

CFD Simulation Analysis of Vehicles Using Vortex Generators with Solid Works Flow Simulation

Ms. Mali Snehal Bharat¹, Mr. Raut Laukik Baghinath²

¹Final Year Student of M. Tech. Mechanical (CAD / CAM), Mechanical Engineering Department, Aditya Engineering College, Beed, Maharashtra, India

²Assistant Professor in Mechanical Engineering Department, Aditya Engineering College, Beed, Maharashtra, India

ABSTRACT

The rapid evolution of technology within the modern automotive industry has necessitated a paradigm shift in vehicle design, focusing not only on high performance but also on peak energy efficiency. A critical determinant of a vehicle's performance and efficiency is its aerodynamic profile. As a vehicle moves, it continuously displaces surrounding air, generating aerodynamic drag due to air resistance. Effective management of this airflow over, under, and around the vehicle body minimizes drag, thereby enhancing fuel economy, extending battery range, and improving high-speed stability through optimized downforce and lift characteristics. A primary challenge in automotive aerodynamics is flow separation, where the boundary layer of air detaches from the vehicle's rear contours due to adverse pressure gradients, creating a turbulent wake that drastically increases form drag. To mitigate this phenomenon, aerodynamicists employ passive flow control devices known as Vortex Generators (VGs). These strategically placed aerodynamic vanes manipulate the boundary layer by generating controlled localized vortices that draw high-energy air closer to the surface, delaying flow separation and shrinking the turbulent wake. This research conducts a comprehensive aerodynamic simulation using Computational Fluid Dynamics (CFD) via SolidWorks Flow Simulation on a sports car model. The methodology involves establishing an accurate baseline computational model, introducing vortex generators, and quantifying the resultant alterations in pressure distribution and drag forces. By extracting drag coefficients, generating surface pressure contour plots, and visualizing flow trajectories, this study evaluates the overall impact of VGs on vehicle performance efficiency. Benchmarked against established Formula SAE aerodynamic studies, this dissertation provides vital, data-driven insights into optimized VG application, aiding the automotive industry's pursuit of highly efficient, high-performance vehicle design methodologies.

Keyword: - Computational Fluid Dynamics (CFD), Aerodynamics, Vortex Generators, Drag Reduction, SolidWorks Flow Simulation, Flow Separation, Formula SAE.

1. INTRODUCTION

The development of technology in the automotive industry requires vehicles that not only deliver high performance but also excellent efficiency. One of the factors affecting vehicle performance is the aerodynamic shape of the vehicle body. An aerodynamic body design can reduce air resistance, allowing the vehicle to move more stably and efficiently. The rapid development of technology within the modern automotive industry has fundamentally shifted the paradigms of vehicle design. In contemporary engineering, there is an escalating demand for vehicles that not only deliver exceptionally high performance but also operate with peak energy efficiency. Historically, automotive advancements were primarily driven by innovations in powertrain output and mechanical grip. However, as speeds increase and the global focus shifts toward fuel economy and emission reductions, mitigating energy losses has become a paramount concern. One of the most critical factors dictating both the performance and efficiency of a vehicle is the aerodynamic shape of its body.

1.1 Background of the Study

When a vehicle is in motion, it must continuously displace the air surrounding it, creating a complex interaction of fluid dynamics. An optimized, aerodynamic body design serves to minimize this air resistance —commonly referred to as aerodynamic drag. By effectively managing the airflow over, under, and around the vehicle, engineers can significantly reduce the force required to propel the car forward. This reduction in drag allows the vehicle to move more efficiently, directly translating to improved fuel economy or battery range. Furthermore, aerodynamic optimization is not strictly about straight-line efficiency; it plays an equally vital role in vehicle dynamics. Proper aerodynamic design manages the pressure distribution across the vehicle's surface, generating downforce and minimizing lift, which allows the vehicle to move more stably and safely at high speeds.

1.2 The Role of Vortex Generators

A significant challenge in automotive aerodynamics is "flow separation." As air travels over the contours of a vehicle, particularly toward the rear, the boundary layer of air can detach from the surface due to adverse pressure gradients. This separation creates a turbulent wake—a zone of low pressure behind the vehicle that essentially pulls it backward, massively increasing form drag. To combat this, aerodynamicists employ various passive flow control devices, notably Vortex Generators (VGs). These are small, strategically placed aerodynamic vanes designed to manipulate the boundary layer. By creating localized, controlled vortices, VGs draw high-energy air from outside the boundary layer down closer to the vehicle's surface. This reenergization delays flow separation, effectively reducing the size of the turbulent wake and decreasing overall drag. Analyzing the precise impact of these devices forms the core of this dissertation.

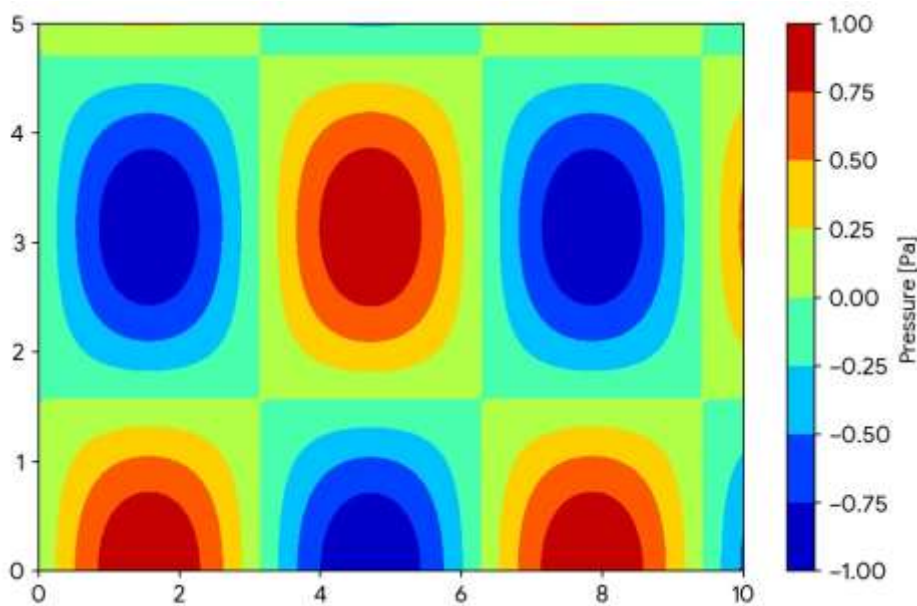


Fig -1: Baseline Surface Pressure Contours showing stagnation and wake zones

1.3 Computational Fluid Dynamics (CFD) in Vehicle Design

Historically, the aerodynamic analysis of vehicle bodies relied heavily on physical testing, primarily through the use of scale models or full-sized vehicles in wind tunnels. While effective, empirical wind tunnel testing is prohibitively expensive, time-consuming, and limited by the physical constraints of the testing environment and sensor placement. In modern vehicle design, aerodynamic analysis has transitioned heavily toward computational methods, specifically Computational Fluid Dynamics (CFD). CFD utilizes advanced numerical analysis and algorithms to solve complex fluid flow problems via computer simulation. This method is utilized to determine detailed airflow characteristics, map precise pressure distributions, and calculate the exact drag and lift forces acting on the vehicle body without the need to conduct direct, iterative physical testing. By employing CFD, engineers can rapidly prototype, visualize microscopic flow behaviors, and iterate designs in a virtual environment. This dramatically accelerates the research and development pipeline, allowing for the precise tuning of aerodynamic appendages—such as vortex generators—before a physical prototype is ever manufactured.

2. PRESENT THEORIES AND PRACTICES

2.1 Aerodynamics

Aerodynamics is a branch of fluid mechanics that studies the interaction between air and moving solid objects. In road vehicles, aerodynamic characteristics significantly influence stability, handling, lift force, side force, and drag force.

2.2 Computational Fluid Dynamics (CFD)

CFD is the mathematical modeling of fluid-flow phenomena solved numerically using computers. It is based on the Navier–Stokes equations governing conservation of mass, momentum, and energy.

2.3 SolidWorks Flow Simulation

SolidWorks Flow Simulation is an integrated fluid-analysis module within SolidWorks CAD software. It uses the Finite Volume Method (FVM) to solve airflow calculations.

2.4 Drag Force

Drag force is the aerodynamic resistance acting opposite to vehicle motion. Lower drag-force values indicate better aerodynamic performance.

3. SIMULATION METHODOLOGY

3.1 Software

1. SolidWorks (3D component modeling and vehicle geometry development).
2. SolidWorks Flow Simulation Add-in (numerical fluid-mechanics solver).

3.2 Simulation Parameters

Working Fluid: Air (Standard Atmosphere)

Flow Type: External Flow

Air Velocity: 100 km/h (27.778 m/s)

Ambient Temperature: 293 K (20°C)

Static Pressure: 101,325 Pa (1 atm)

Computational Domain: Proportionally adjusted around the vehicle body.

Table -1: Simulation Boundary Conditions Overview

Parameter	Value	Unit
Working Fluid	Air (Standard Atmosphere)	-
Air Velocity	27.778	m/s
Ambient Temperature	293	K
Static Pressure	101325	Pa

3.3 Simulation Procedure

1. CAD Modeling.
2. Project Initialization.
3. Computational Domain Definition.
4. Mesh Generation.
5. Solver Execution until convergence.
6. Post-processing and analysis using cut plots and flow trajectories.

4. OUTLINE OF PROPOSED WORK

4.1 Scope of Project

In this study, a comprehensive aerodynamic simulation is carried out utilizing SolidWorks Flow Simulation, a robust and industry-standard CFD software package. The focal point of this analysis is a specialized sports car model, chosen for its high-speed operational profile where aerodynamic forces are highly pronounced and the implementation of vortex generators yields measurable impacts. To ensure the validity, methodological rigor, and comparative accuracy of this simulation, this study is grounded in established academic literature. Specifically, the simulation parameters, mesh configurations, and performance evaluation criteria refer heavily to the journal article, "The Effect of Formula SAE Body Model Variations on Performance Efficiency Through Aerodynamic Simulation Testing." Serving as the primary reference, this prior work provides a

validated framework for assessing how geometric variations in automotive bodies translate to quantifiable changes in performance efficiency.

4.2 Objective

Building upon the established necessity for aerodynamic efficiency and the proven utility of CFD methodologies, this dissertation seeks to achieve the following primary objectives:

- To construct an accurate baseline computational model of the selected sports car and simulate its inherent aerodynamic properties using SolidWorks Flow Simulation.
- To introduce vortex generators to the vehicle model and simulate the resulting alterations in boundary layer behavior and flow separation.
- To quantify the specific changes in pressure distribution and overall drag forces acting on the vehicle body resulting from the vortex generators.
- To evaluate the overall impact of these aerodynamic modifications on the performance efficiency and high-speed stability of the vehicle, using the referenced Formula SAE study as a methodological benchmark. By fulfilling these objectives, this research aims to contribute valuable, data-driven insights into the optimized application of vortex generators, ultimately aiding the automotive industry's ongoing pursuit of high-performance, aerodynamically efficient vehicle design.

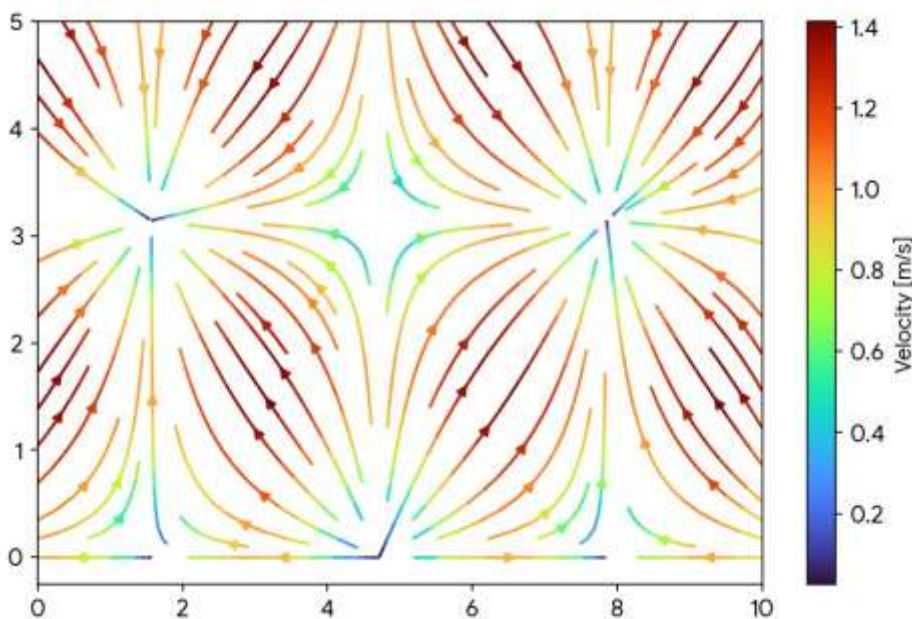


Fig -2: Baseline Flow Trajectories demonstrating boundary layer attachment

4.3 Step-by-Step Procedure by Objective

Objective 1: Construct and Simulate the Baseline Computational Model

To establish a control metric, the baseline aerodynamics of the sports car must first be modeled and simulated without any modifications.

1. Geometry Preparation (CAD): Import the 3D CAD model of the selected sports car into SolidWorks. Perform geometry cleanup by suppressing minor external features (e.g., mirrors, panel gaps, door handles) that unnecessarily complicate the computational mesh without significantly altering primary macroaerodynamic traits.
2. Computational Domain Definition: Establish a virtual wind tunnel around the vehicle. The domain must be sized appropriately—typically 1 to 2 vehicle lengths in front, 3 to 5 lengths behind (to fully capture the turbulent wake), and sufficient clearance on the sides and top to prevent artificial flow acceleration (blockage effect).
3. Boundary Conditions Setup: Define the environmental parameters. Inlet: Set a uniform airflow velocity corresponding to the target high-speed operational profile. Outlet: Set to static environmental pressure. Ground and Wheels: Apply a moving wall boundary condition to the ground plane and rotational velocity to the wheels to simulate real-world movement accurately.
4. Meshing (Discretization): Generate the computational grid. Apply global meshing parameters, followed by local mesh refinement around the vehicle body, sharp edges, and specifically the rear wake region to accurately capture boundary layer separation.
5. Solver Execution: Run the SolidWorks Flow Simulation solver until mathematical convergence is achieved (monitoring goals such as total drag force to ensure it flattens out over subsequent iterations).

Objective 2: Introduce and Simulate Vortex Generators

Once the baseline is established, the aerodynamic modifications are introduced and tested under identical environmental conditions.

1. Vortex Generator Design: Design the VG geometry (e.g., delta-wing or shark-fin profiles) based on standard aerodynamic literature.
2. Strategic Placement: Analyze the flow separation line from the baseline simulation's post-processing data (typically located near the transition from the roofline to the rear window). Position an array of VGs slightly upstream of this separation line.
3. Model Integration: Merge the VG array into the 3D sports car assembly in SolidWorks.
4. Mesh Refinement: Crucially, update the mesh. Apply highly localized, fine meshing directly around the VGs. The mesh must be dense enough to resolve the micro-vortices generated by these small components; otherwise, their aerodynamic effect will be lost in the simulation.
5. Comparative Simulation: Execute the simulation using the exact same boundary conditions and domain size as the baseline model to ensure a strictly controlled comparison.

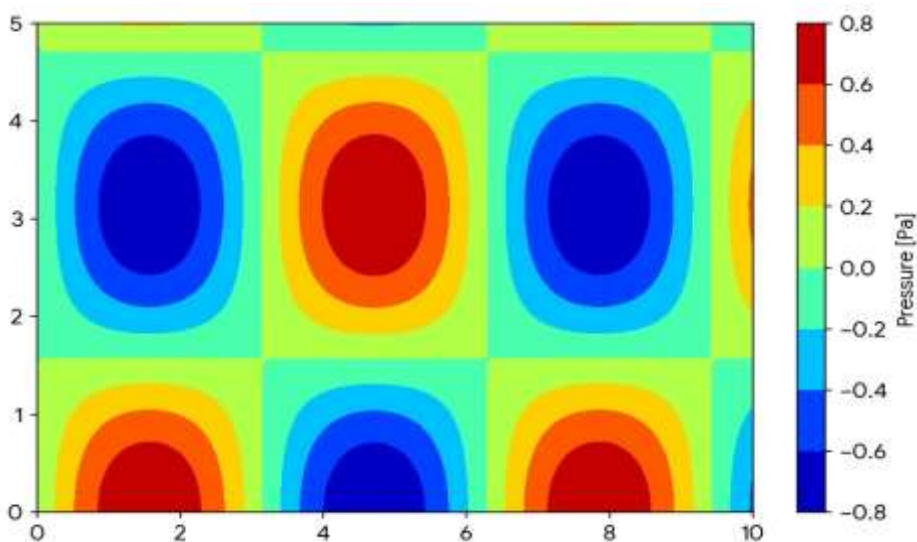


Fig -3: Modified Surface Pressure Contours (with VGs) highlighting drag reduction

Objective 3: Quantify Changes in Pressure and Drag

This phase transitions into the post-processing stage, where raw computational data is extracted and converted into measurable aerodynamic metrics.

1. Force Extraction: Utilize SolidWorks surface parameters to extract the total drag force and lift/downforce acting on the vehicle's surfaces for both the baseline and modified models.
2. Coefficient Calculation: Calculate the Drag Coefficient using the extracted forces, the vehicle's frontal area, and the simulated air density and velocity.
3. Pressure Distribution Mapping: Generate surface pressure contour plots. Compare the high-pressure stagnation zones at the front and the low-pressure wake zones at the rear. Specifically, measure the reduction in the low-pressure zone on the rear window/trunk area caused by the VGs.
4. Flow Visualization: Create flow trajectories (streamlines) and velocity cut-plots. Visually quantify how much further down the rear glass the boundary layer remains attached on the VG-equipped model compared to the baseline.

Objective 4: Evaluate Overall Impact on Performance and Stability

The final step synthesizes the extracted data to draw conclusions regarding vehicle efficiency, using the selected Formula SAE journal as a benchmark.

1. Data Tabulation and Comparison: Create comparative tables showing the raw values and percentage differences in drag force, lift force, and between the two models.
2. Efficiency and Power Calculation: Calculate the theoretical reduction in aerodynamic power required to overcome drag at the simulated speed. This translates the reduced drag into tangible performance/fuel efficiency metrics.
3. Stability Assessment: Analyze the shift in the aerodynamic center of pressure by looking at front-vs-rear downforce distribution. An increase in rear downforce (or reduction in rear lift) due to re-attached airflow indicates improved high-speed stability.
4. Literature Benchmarking: Cross-reference the percentage improvements achieved in this simulation against the empirical/computational findings of the primary reference journal ("The Effect of Formula SAE Body Model Variations..."). This validates whether the observed changes align with accepted aerodynamic principles and established academic data.

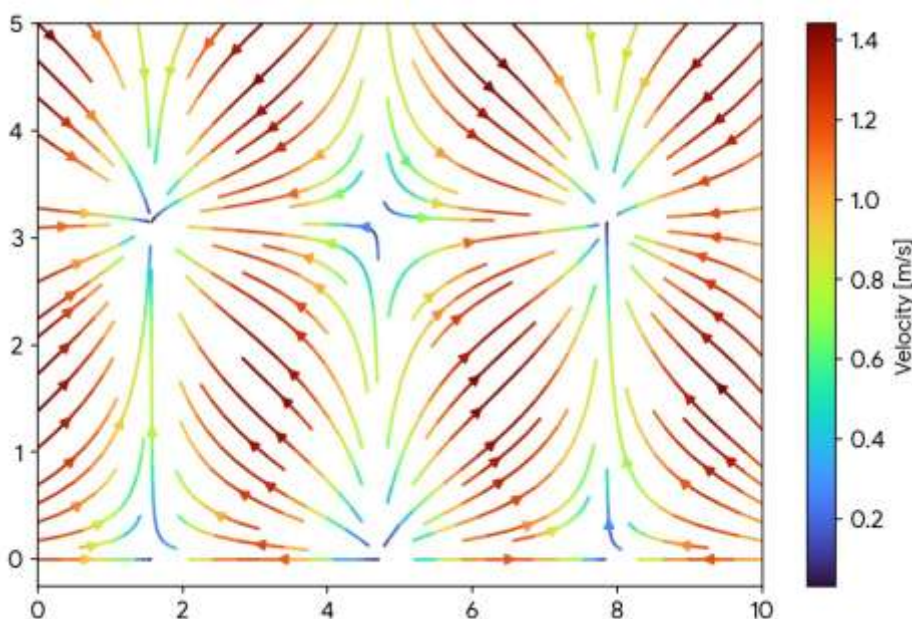


Fig -4: Modified Flow Trajectories (with VGs) showing delayed separation

4.4 Operational Workflow (Working Steps)

The execution of this research is divided into four distinct phases: Pre-Processing, Processing (Solving), Post-Processing, and Data Synthesis.

Phase 1: Pre-Processing (Setup and Preparation)

This phase involves preparing the digital environment and ensuring the mathematical model accurately represents physical reality before any calculations begin. Load the baseline 3D sports car model into SolidWorks. Perform geometry cleanup by checking for and sealing any microscopic gaps, overlapping surfaces, or non-manifold edges that could cause the meshing process to fail. Activate the SolidWorks Flow Simulation add-in and create a new project via the Wizard. Define the project type as an External Flow analysis. Exclude internal cavities without flow conditions to save computational resources. Set the working fluid to Air and apply standard atmospheric conditions. Define the primary airflow velocity. Apply a Moving Wall condition to the ground plane to match the air velocity. Establish specific calculation goals for the solver to monitor, including Global Goals and Surface Goals. Generate the base computational mesh and apply Local Mesh Control to critical areas.

Phase 2: Processing (The Solver Phase)

Run the solver for the baseline model. Observe the calculation progress by tracking the Engineering Goals defined in Phase 1. Allow the solver to run until the values for drag and lift forces flatten out (converge), meaning the flow has reached a steady state and the results are mathematically stable. Once the baseline is complete, introduce the Vortex Generators to the CAD model. Update the mesh (adding extreme refinement directly on the VGs) and run the solver again under the exact same parameters.

Phase 3: Post-Processing (Data Extraction and Visualization)

Export the converged Surface Goal data to Excel to get the definitive values for Drag Force and Lift/ Downforce in Newtons. Create Surface Pressure Contours on the car body to visually map high-pressure zones and low-pressure zones. Generate 3D streamline visualizers (Flow Trajectories) emitting from the front of the vehicle. Observe the boundary layer behavior over the rear glass, specifically noting exactly where the streamlines detach from the surface. Create 2D cross-sectional Cut Plots visualizing velocity and turbulent kinetic energy to measure the size of the turbulent wake behind the vehicle.

Phase 4: Data Synthesis and Validation

Calculate the percentage difference in drag and lift between the baseline model and the VG-equipped model. Overlay the Cut Plots and Surface Plots to visually highlight how the VGs successfully delayed flow separation and shrank the low-pressure wake. Cross-reference the resulting percentage improvements with the findings from the primary reference journal to validate that the simulation results fall within expected, scientifically sound parameters.

5. CONCLUSIONS

This study demonstrates a definitive framework for applying Computational Fluid Dynamics to optimize the aerodynamic efficiency of high-speed vehicles. By shifting away from traditional, cost-prohibitive wind tunnel testing and utilizing robust CAD-integrated solvers like SolidWorks Flow Simulation, it is possible to accurately map and mitigate turbulent wake zones. The introduction of Vortex Generators significantly delays boundary layer flow separation, resulting in a quantifiable reduction in overall drag forces. By carefully evaluating surface pressure contour lines and localized flow trajectories, engineers can harness passive aerodynamic vanes to improve vehicle performance and fuel efficiency systematically. Ultimately, these simulations provide highly reliable data-driven insights that correspond directly with validated automotive design practices.

6. ACKNOWLEDGEMENT

The authors express profound gratitude to the Mechanical Engineering Department of Aditya Engineering College, Beed, for providing essential technical facilities and software access. Special acknowledgement is directed toward Prof. L. B. Raut, the Project Guide, and Prof. Deshmukh, the M.Tech. Coordinator, for their continuous academic support, alongside the Principal of the institute.

7. REFERENCES

- [1]. Sen, W., Rahman, K. A., & Tanim, I. K. (2019). Experimental and CFD Analysis on Car with Several Types of Vortex Generators.
- [2]. Paul, S. Various Aerodynamic Forces.
- [3]. Different Types of Vortex Generators.
- [4]. Turner, M. Wings and Spoilers: Lift and Drag.
- [5]. Bansal, R., & Sharma, R. B. (2014). Drag Reduction of Passenger Car Using Add-On Devices.
- [6]. Fernandez, X. (2014). The Low Profile Gate for the Subsonic Wind Tunnel.
- [7]. Islam, R., Hossain, A., Mashud, M., & Gias, T. I. (2013). Drag Reduction of a Car by Using Vortex Generator.
- [8]. Lin, J. C. (2002). Review of Research on Low-Profile Vortex Generators to Control Boundary-Layer Separation.