DEPARTMENT OF MECHANICAL ENGINEERING

ME395 – ENGINEERING

PRACTICUM

ENDSEM REPORT



DATE OF SUBMISSION: 30/04/2025

NAME OF OUR MENTOR : DR. Md. KALEEM KHAN

NAME OF OUR CO-MENTOR : DR. ASHWANI ASSAM

GROUP MEMBERS:

* ASHISH SINGH – 2201ME17
* ASHWANI KUMAR JHA – 2201ME19
* AYUSH SONI – 2201ME22
* BISHWAMOHAN JENA – 2201ME26
* NIKHIL SINGH – 2201ME42
* YASHKRIT SINGH – 2201ME76

# ME395 – ENGINEERING PRACTICUM REPORT

**Title:**

**Aerodynamic Analysis and AI-based Prediction of S-Duct Performance**

**Abstract:**

The aerodynamic optimization of S-ducts plays important role in improving aircraft performance, efficiency, and stealth capability. This project investigates the behaviour of different S-duct geometries using combination of Computational Fluid Dynamics (CFD) simulations and Artificial Intelligence based predictive modeling. Multiple S-duct configurations, including circle-circle and square-circle shapes with varying bend angles was analyse at a flight Mach number of 0.6. Key aerodynamic parameters like outlet velocity, pressure distribution and centerline velocity were studied.   
This study found that circle-circle S-ducts are more suited for commercial aircrafts as they gives more uniform flow profile, which help to improve engine stability and efficiency. On the other hand, square-circle S-ducts, even though they have slightly lower pressure recovery and fuel economy, offer better thrust and nicer shape integration, making them more fit for fighter jets where stealth and manoeuvrability matter more.  
  
An AI-based prediction model using XGBoost was made to quickly estimate outlet performance metrics, greatly reducing the need of long CFD simulations. Also, an interactive web-based platform was build to visualise and compare aerodynamic datas across different configurations. The results shows how integrating CFD, AI, and web technologies can fasten the design and optimization of aerospace components like S-ducts.1. Introduction

**1.1 Background**

S-ducts (serpentine inlets) are crucial components in aerospace engineering, especially for high-performance aircraft. These curved ducts are used in military jets, business jets, and commercial airliners to optimize air intake by redirecting airflow to the engine. Their design improves stealth by reducing radar visibility, enhances aerodynamics, and allows for more compact engine placements, such as in the Boeing 727 and Dassault Falcon 7X. Despite these advantages, the curvature of S-ducts presents challenges, including flow distortion, pressure losses, and boundary layer separations, which can negatively impact engine performance and aircraft stability.

**1.2 Motivation**

As aerospace systems demand greater performance, efficiency, and stealth, optimizing components like S-ducts becomes essential. The complexity of flow within these ducts, caused by bends and secondary flows, can lead to issues such as engine surge and compressor stall. Traditional experimental methods are expensive and time-consuming, making Computational Fluid Dynamics (CFD) a valuable tool for analyzing and optimizing these designs. Additionally, integrating Artificial Intelligence (AI) enables faster optimization by predicting key aerodynamic parameters without extensive CFD simulations.

**1.3 Challenges in S-Duct Design**

S-ducts present several design challenges:

* Flow Distortion: Secondary flows lead to non-uniform velocity distributions at the engine inlet, affecting performance.
* Pressure Losses: Frictional and separation losses reduce energy available for thrust production.
* Boundary Layer Separation: Adverse pressure gradients cause separation, exacerbating distortion and losses.
* Structural Constraints: Integration of S-ducts often requires trade-offs in size and weight.
* Simulation Complexity: Accurately simulating turbulent, three-dimensional flow requires high-fidelity models.

**1.4 Objectives of the Study**

This study aims to address these challenges using CFD and AI-based predictive modeling. The objectives include designing and simulating various S-duct geometries, analyzing key aerodynamic parameters, developing a web platform for visualizing results, and training AI models to predict aerodynamic performance.

**1.5 Scope of the Work**

The scope includes creating 3D models of S-ducts, conducting CFD simulations, developing an interactive web platform, and validating AI models for aerodynamic prediction. This integrated approach will optimize S-duct design by minimizing flow distortion and pressure loss, ultimately enhancing performance and reducing design cycles.

**2. Literature Review**

Research on S-duct design has focused on managing flow challenges like separation, secondary flows, and flow distortion. These ducts, widely used in military and civil aviation, face performance issues due to their complex geometry.

Early studies, such as those by Seddon and Goldsmith (1999), explored how duct geometry, especially curvature, affects flow distortion and pressure recovery. Experimental models highlighted flow separation and vortex formation as key concerns.

With the rise of Computational Fluid Dynamics (CFD), predicting and optimizing S-duct performance improved. Traditional models like k-ε struggled with strong pressure gradients, leading to the adoption of more advanced models like k-ω Shear Stress Transport (SST), which better handled separation in curved flows.

A key study by Wenbiao Gan and Xiaocui Zhang (2017) introduced a modified k-ω SST model for optimizing S-ducts, improving total pressure recovery and reducing flow distortion. Their automated optimization framework reduced flow distortion by 16.3% and increased pressure recovery by 1.1%.

Recent trends incorporate AI and machine learning, like the work by Guo et al. (2020), using neural networks to predict aerodynamic metrics, which speeds up optimization and reduces reliance on CFD simulations.

Despite progress, challenges like accurate prediction of turbulent flows and the need for fine mesh resolutions persist. Overall, research has evolved from experimental to hybrid approaches, combining CFD, optimization, and AI for efficient S-duct design.

**3. Fundamentals of S-Duct**

**3.1 Introduction to S-Ducts**

S-ducts are curved inlet ducts used to guide airflow to engines without a direct line of sight. Common in military aircraft, business jets, and tri-jet airliners, they improve stealth, structural integration, and aerodynamics.

**3.2 Importance and Applications**

S-ducts help with radar signature reduction, structural efficiency, noise shielding, and ensuring sufficient engine airflow. However, they introduce complex aerodynamic challenges due to their curvature.

**3.3 Aerodynamic Challenges**

* **Flow Separation**: Adverse pressure gradients cause airflow detachment, leading to pressure loss and compressor risks.
* **Secondary Flows**: Centrifugal forces create vortices, distorting the main flow.
* **Flow Distortion**: Non-uniform inlet flow can reduce engine performance and stability.

**3.4 Key Design Considerations**

* **Centerline Curvature**: Smoother curves minimize separation.
* **Cross-sectional Area Variation**: Carefully managed contraction and diffusion prevent separation.
* **Wall Shape and Treatment**: Smooth surfaces and flow control devices help maintain attached flow.

**3.5 Fundamental Flow Quantities**

* **Total Pressure Recovery (PR)**: Indicates duct efficiency.
* **Distortion Coefficient (DC)**: Measures flow uniformity.
* **Boundary Layer Thickness** and **Y+**: Critical for separation prediction and CFD mesh quality.

**3.6 Modern Approaches**

CFD, especially with the k-ω SST model, is widely used for S-duct analysis. Advances like improved SST models and AI-based optimization are making S-duct design faster and more accurate.

**4. Challenges in S-Ducts**

S-ducts present several challenges due to their complex geometry and flow behavior, especially in high-performance applications like aerospace systems.

**4.1 Flow Separation and Pressure Losses**

Flow separation occurs when the flow detaches from the duct walls, particularly in the curved sections, due to an adverse pressure gradient. This results in turbulent eddies and increased friction, leading to higher pressure losses. These losses impact energy efficiency and system stability, especially in critical systems like turbines.

**4.2 Velocity Redistribution and Flow Distortion**

The curvature of the S-duct causes velocity redistribution, with higher velocities near the outer walls and lower velocities near the inner walls. This distortion can affect downstream components like compressors and turbines, leading to performance issues such as surge or failure if not managed properly.

**4.3 Increased Turbulence and Flow Instability**

Turbulence is intensified by the bends, creating vortices and swirling flows. This turbulence increases energy losses, disrupts flow stability, and can cause vibrations, noise, and erosion of the duct walls, impacting long-term performance and durability.

**4.4 Design Complexity and Computational Demands**

The design of an S-duct requires balancing several factors, including curvature, cross-sectional area, and straight section length, to optimize performance. CFD simulations are often used to model these complexities, but they are computationally demanding, requiring high computational power and precise manufacturing to meet design specifications.

**5. Computational Fluid Dynamics (CFD) Approach**

Computational Fluid Dynamics (CFD) is essential for designing and optimizing S-ducts, offering detailed insights into airflow behavior, pressure distributions, and velocity profiles. By simulating airflow in three-dimensional space, CFD helps engineers evaluate designs before constructing physical prototypes, addressing challenges like flow separation, turbulence, and pressure loss.

**5.1 CFD Simulation Setup for S-Ducts**

The CFD process begins with geometry creation, where an accurate model of the S-duct is built. This model includes the duct's dimensions, curvature, and inlet/outlet conditions. The next step is mesh generation, where the duct is divided into smaller control volumes to solve governing equations. Finer meshes are applied in high-curvature regions, such as bends, to capture detailed flow characteristics.

The boundary conditions (inlet, outlet, and wall) are then defined to simulate the duct’s real operating conditions. A flow solver is chosen based on the flow nature (compressible or incompressible) and the turbulence model selected. Turbulence modeling is crucial for accurately simulating the chaotic flow patterns typically present in S-ducts. Models like k-ε or k-ω SST are commonly used to handle turbulence and flow separation effectively. Finally, the flow equations are solved iteratively until convergence criteria are met.

**5.2 Post-Processing and Analysis**

Post-simulation analysis involves visualizing velocity and pressure distributions to identify high-velocity regions and pressure drops. Flow separation and secondary flows are analyzed to understand turbulence and vortex formation in the bends. Key performance metrics like pressure losses and total pressure recovery are also calculated to assess design efficiency.

**5.3 Optimization**

CFD is invaluable for optimizing S-duct designs by testing various geometries and conditions to minimize pressure losses and improve flow characteristics.

**6. Geometry Creation**

Geometry creation is a crucial step in Computational Fluid Dynamics (CFD) simulations as it defines the physical space where the flow will be analyzed. For the S-duct, the geometry must accurately represent the duct's shape, which typically includes an inlet section, a curved section, and an outlet section.

**6.1. Defining the Duct Shape**

The S-duct geometry includes:

* Inlet Section (Straight): The fluid enters through a straight pipe, usually with a slight expansion.
* Curved Section: The bend in the duct is essential and can range from simple arcs to more complex non-circular cross-sections.
* Outlet Section (Straight): The fluid exits through another straight section, with a different diameter than the inlet, typically expanding or contracting.

The given data includes the inlet diameter (8.04 inches), outlet diameter (9.90 inches), and the lengths of the straight sections and the increasing diameter section. These parameters define the S-duct's overall shape.

**6.2. CAD Software and CFD Pre-processing Tools**

Various tools can be used to create the geometry:

* CAD Software (SolidWorks) allows for precise control of dimensions and complex features.
* CFD Pre-processing Tools (ANSYS DesignModeler) enable direct geometry creation, useful for duct-like structures.

For the S-duct, the geometry is created by modeling the inlet, outlet, and curved sections, ensuring smooth transitions between them.

**6.3. Key Considerations**

Important factors during geometry creation include:

* Smooth Transitions: Sharp transitions can lead to turbulence and performance loss.
* Curvature: The bend affects secondary flows, vortex formation, and pressure distribution.
* Symmetry: If applicable, symmetry can reduce computational cost by simulating only part of the geometry.

**6.4. Exporting to CFD Software**

Once the geometry is completed, it’s exported in formats such as STL, IGES, or STEP, and then imported into CFD software for meshing and simulation. This step ensures that the geometry is prepared for the analysis phase.

**7. Meshing Strategy in ANSYS Fluent**

Creating a high-quality mesh is crucial for accurate CFD simulations in ANSYS Fluent. The meshing process begins with geometry cleanup in **DesignModeler**, ensuring the model is free of unnecessary features and is ready for simulation. After importing the geometry into **ANSYS Meshing**, it’s essential to select the appropriate mesh type. For an S-duct, a combination of **hexahedral** and **tetrahedral elements** is often used, with a preference for **structured mesh** in the straight sections and **unstructured mesh** in the curved areas.

Next, mesh refinement is applied around critical regions like the inlet, outlet, and the bend, using **local refinement** to capture complex flow features accurately. **Inflation layers** are added near the walls to resolve the boundary layer. Setting the **growth rate** to 1.2 and ensuring a **maximum of 40 layers** helps maintain mesh quality while capturing flow gradients near surfaces.

After mesh generation, the quality is checked by examining skewness, aspect ratio, and cell count. Proper boundary conditions are set for the inlet, outlet, and walls, ensuring correct flow behavior. Finally, the solver is configured, and the solution is initialized. Post-processing tools help visualize results such as velocity profiles and pressure distribution, providing valuable insights into the duct’s aerodynamic performance.

**8. CFD Simulation Setup**

The CFD simulation setup for this study involves several key components, including geometry creation, mesh generation, turbulence modeling, boundary conditions, and solution methods. These elements are designed to simulate airflow through the S-duct accurately.

**8.1 Geometry Creation**

3D models of the S-duct are created with different cross-sections (circle-circle, square-square, circle-square, and square-circle) and bend angles ranging from 0° to 45°. These variations help analyze how changes in geometry affect flow behavior.

**8.2 Mesh Generation**

A structured mesh is created with fine resolution near the duct walls to capture boundary layer behavior accurately. The mesh quality, including y+ values, is carefully monitored to ensure proper representation of turbulence. Mesh independence studies are conducted to ensure that results are not influenced by the mesh size.

**8.3 Turbulence Model**

The **k-ω Shear Stress Transport (SST)** model is used for its accuracy in handling adverse pressure gradients and flow separation, which are common in S-ducts. It balances computational efficiency with the ability to resolve boundary layer behavior in curved regions.

**8.4 Boundary Conditions**

Uniform velocity is applied at the inlet with a Mach number of 0.6, and a pressure outlet condition is used. No-slip conditions are set for the walls, and symmetry may be applied in certain configurations.

**9. Results and Discussion: Single S-Duct Geometry**

The simulation results for a single S-duct geometry provide insights into the aerodynamic performance, highlighting key factors such as flow separation, pressure loss, and velocity distribution. The analysis focuses on the duct’s behavior at a flight Mach number of 0.6, with various cross-sectional shapes and bend angles to identify the optimal configuration.

**9.1 Flow Patterns and Separation**

The flow within the S-duct exhibits significant secondary flows due to the curvature, which leads to the formation of Dean vortices. These vortices cause non-uniform velocity profiles, especially near the bend regions. As expected, the curvature induces adverse pressure gradients along the inner walls of the duct, contributing to localized flow separation. This phenomenon is most pronounced at sharper bend angles, where the separation points shift further upstream, creating recirculation zones that impact the overall flow quality.

**9.2 Velocity Distribution and Flow Distortion**

The velocity profiles across the duct’s cross-section are non-uniform, with higher velocities near the outer walls and lower velocities near the inner bend. This distortion leads to non-uniform flow at the duct exit, which can affect downstream components like compressors. Sharp bends exacerbate this effect, leading to more pronounced distortion, which increases the risk of engine instability, such as surge or stall.

**9.3 Pressure Losses and Total Pressure Recovery**

Pressure losses increase with the severity of the bend, as the duct experiences higher frictional losses and pressure drop due to flow separation. The pressure recovery is lower for configurations with sharper bends and more significant separation zones. However, configurations with smoother transitions and less pronounced bends show improved pressure recovery, indicating better performance in terms of energy efficiency.

**9.4 Impact of Cross-Sectional Shape**

The choice of cross-sectional geometry (circle-circle, square-square, circle-square, square-circle) affects flow uniformity. Circular cross-sections generally maintain smoother flow patterns and reduced turbulence compared to square cross-sections, which tend to induce more turbulent eddies and increased drag.

In conclusion, the results demonstrate that smoother bends and circular cross-sections offer better aerodynamic performance, reducing flow distortion and pressure loss, which is crucial for optimizing S-duct design.

**10. Multi-Geometry Analysis (28 S-Duct Configurations)**

The multi-geometry analysis examines 28 different S-duct configurations, varying in cross-sectional shapes and bend angles to assess their aerodynamic performance. The configurations include combinations of circular and square cross-sections (circle-circle, square-square, circle-square, square-circle) with bend angles ranging from 15° to 45°, representing a range of typical S-duct designs in aerospace applications.

**10.1 Key Findings**

The analysis reveals that the cross-sectional shape plays a significant role in flow uniformity and pressure recovery. Circular cross-sections consistently show smoother flow with reduced turbulence, leading to lower flow distortion and better pressure recovery compared to square cross-sections. Square sections, on the other hand, cause more turbulence and increase drag, especially at higher bend angles.

**10.2 Bend Angle Impact**

As the bend angle increases, both flow separation and pressure losses become more pronounced. Sharp bends (closer to 45°) lead to higher turbulence and a larger region of flow separation, which negatively impacts the total pressure recovery. In contrast, gentler bends (15° to 30°) maintain smoother flow and reduce separation, improving overall duct efficiency.

**10.3 Optimal Configuration**

The most optimal configurations combine circular cross-sections with moderate bend angles (30°), offering the best balance of low flow distortion, high pressure recovery, and minimal turbulence. These configurations are ideal for applications requiring efficient airflow with minimal energy loss.

**11. Development of Web-Based Visualization Platform**

The development of a web-based visualization platform aims to provide an interactive tool for users to explore and analyze the aerodynamic performance of various S-duct configurations. The platform enables real-time visualization of key flow metrics, such as velocity contours, pressure distributions, and turbulence characteristics, allowing users to compare and contrast the performance of different duct geometries with ease.

**11.1 Features and Functionality**

The platform is designed with a user-friendly interface, featuring:

* **Geometry Selector**: Users can choose from a range of S-duct geometries, including different cross-sectional shapes and bend angles.
* **Real-Time Data Visualization**: The platform displays key aerodynamic parameters, such as velocity profiles, pressure recovery, and turbulence intensity, through dynamic charts and graphical representations.
* **Comparison Tools**: Users can compare multiple configurations side-by-side, highlighting differences in flow performance and identifying optimal designs.
* **Downloadable Reports**: The tool allows users to generate and download detailed reports of the simulation results, facilitating further analysis.

**11.2 Technical Implementation**

The platform is built using modern web technologies such as React.js for the front-end, ensuring a responsive and interactive user experience. Data visualization is handled through the **Recharts** library, which enables the display of complex aerodynamic metrics in an accessible format. The platform is designed to be compatible with both desktop and mobile devices, ensuring wide accessibility.

**11.3 Benefits**

The web-based platform streamlines the process of S-duct analysis, enabling quick decision-making and design optimization. By visualizing simulation results interactively, engineers and designers can rapidly assess the impact of different design choices, leading to faster development cycles and more efficient designs.

**12. AI-Based Prediction Model**

To accelerate the evaluation of S-duct performance without relying on time-consuming CFD simulations, an AI-based prediction model was developed. The model leverages machine learning techniques to predict key aerodynamic outputs—such as outlet velocity, outlet pressure, velocity loss, and pressure loss—based on input parameters like Mach number, bend angle, inlet conditions, and duct geometry.

The dataset for model training was generated from detailed CFD analyses of 28 different S-duct configurations. Each data entry included inlet conditions and geometric information alongside the corresponding aerodynamic outputs. One-Hot Encoding was used to represent categorical geometry types effectively.

An ensemble method, **XGBoost Regressor** wrapped inside a **MultiOutputRegressor**, was selected due to its robustness in handling tabular data and its superior predictive accuracy. The model was trained using an 80-20 train-test split and achieved high R² scores across all output variables, indicating excellent predictive capability.

Additionally, the trained model and encoder were serialized using joblib, allowing for quick deployment and reuse. A utility function was also developed to predict outcomes for new configurations instantly.

By integrating this AI-based predictor with the web platform, users can now obtain rapid estimates of aerodynamic performance, enabling faster design iteration without the need for repeated CFD simulations.

**13. Key Findings and Interpretation**

The aerodynamic analysis and AI-based predictions across 28 different S-duct configurations led to several important insights regarding optimal designs for different classes of aircraft:

* **Circle-Circle Geometry:**  
  The Circle-Circle S-duct emerged as the most suitable configuration for **commercial aircraft**. Although it demonstrated slightly lower pressure recovery compared to other geometries, it provided **highly uniform outlet flow**. This uniformity is critical for ensuring efficient and stable engine operation, resulting in **better overall efficiency and fuel economy** for long-duration flights typical of commercial aviation. Its simpler aerodynamic behavior also reduces risks of compressor stall, making it a reliable choice for airliners and business jets where operational stability and efficiency are prioritized.
* **Square-Circle Geometry:+**  
  The Square-Circle S-duct was found particularly advantageous for **fighter jets** and **high-performance military aircraft**. Despite offering lower pressure recovery and reduced fuel economy compared to commercial needs, this geometry produced **higher thrust levels** and enabled **superior shape integration** with the aircraft body. Additionally, its form factor helps significantly in **reducing the Radar Cross Section (RCS)**, thereby enhancing **stealth capabilities** — a vital feature for modern combat aircraft. Thus, for missions where maneuverability, thrust, and stealth outweigh fuel efficiency, the Square-Circle configuration is highly beneficial.

**14. Challenges Encountered**

During the course of this project, several technical and practical challenges were encountered, each offering valuable learning experiences:

* **Complexity of S-Duct Flow Simulation:**  
  Accurately capturing the highly three-dimensional and turbulent nature of flow within S-ducts was challenging. Ensuring correct modeling of separation, secondary flows, and pressure losses required fine mesh resolutions and careful selection of turbulence models, significantly increasing computational cost.
* **Mesh Generation and Quality Control:**  
  Generating high-quality structured meshes for multiple complex geometries proved time-consuming. Achieving appropriate boundary layer resolution (targeting optimal *y+* values) without excessively increasing cell count required iterative refinements.
* **High Computational Demand:**  
  Running steady-state CFD simulations for 28 different S-duct configurations at acceptable levels of accuracy demanded substantial computational resources and time. Managing simulation stability across all cases, particularly for aggressive bend angles, was difficult.
* **Data Preparation for AI Model Training:**  
  Preprocessing diverse aerodynamic datasets into a clean, machine-readable format involved challenges like encoding categorical geometry features, normalizing input parameters, and balancing data for different configurations to avoid bias during model training.
* **Model Generalization:**  
  Ensuring the AI model could generalize well across unseen geometries and conditions was a key hurdle. It required careful tuning of hyperparameters, robust validation, and ensuring sufficient variability in the training data.
* **Web Platform Integration:**  
  Developing an interactive, user-friendly platform to display CFD and AI results required efficient back-end data handling and front-end visualization skills, combining engineering analysis with software development.

Despite these challenges, each issue was systematically addressed, leading to a successful completion of the objectives and valuable insights into both aerodynamic optimization and AI-assisted engineering workflows.

**15. Conclusion**

This project successfully explored the aerodynamic behavior of S-ducts through a combined approach of Computational Fluid Dynamics (CFD) simulations and Artificial Intelligence (AI)-based predictive modeling. Multiple S-duct geometries were analyzed across various bend angles to understand their effects on flow uniformity, pressure recovery, and overall aerodynamic efficiency.

The CFD analysis revealed that **circle-circle S-ducts** offer the most uniform outlet flow, making them ideal for **commercial aircraft** where engine stability and fuel efficiency are prioritized, despite slightly lower pressure recovery. In contrast, the **square-circle S-duct** was found to be highly suitable for **fighter jets**, where enhanced thrust, better shape integration into the airframe, and reduced radar cross-section (RCS) are more critical than fuel economy.

Additionally, an AI-based model was developed and successfully trained to predict key aerodynamic parameters with high accuracy, reducing reliance on time-consuming CFD simulations. A user-friendly web-based visualization platform was also created to assist engineers in comparing and evaluating S-duct designs efficiently.

Overall, the study demonstrates that integrating CFD with AI tools offers a powerful and efficient methodology for optimizing complex aerospace components like S-ducts, paving the way for faster design cycles, lower development costs, and improved aerodynamic performance in future aircraft designs.

**References**

* <https://youtu.be/MJQVpvWBwXg?si=oycIkb3c4b41rUOZ>
* <https://youtu.be/1woKtFjN-bE?si=bQ1waJTgeUwXjkoi>
* <https://www.sciencedirect.com/science/article/abs/pii/S1270963816313517>
* <https://en.wikipedia.org/wiki/S-duct>
* <https://www.reddit.com/r/aviation/comments/eki2bk/ever_wondered_how_an_sduct_setup_looks_like_with/>
* <https://www.reddit.com/r/WarplanePorn/comments/xsonox/sshaped_duct_of_kf21_boramaealbum/>