Analyzing the Impact of Venturi-Induced Wingtip Suction on the Induced Drag and Wingtip Vortex

Clayton Spaulding[1]
Mississippi State University, Mississippi State, Mississippi, 39762, United States

The goal of this report is to analyze and test a patented wingtip vortex reduction method from the 1930's. This report will review and explain the methods of wingtip vortex analysis, as well as the physics behind the reduction of vortices. Additionally, this paper will highlight the method used to test the design, computational fluid dynamics. Finally, by exploring the drag and vorticity difference between the two wings, this paper will explore whether or not the patented design is feasible for real-world use.

Nomenclature

A = *Area*
ρ = Density
μ = Dynamic viscosity
L = Length
$\dot{m}$ = Mass flow rate
N = Newton
R = Reynold's number
V = Velocity

I. Introduction

SINCE the day manned flight was introduced into the world, man has strived to improve it. From the development of laminar flow wings, to the addition of vortex generators on the skin of a wing, with every decade there seems to be a new breakthrough in flight efficiency. In this current age, cost and efficiency play a vital role in the production and operation of aircraft. Looking at the operating costs of any airline or private flight company will quickly bring to light that a large portion of operating costs comes from fuel consumption alone. Due to this fact, aircraft designers and operators aim to improve efficiency and fuel economy at nearly every turn within the design process.

In 1935, Arthur Loerke published a patent for a wingtip vortex-reduction device which could be mounted into a wing to reduce drag [1]. The design incorporates a Venturi tunnel mounted inside of a wing section, with its mouth facing into the freestream flow as seen in Figure 1. As the flow moves through the Venturi's throat, its velocity increases, and a low pressure region forms. A tube then runs span-wise from this region to the tip of the wing through a small tube. In flight conditions this setup would, in theory, create a suction at the wingtip thus reducing the induced wingtip vortex. Since the patent's publishing, there have been no apparent attempts to reproduce or test this design. Therefore, the question of its effectiveness remains unanswered.

---

[1] Undergraduate Student, Aerospace Engineering, Student Member

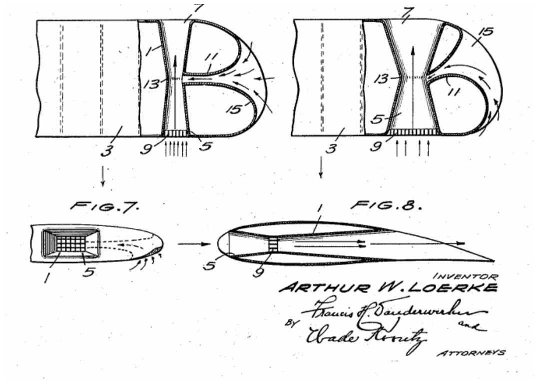


Figure 1: Patented wingtip venturi design

II. Wingtip Vortex Theory

A. Vortex Overview

There are different forms of drag which act on an aircraft in flight, however, one of the key forms is induced drag. Induced drag is drag which is created due to the pressure differential between the upper and lower surfaces of a wing. As a wing produces lift, there is a higher pressure under the wing than there is on top of the wing. This gradient encourages flow to "leak" around the wingtip from bottom to top, causing a rotating motion. The rotating air is referred to as a vortex and makes up a noticeable portion of energy loss in the free stream air as the aircraft moves forward in flight. There have been significant advances in the field of wingtip vortex reduction in the past few decades, including modified-tip wing designs; however, to this day wingtip vortex-induced drag remains an issue.

B. Vortex Reduction Methods

As aforementioned, in the past few decades there have been various attempts to mitigate vortex drag on aircraft. One of the most used methods is modified wingtips. Since vortices form from air moving around the tip of the wing, it would be logical to assume that by modifying the shape of the tip body, the air movement could be interrupted. A widely used tip design is the upward-pointed tip. This design effectively increases the distance the air must move from the lower surface of the wing to the upper surface. By increasing the distance the air has to travel, the design helps to mitigate the mixing of high pressure and low pressure air: thus decreasing the strength of the tip vortices. Other variations of this tip design exist as well, however the theory behind their function remains the same.

Active suction is another form of vortex reduction. The active sucking method works by incorporating a low pressure at a specific point/s on the wing, in order to disrupt the movement of air. For instance, if active sucking is applied at the wingtip, the air moves from underneath the wing, up into the suction environment. This, in turn, reduces the quantity of high pressure air that interfaces with the low pressure air on top. The same concept applies to active blowing, however with an incorporated high pressure environment.

Methods to induce such a suction have varied over the years, however, a commonly used method is motor induced-suction. This method utilizes an internal electric motor implanted in the body of the aircraft, which creates a low pressure environment within a cavity. The cavity is then linked, via internal plumbing, to the point of desired effect where low pressure is needed [3].

C. The Experimental Case

The patented design shown in Figure 1 uses a tip suction method similar to the method explained above. The difference, however, is the source of suction. Rather than relying on an electric motor to create a favorable pressure gradient, the design uses a venturi tunnel to create a low pressure. Since the tunnel faces the leading edge of the wing, the movement of the aircraft creates a resulting freestream air which flows through the venturi tunnel. This flow in turn accelerates as it moves through the tunnel, creating low pressure at the throat. The low pressure region is then linked to the wingtip where the circulating tip flow is reduced. There are many benefits to such a design. First, a venturi tunnel cannot stop running as long as there is air flowing through it. This is an important note to make, considering an electric motor can fail or become over heated in flight. An additional advantage of a venturi-induced suction is the progressively increasing suction achieved by steadily accelerated flight. This aspect takes away the necessity of a pilot or control system to operate or adjust the suction.

Although the patented design holds numerous advantages over other vortex reduction solutions, there are disadvantages as well. Perhaps the biggest disadvantage is the induced drag of the venturi tunnel itself. While the baseline wing has only a smooth curved surface for air to flow over, the test surface with a venturi opening provides a round surface for a pressure increase to occur. This increase in pressure on the leading edge of the wing provides a directly proportional increase in drag, particularly for non-cambered foils at zero angle of attack. Since the venturi itself creates drag, the problem then becomes whether the system can reduce vortex drag enough to overcome the induced drag of the system's integration. An additional flaw of the venturi is its lack of flexibility in regard to angle of attack. For instance, if the venturi is installed in a non-cambered airfoil so that its opening faces the leading edge of the wing at zero angle of attack, it will not perform as effectively when the pilot increases or decreases the angle of attack. This is due to the fact that the venturi is succumbing to an induced angle of attack at which it was not designed to operate, and therefore will not operate at its maximum performance.

III. Model Creation

To begin the experiment, a 3d CAD model was created to use within a CFD environment. A sketch of a **NACA** 2412 airfoil was extruded and sized to 2.5 feet in span and 1.5 feet in chord. From there a venturi tunnel was cut through the wing, from leading edge to trailing edge as seen in Figure 3. The size of the Venturi was made such that the mouth was as big as the wing structure would allow. The exact Venturi sizing was not as important in this stage, considering the purpose of the model is to understand its general flow characteristics. Ducting was then cut spanwise through the wing from the throat of the venturi to the wingtip as seen in Figure 4. In order to create a distributed low pressure zone on the wingtip, the ducting was separated into a series of small holes lining the tip of the wing. Additionally, in order to shape the flow of the air traveling around the tip of the wing, the wingtip was shaped with a crease along the midline of the tip as seen in Figure 2, traveling from leading edge to trailing edge. The ducted holes were then placed slightly under this crease. The positioning of the suction holes was established to capture the maximum amount of air traveling from the under-side of the wing.

Figure 2: Leading edge of modeled foil, with venturi opening and creased tip

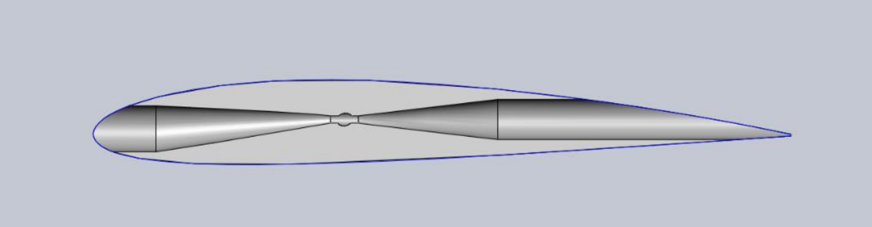


Figure 3:Modeled leading to trailing edge venturi tunnel

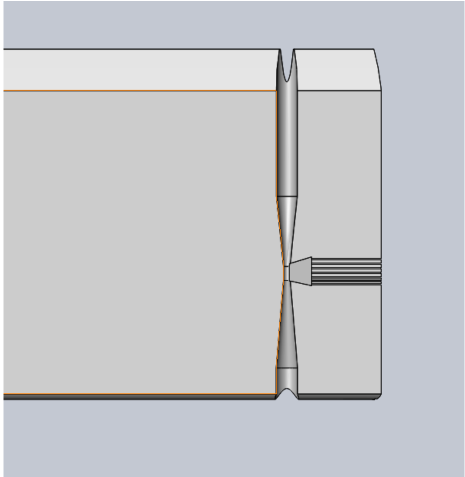


Figure 4:Top-level view of venturi and wingtip plumbing

IV. Results

Computational fluid dynamics (CFD) was the chosen method of analysis for this project. The CFD software used was SolidWorks [2]. Although CFD data does not always contain a high degree of accuracy, it gives the experimenter a good idea of the functionality of a flow problem. To accurately break down the flow characteristics of the wing, the CFD analysis was divided into three parts. The first part was a full-wing drag analysis which only measured the pressure drag experienced by each wing, excluding any induced suction. The second part was a wingtip analysis, which isolated the wingtip of the experimental and control wing as seen in Figure 5, incorporating suction at the tip of the experimental wing. The final part was a vorticity analysis which analyzed the vorticity distribution along the chord of each wingtip.

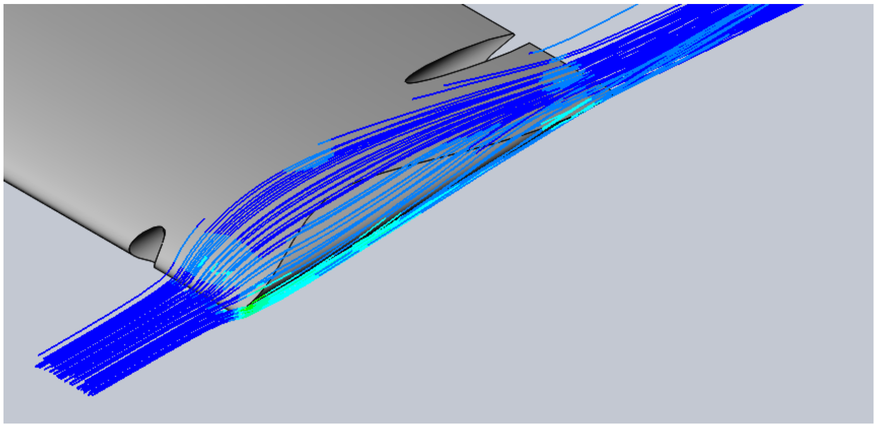


Figure 5: Wingtip flow analysis

Before any CFD tests took place, the experimental environment was established. The velocity at which the test took place was calculated using the Reynold number equation. To avoid turbulent flow transition within the simulation, a turbulent Reynold's number of 1,000,000 was chosen. Once the Reynold's number was chosen, Equation 1 was solved for velocity to determine the test speed. The calculated velocity for this experiment was 25 m/s. All temperature, pressure, and density values were set for standard sea level conditions.

$$R = \frac{\rho V L}{\mu} \tag{1}$$

A. Full Wing Drag

The first test was a full wing drag analysis in order to determine the drag effects of the Venturi tunnel imbedded in the leading edge. Because this test was focused on determining the drag produced by the tunnel alone, this test did not include wingtip suction.

Table 1:Control, clean wing drag

Clean Wing	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Normal Force (X)	[N]	4.49	4.63	4.47	4.89

Table 2:Experimental, embedded-venturi wing drag

With Venturi	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Normal Force (X)	[N]	7.00	6.98	6.90	7.12

Table 1 above displays the drag values experienced by the clean wing, while Table 2 displays the values for the Venturi wing. The data shows that the wing with the embedded Venturi tunnel demonstrates a 56% drag increase from the clean condition. This is to be expected considering the implications of the altered wing design. A Venturi tunnel operates by creating a high pressure upstream of its throat so that the throat may experience a fast, low pressure flow. In the case of the experimental wing, this upstream high pressure results in a drag buildup on the leading edge. Additionally, because a portion of the wing's trailing edge is eliminated from the design, the flow will likely tend to separate as it travels over and under the wing, creating further induced drag. Because of the drag which is induced by the Venturi tunnel, the data from the full wing drag analysis favors the clean wing design over the experimental design.

B. Wingtip Drag

After the general wing drag was established for the experimental and clean conditions, the wingtip drag was analyzed. Since this test would isolate a small portion of the wing, tip suction was applied on the experimental wing. One challenge which arose in working within the SolidWork's environment was the method of inducing suction. When creating a suction boundary condition within the environment, the accuracy of the simulation tended to vary substantially. To circumvent this issue, a mass flow rate was instead established at the wingtip. Through a series of four flow simulations, four different mass flow rates relating to different degrees of suction were induced at the wingtip. The mass flow rate was calculated using Equation 2 below. Velocity was found by using the desired pressure differential in conjunction with the Bernoulli equation. The four pressures used in the test were 3, 3.5, 4, and 4.5 psi. Analysis did not go further than 4.5 psi due to SolidWork's inherent inaccuracies when approaching transonic flow through small ducts.

$$\dot{m} = \rho V A \tag{2}$$

Table 3: No suction, clean wingtip drag

No suction	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Normal Force (X)	[N]	0.32	0.33	0.32	0.33

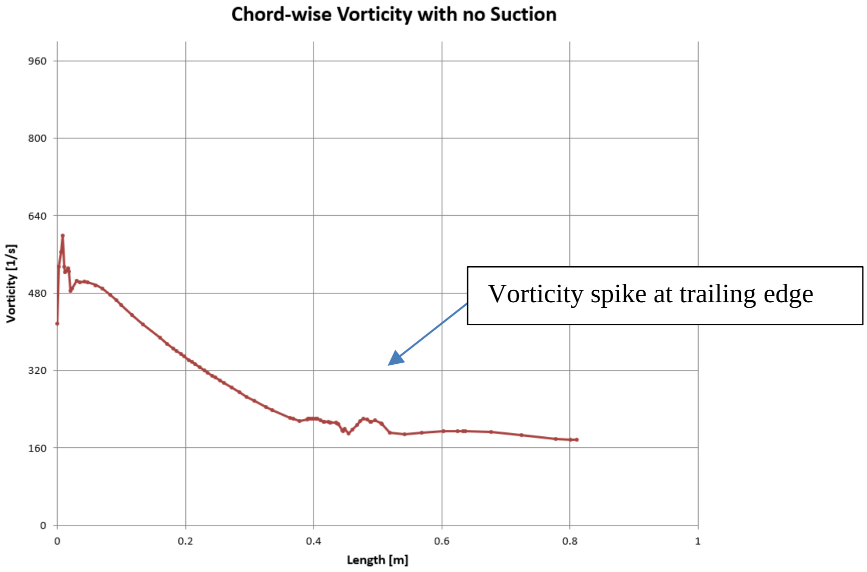
Table 4: Suction, experimental wingtip drag

3 psi	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Normal Force (X)	[N]	0.29	0.29	0.29	0.30
4.5 psi	Unit	Value	Averaged Value	Minimum Value	Maximum Value
Normal Force (X)	[N]	0.28	0.28	0.28	0.29

Table 3 above shows the drag values experienced by the clean wingtip with no suction applied, while Table 4 displays the drag values of the experimental wingtip with 3 and 4.5 psi of suction applied. All suction cases demonstrated reduced drag compared to the clean condition. Although the drag difference does not approach the scale of the differential seen by the venturi tunnel comparison, the 4.5 psi scenario demonstrates an appreciable 12.5% decrease in drag from .32 N to .28 N. When considering the definition of induced drag, it makes sense that the suction case would prove to have less drag. Induced drag is the result of lift; when a wing creates lift there tends to be leakage from the underside of the wing which travels to the top around the tip. This movement creates vortices which remove energy from the flow, creating drag. Therefore, by inducing a suction which captures a fraction of this leakage, the wing has a smaller energy loss and thus less drag.

B. Chord-Wise Vorticity

The final test was a chord-wise vorticity analysis. Vorticity is a measure of a fluid's local movement, which corresponds to a flow's energy output/loss. By finding the vorticity distribution along the chord of both the clean and experimental wings, the efficiency of the wing could be determined. To find the vorticity, a reference plane was established in the SolidWork's environment which sat along the chord of the wingtip. Once the desired flow simulation was completed, a graph of the vorticity distribution along the reference plane was created.



Vorticity spike at trailing edge

Figure 6:Vorticity distribution, no suction

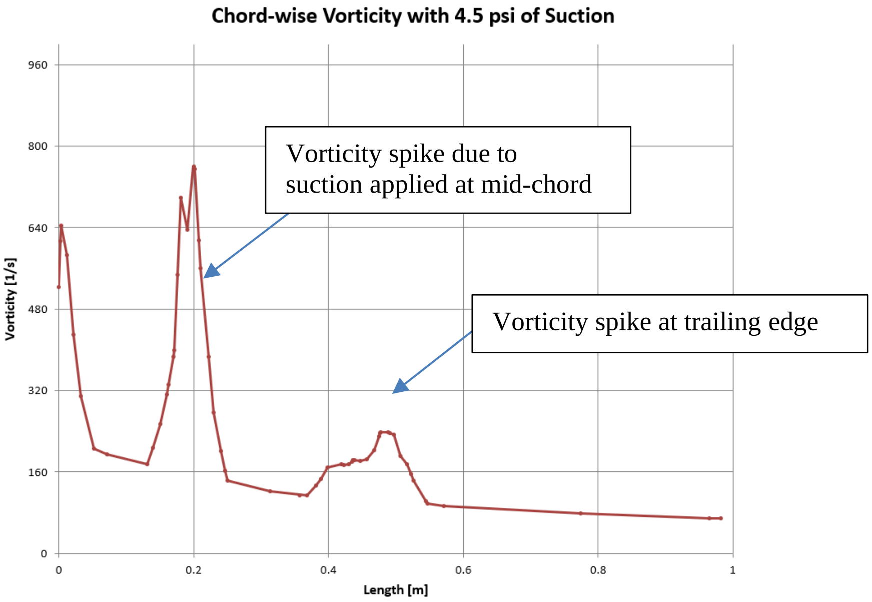


Figure 7:Vorticity distribution, 4.5 psi of suction

Figure 6 displays the vorticity distribution along the tip of the clean wing, while Figure 7 shows the distribution for the experimental tip with 4.5 psi of suction applied. Both wings display a vorticity peak at the leading edge, largely due to inconsistencies in the SolidWorks model itself. However, shortly after the initial peak, the vorticity changes rapidly between the two wings. The clean wing's vorticity appears to decrease smoothly along the chord, while the experimental wing's vorticity drops sharply and then rise suddenly to a peak. The succeeding increase is in direct response to the applied suction. It appears as though the suction calms the flow from the leading edge and then proceeds to mix it up again. This result should, however, be taken with restraint. Considering the complexity of this flow scenario, it is possible that this large spike in vorticity relates to SolidWork's inability to process the entire range of the location's calculation. Following the spike, the vorticity drops sharply again only to peak slightly as the flow leaves the trailing edge. Additionally, it is important to note that as the flow leaves the trailing edge, the experimental wing demonstrates a lower steady state vorticity value than that of the clean wing. This signifies that indeed the experimental wing has the potential to decrease the magnitude of the vortex production at the wingtip.

C. Future Testing Efforts

For this experiment, all tests were done through a CFD simulated environment. Although CFD is a useful method for understanding flow problems and obtaining initial data, it is generally not a good method for finalizing solid concepts. CFD is a simulation, and its data should therefore be used in conjunction with further research. Because a functional wind tunnel facility was not a viable option at the time of this experiment, CFD was the chosen medium for research. In exploring this topic further, a wind tunnel test would be the next test method. A wind tunnel is a great tool for exploring two specific aspects of this experiment: vortex size and drag. Determining the size of the vortices at the wingtip would allow the visualization of how the experimental case's efficiency differs from the baseline. One method for finding vortex strength is a hot wire anemometer test. This would involve using a hot wire anemometer to take point readings behind the tip of both wings to determine relative energy loss within the trailing flow. On the other hand, determining the drag of the wings would simply require a strain guage. As the wing endures the wind tunnel's flow, the strain guage would record the drag force experienced by the wing. This data could then be compared with the CFD force data to guage simulation accuracy.

V. Conclusions

The purpose of this experiment was to reexamine a design made in the 1930's, which suggested that the Venturi effect in conjunction with wingtip suction could be utilized to reduce drag on a wing. The method chosen to analyze this problem was CFD. Using the CFD environment found within SolidWorks, a series of three tests were run to compare flow conditions of a clean wing, to that of an experimental Venturi wing. The results reflected that the experimental wing's aerodynamics differ significantly for the clean wing.

The suction at the experimental wing's tip, caused by the venturi's flow, proved to indeed impact the drag experienced by the wing. Additionally, the suction had a positive impact on the vorticity distribution along the wing's chord. While these positive aspects of the experimental wing do highlight the increased efficiency of the design, they are overshadowed by the drag experienced at the Venturi tunnel. The Venturi tunnel also incorporates other negative aspects into the experimental wing design, such as structural and packaging issues. A tunnel incorporated into a wing will effectively eliminate wing structure in that area, as well as any internal space which could otherwise be used to house aileron/flap controls, wires, servos, etc. Finally, the suction is limited by the size of the Venturi tunnel. Bernoulli's equation states that the pressure present at the throat of a Venturi is directly related to the size of the throat and mouth of the tunnel. Therefore, since the cross sectional area of the Venturi throat and mouth are limited by the thickness of the airfoil section, the suction is also limited by the same factor.

In the end, analysis of Loerke's patent highlights that wingtip suction is an effective means to creating a more efficient wing. However, it also highlights that the method of creating such a suction is equally as important. Methods which might provide the same suction performance, without the included pressure drag, might include internal pump-driven suction, or similar methods.

VI. References

[1] Arthur W. Loerke., Dayton, Ohio, U.S. Patent Application for a "Wing Vortex Reducer," Docket No. Cl-244-130, filed 17 Aug. 1934.

[2] SolidWorks, Software Package, Ver. 2019, Dassault Systems, Velizy-Villacoublay, France.

[3] Dash, Pramod., "Reduction of Wingtip Vortex from Suction at Wingtip," *ResearchGate* URL:https://www.researchgate.net/publication/269672219_Reduction_of_Wingtip_Vortex_from_Suction_at_Wingtip [retrieved 18 September 2020].