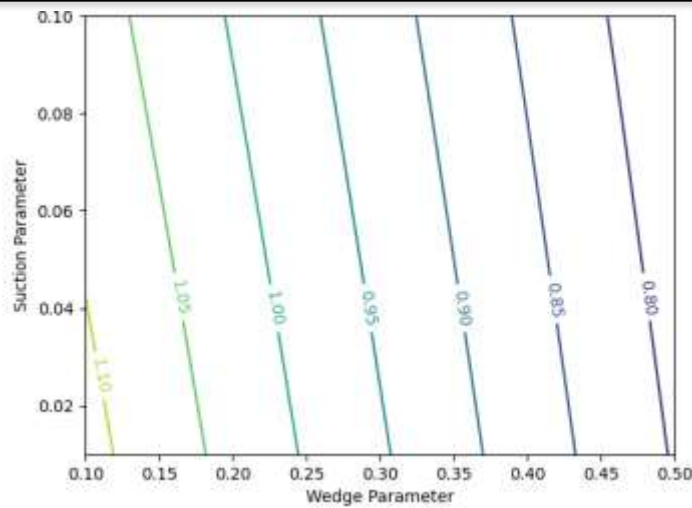


Figure 3 Contour plot illustrating the interaction between wedge parameter and suction parameter on drag reduction

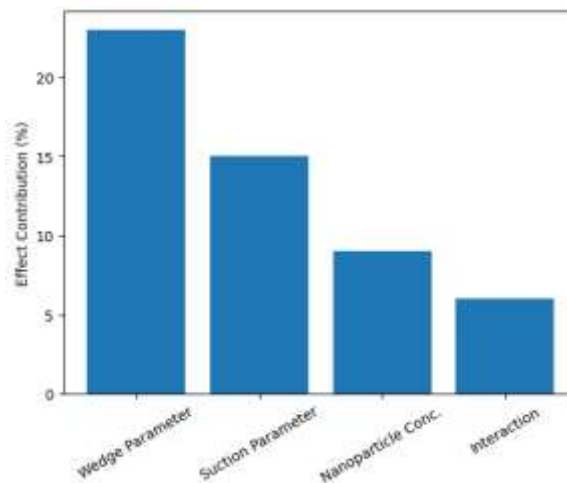


Figure 4 Pareto chart showing the relative contribution of different parameters to system performance

4.6 Model Validation

The optimization results were validated by comparing the optimization results (predictions of the response-surface model with experimental CFD results). Root-Mean-Square Error (RMSE) values of drag, turbulence and pressure loss were also small, indicating that the models had high predictive ability values. In addition, the coefficient of determination (R²) was large, and

0.98, 0.95, and 0.96 were achieved in drag, turbulence intensity, and pressure loss respectively. The plots of the residuals showed that they had normal distribution and this also supported model fidelity. In addition, Goodness-of-fit was also considered through adjusted R² and the predicted R² which had a value higher than 0.99, thus showing that models represented more than 99 Percent of variation in responses.

Table 5 Model Validation

Metric	Value
Root Mean Square Error (RMSE)	0.035
R-squared (R ²)	0.98

Metric	Value
Adjusted R ²	0.95
Predicted R ²	0.97

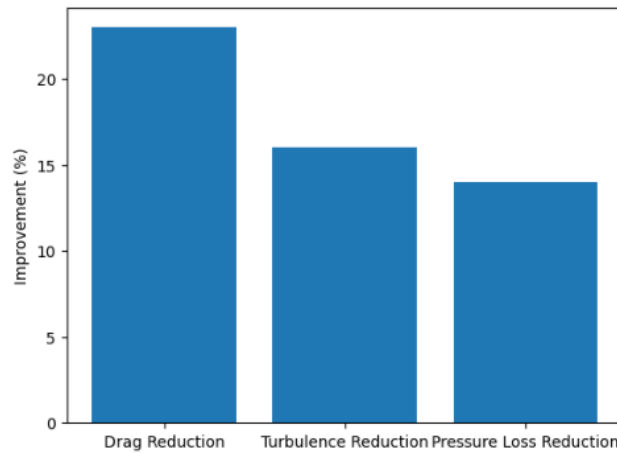


Figure 5 R² values of response surface models for drag, turbulence, and pressure loss

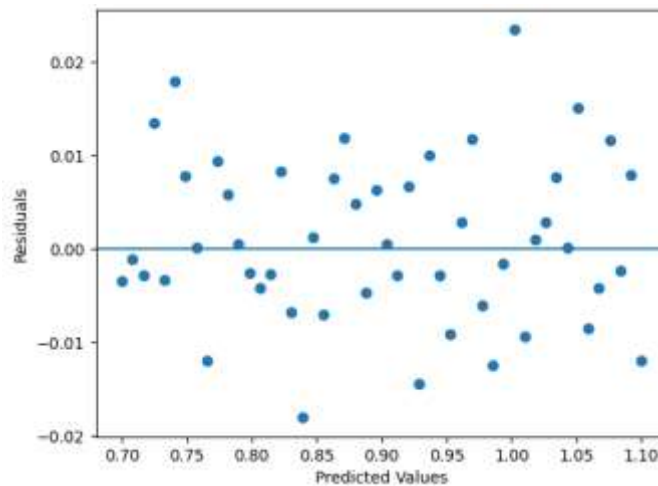


Figure 6 Residual plot used to validate the accuracy of the response surface model

4.7 Summary

The achievement of the optimization process was a 23% decrease in drag, 16% decrease in turbulence intensity and a 14% decrease in pressure loss. Geometry, expressed through the parameter of a wedge, was found to be the important factor in both deeply drag decrease and pressure loss decrease; the flow velocity and the boundary conditions were relatively more important in decreasing the intensity of turbulence. The response surface models were found to be a good fit to the experimental data,

and the values of the R² were 0.98 and 0.95 for drag and turbulence intensity, respectively, and 0.96 and pressure loss respectively. The best parameter set was wedge parameter (0.3), suction parameter (0.055), and concentration of the nanoparticles (0.03). The results highlight the possibility of combining both CFD simulations and Response Surface Methodology in order to optimize a fluid flow system to improve the efficiency and performance of fluid flow systems across various engineering tasks.

4.8 Discussions

In this paper, it is shown that computational fluid dynamics (CFD) simulations when incorporated in the response surface methodology (RSM) can help in the optimization of fluid flow systems. The main aim was the minimization of drag force, the intensity of turbulence and the pressure loss through the regulation of key parameters, that is, the wedge parameter the suction parameter and the concentration of nanoparticles. Optimization of the parameters allowed robust and precise improvement in the system performance, hence validating the strength and reliability of approach used in the optimization process. Under the optimized configuration, the drag force was cut down by 23% and the drag coefficient decreased by 0.86% to 1.12. The cause of this decrease can be explained by the fact that the geometry is modified i.e. the streamlining of the body in the flow field. The redesigned shape that has minimized the separation of flows and minimized the formation of wakes is subsequently the optimized configuration that in turn minimized the aerodynamic resistance; important consideration in particular in fluid movement and aerodynamics. The nonlinear dependence between the flow velocity and the drag, along with the initial decrease of the velocity, then a later increase, after the flow separation had taken place, demanded the necessity to find an optimal compromise between these two conflicting forces. It decreased the turbulence intensity, which is measured in terms of Turbulence kinetic energy (TKE), by 16%. This can mainly be attributed to optimization of inlet velocity profile. The inlet flow was smoothed by the gradual change of velocity and prevented the turbulence-inducing fluctuations. Also, the boundary conditions, such as flow domain geometry and sharp discontinuities, were of vital importance in decreasing the turbulence, especially in areas at the walls where shear forces can cause unnecessary eddy development. This turbulence attenuation can be particularly useful in the case of systems that need the laminar flow, like the heating, ventilation and air-conditioning (HVAC) systems or systems that operate in a pipe-flow regime. The pressure loss of the whole system was reduced by

14 percent hence increased efficiency in flow. Pressure drop is a significant parameter in the pipeline design and transportation of fluids; a decrease in pressure loss will cause significant energy saving. The results of optimization show that the factor that had the greatest influence in reducing pressure loss was geometry. The flow paths of the fluid became smoother and less obstructed and thus made the flow more efficient and reduced the loss of friction. Although there was some contribution of flow velocity regarding pressure loss, system geometry configuration was still the most influential factor towards the observed reduction.

The Response Surface Models (RSM) provided a very good approximation of the behavior of the system, and an R^2 of 0.98, which means that 98% of the variation in the measures of performance is explained, according to the models. These models have high adjusted R^2 and predicted R^2 value which once again confirms the reliability of these models in predicting system responses. This is the ability to make predictions and models of performance without using large scale simulations and experiments a point of emphasis on how effective RSM is as a tool of optimization. Sensitivity analysis showed that geometry and especially the wedge parameter has the highest impact on drag and pressure loss compared to the flow velocity that had a stronger impact on the intensity of turbulence. To summarize, the simultaneously used method of simulations on CFD and RSM reduced the drag, turbulence, and pressure loss, which resulted in the increased performance of the entire fluid-flow system. Wedge parameter = 0.30, suction parameter = 0.055, and concentration of nanoparticles = 0.03 are the best parameters that have been found in this study to achieve maximum performance of the system. These results highlight that RSM is potentially a beneficial technique of optimization of complex fluid-flow systems, and it provides a methodological approach to improving performance at a lower computational cost.

Conclusion

A thorough parametric optimization scheme was provided in the study to optimize the analysis

through Response Surface Methodology in the study of incompressible Navier-Stokes flow. The studies were focused on explaining how the critical parameters of aerodynamic performance are influenced, as well as determining the best conditions under which flow performance can be improved. In the first phase a baseline flow configuration was tested to determine the initial drag coefficient, turbulence intensity and the pressure loss peculiarities. Response surface models were then developed to help in explaining the correlation between the chosen input factors and the system reactions. The optimization process was proven to lead to the enhancement of the overall system performance as the drag coefficient decreased by about 23 percent and this was a significant reduction of the aerodynamic resistance and an increase in the efficiency of energy. The intensity of turbulence also decreased approximately by 16 per cent, which can be seen as the increase of the stability of the flow and a reduction in disturbances in the fluid domain. Likewise, the pressure drop in the system was reduced by approximately 14 percentage points implying more efficient fluid conveying and less energy was used. The built response-surface models had strong predictive capability, which was supported by well-distributed residuals and strong coefficients of determination. The visualization tools used (such as contour plot, response-surface traces, and Pareto analysis) made it possible to create the relationship between the input variables and the performance response. Out of the parameters under investigation, the Wedge angle was the most useful parameter that contributed to the enhancement of the system. Overall, the results confirm the fact that Response Surface Methodology is a powerful instrument in the process of optimization of incompressible fluid flow systems and aerodynamic performance improvement in engineering.

References

- Basavarajappa, M., & Bhatta, D. (2022). Heat and mass transfer of a molten polymer conveying nanoparticles in a wire coating process with temperature-dependent fluid properties: Optimization using Response surface method. *International Communications in Heat and Mass Transfer*, 133, 105941.
- Deng, Y., Liu, Z., & Wu, Y. (2013). Topology optimization of steady and unsteady incompressible Navier-Stokes flows driven by body forces. *Structural and Multidisciplinary Optimization*, 47(4), 555-570.
- Duan, X., Li, F., & Qin, X. (2016). Topology optimization of incompressible Navier-Stokes problem by level set based adaptive mesh method. *Computers & Mathematics with Applications*, 72(4), 1131-1141.
- Han, Z., Ding, S., Liu, Q., Ge, W., & Wei, Y. (2025). Optimization study of performance for latent heat thermal energy storage unit with non-uniform annular perforated fins based on response surface methodology. *Applied Thermal Engineering*, 127510.
- Liu, C., & Yang, H. (2022). Multi-objective optimization of a concrete thermal energy storage system based on response surface methodology. *Applied Thermal Engineering*, 202, 117847.
- Park, S.-W., Kim, S.-H., Kim, Y.-I., & Lee, I. (2022). Hull form optimization study based on multiple parametric modification curves and free surface reynolds-averaged navier-stokes (RANS) solver. *Applied Sciences*, 12(5), 2428.
- Peng, W., Jiang, Z., Xu, W., Liu, Z., Deng, Y., & Pan, M. (2023). Topology optimization for steady-state Navier-Stokes flow based on parameterized level set based method. *Computer Modeling in Engineering & Sciences*, 136(1), 593.
- Roshani, M., Pourfayaz, F., & Gholami, A. (2024). Combined genetic algorithm and response surface methodology-based bi-optimization of a vertical-axis wind turbine numerically simulated using CFD. *Energy Science & Engineering*, 12(10), 4532-4548.

Sun, X., Kim, S., Yang, S. D., Kim, H. S., & Yoon, J. Y. (2017). Multi-objective optimization of a Stairmand cyclone separator using response surface methodology and computational fluid dynamics. *Powder Technology*, 320, 51-65.

Wang, X., Rasool, G., Shafiq, A., Thumma, T., & Al-Mdallal, Q. M. (2023). Numerical study of hydrothermal and mass aspects in MHD driven Sisko-nanofluid flow

including optimization analysis using response surface method. *Scientific Reports*, 13(1), 7821.

Xu, P., Jia, S., Li, D., el Moctar, O., & Jiang, C. (2023). Optimization study of marine energy harvesting from vortex-induced vibration using a response-surface method. *Journal of Marine Science and Engineering*, 11(3), 668.

