Spring 1995

A Computer Model For Airfoils With Moving Surface Boundary-Layer Control

Jacob Kimball
Carroll College, Helena, MT

Follow this and additional works at: https://scholars.carroll.edu/mathengcompsci_theses

Part of the Aerodynamics and Fluid Mechanics Commons, and the Applied Mathematics Commons

Recommended Citation
https://scholars.carroll.edu/mathengcompsci_theses/91

This Thesis is brought to you for free and open access by the Mathematics, Engineering and Computer Science at Carroll Scholars. It has been accepted for inclusion in Mathematics, Engineering and Computer Science Undergraduate Theses by an authorized administrator of Carroll Scholars. For more information, please contact tkratz@carroll.edu.
A COMPUTER MODEL FOR AIRFOILS WITH MOVING SURFACE
BOUNDARY-LAYER CONTROL

Submitted in Partial Fulfillment of Requirements for
Graduation with Honors to the Department of Mathematics, Engineering,
Physics, and Computer Science, Helena, Montana.

Jacob Reed Kimball
April 7, 1995
A Computer Model for Airfoils with Moving Surface Boundary-Layer Control

By

Jacob Reed Kimball

This thesis for honors recognition has been approved for the Department of Mathematics, Engineering, Physics and Computer Science, Carroll College, Helena, Montana.

April 7, 1995
A Computer Model for Airfoils with Moving Surface Boundary-Layer Control

Abstract

By

Jacob Reed Kimball

One method that is used to enhance airplane safety and performance is to use moving surfaces on wings to increase lift. A modification of the panel method is used to model airfoils which incorporate moving surface boundary-layer control. The modification is to apply a vortex sheet of constant strength over the moving portions of the airfoil. The model is demonstrated on an airfoil that has a rotating cylinder on the leading edge. The modified panel method was programmed in Mathematica (Wolfram Research Incorporated). The results generated by the Mathematica program are displayed graphically and show an increase in lift for a symmetric airfoil.
# Table of Contents

List of Figures................................................................................................................iii

Acknowledgments..........................................................................................................iv

Introduction......................................................................................................................1

Fundamental Principles of Flight.....................................................................................1

Purpose of Research........................................................................................................4

Experimental Results......................................................................................................4

Aerodynamics..................................................................................................................5

Basic Concepts...............................................................................................................7

  a. Terms. ..................................................................................................................7

  b. Concepts. .............................................................................................................8

Panel Method................................................................................................................13

Moving Surface Modification of the Panel Method.......................................................20

Results and Conclusions...............................................................................................22

References....................................................................................................................29
List of Figures

Figure 1. The combination of circulation and uniform flow. ........................................2
Figure 2. A uniform flow field. ....................................................................................9
Figure 3. A source flow field. ...................................................................................9
Figure 4. A sink flow field. .......................................................................................10
Figure 5. A vortex flow field. ..................................................................................11
Figure 6. A cylinder in a uniform flow. ....................................................................12
Figure 7. A rotating cylinder in a uniform flow. .......................................................12
Figure 8. A symmetric airfoil and panel approximation. ...........................................14
Figure 9. A single panel method panel. ....................................................................15
Figure 10. Airfoil with a rotating cylinder. .................................................................21
Figure 11. Kuethe and Chow airfoil: $\alpha = 0^\circ$. ..................................................23
Figure 12. Kuethe and Chow airfoil: $\alpha = 3^\circ$. ...................................................23
Figure 13. Kuethe and Chow airfoil: $\alpha = 7^\circ$. ...................................................24
Figure 14. Symmetric airfoil with rotating cylinder: $\gamma_0 = 0$. ...............................25
Figure 15. Symmetric airfoil with rotating cylinder: $\gamma_0 = 1.5$. .............................25
Figure 16. Symmetric airfoil with rotating cylinder: $\gamma_0 = 2.5$. .............................26
Figure 17. Lift of rotating cylinder vs. non-rotating. ...............................................27
Acknowledgments

There are many people who I would like to thank, without their help I would have never finished this thesis. First and foremost is Dr. John Scharf, who no only directed my thesis but also provided and unending source of enthusiasm for the subject matter. Had it not been for Dr. Scharf, this thesis would have been a nightmare. Next are my readers: Dr. Glenn Gebert, who supplied the idea for the thesis and Dr. Ron Stottlemyer, who helped me clean up my writing.

I would also like to express my gratitude to the other faculty members who helped with my project. First, Dr. Szpilka, who helped in the early stages of the project. Next, Darrell Hagen who helped me remember how to program in FORTRAN again.

Finally, I would like to convey a special thanks to my family and friends for giving me support when I needed it. My parents seemed to know when to push me along and my friends seemed to know when I needed to escape. I did not realize before how much I needed all of these people.
Introduction

Airplanes and flight technology are integral parts of modern society. Today, things happen at a fast pace. People need to be able to get from point A to point B as quickly as possible. Airplanes are the fastest, most convenient way to travel. Because of the speed and convenience of airplanes, air travel is a vital mode of transportation. Since airplanes are so important in day-to-day life, it is essential that we continually strive to make them safer and more efficient.

The purpose of this paper is to explore one method used to enhance airplane safety by improving the lift generated by the wings and to develop a mathematical model that characterizes how the method works. The method incorporates moving surfaces on the wings, such as a rotating cylinder, which result in increased lift generated by the wings. However, to fully understand the method and the process of analysis, it is best to lay a solid foundation on which to build the analysis and model. The foundation starts with an overview of the fundamental principles of flight.

Fundamental Principles of Flight

There are four forces that act on an airplane in flight. The forces are lift, drag, thrust, and gravity. Lift is significant because it is the force that overcomes gravity and enables flight. Therefore, the basic ways that an airfoil generates lift are also important.

As an airplane moves through the air, the airflow over its surfaces generate lift and drag forces. The drag force is caused by friction between the air molecules and the skin of the airplane; however, drag is compensated for by the thrust force provided by the
engines. The most important force is the lift because it is the force that overcomes the weight of the airplane and enables flight. The nature of the airflow around its wings determines the lift. For an object to generate lift, there must be a uniform airflow past it in combination with a circulation of airflow around the object (see Figure 1).

Figure 1  The combination of circulation and a uniform flow.

A net circulation can be generated by putting a camber on the wing or by tilting the wing up in the uniform airflow. A wing is cambered by curving the upper surface to make it
longer than the lower surface. The camber forces the air over the top of the wing to go a longer distance in the same period of time as the air across the bottom surface. The greater curvature results in a faster flow over the top, giving a net circulation around the wing. Tilting the airfoil in the airflow has much the same affect as camber. The angle of tilt with respect to the direction of the uniform airflow is called the angle of attack.

Another way to generate lift on an object in an uniform airflow is by rotating it which creates a circulation around it. This effect is widely known as the Magnus effect. The most common example of the Magnus effect is the different motions of a baseball that can be caused by throwing the ball with various spins. The lift on airplane wings can also be enhanced by rotating objects attached to the wing, such as a cylinder along its leading edge.

To generate lift, the flow around the wings must be smooth. At angles of attack less than 15° - 20°, the flow over an airfoil is smooth. However, if the angle of attack is large enough, the flow near the downstream end begins to separate from the upper surface of the airfoil. At first the point of separation is near the trailing edge but as the angle of attack is increased, the point of separation moves forward on the upper surface of the airfoil. Eventually, the separation point reaches a critical location where the wing's lift decreases dramatically. This condition is called a stall.

Therefore, one of the main objectives of aeronautical engineers is to design an airplane wing that maximizes the angle of attack that can be achieved at a given airspeed without stalling. Improving the design in this way also allows the airplane to operate at
lower airspeed for a given angle of attack before a stall occurs. One method that engineers have considered to improve a wing's design is to place rotating cylinders (or moving belts) at various places on the wing's surfaces. The moving elements increase the net circulation around the wing at a given airspeed. Practically speaking, the induced circulation allows higher angles of attack before the separation of the flow and the subsequent stall.

**Purpose of Research**

Since the lift generated by an airfoil is important, the purpose of this project is to analyze the effects a rotating cylinder has on the lift. Generally speaking, the rotating cylinder will induce more circulation around the airfoil and thus increase the wing's lift (Mokhtarian and Modi, 1986). To analyze the results of this design and see how lift is actually improved, a numerical model called the panel method will be modified to accommodate the rotating cylinder's effect on the airfoil.

**Experimental results**

Wind-tunnel tests have been performed by researchers on a wing with a rotating cylinder on the leading edge. These tests show that a wing fitted with a rotating cylinder improves the lift and stall characteristics of an airfoil (Mokhtarian and Modi, 1986 and Modi, Mokhtarian, Fernando, and Yokomizo, 1989).
Aerodynamics

The following descriptions are adapted from Anderson, 1993. Their terminology and definitions are appropriate for the purpose of this study.

There are four forces that work in opposing pairs on the airplane: thrust vs. drag, and lift vs. weight. Briefly, thrust is the force that propels the plane forward. It is important because it creates the uniform flow by moving the airplane through the air. The drag force opposes the thrust because of friction between the air particles and the surface of the plane. The other opposing force pair is lift and weight. Lift is the upward force on the airplane generated primarily by the wings. Weight, on the other hand, is the earth's gravitational pull on the airplane.

The most important force in this study is the lift, the force generated primarily by the airplane's wing. The reason the wing generates lift is based upon the principle of conservation of energy. In fluid mechanics, conservation of energy is stated as Bernoulli's equation:

\[ p + \frac{1}{2} \rho v^2 = \text{constant} \]

This equation states that as the velocity, \( v \), of a fluid, the pressure, \( p \), must decrease so that the mathematical expression on the left remains constant. The mass density of the fluid is \( \rho \). Bernoulli's equation only applies in situations where the flow is steady, inviscid and incompressible. Incompressible flow means the mass density is constant under all conditions.
Bernoulli’s equation can be applied to the wing because the velocity of the flow over the top surface of the wing is greater than velocity of the flow over the bottom surface of the wing. The difference in velocity is the result of two things. First, the wing has camber, meaning the top surface of the airfoil is longer than the bottom surface due to curving of the surface. Second, the conservation of mass states that when the flow is broken apart, the particles that were together at the leading edge of the airfoil must come back together at the trailing edge. Since the upper surface is longer, the particles traveling over it must go faster to meet their counterparts from the lower surface at the trailing edge. The result is a net force difference upward that causes lift.

Another part of the aerodynamics of the situation is drag. Drag is a result of friction. In fluid dynamics friction has a specific name: viscosity. Viscosity plays a large part in the stall characteristics of airfoils. The model used in this study assumes that the flow around the airfoil is steady and that the particles that separate at the leading edge come back together at the trailing edge for all angles of attack. In reality this assumption is accurate for small angles of attack, less than 10°-15°, but for larger angles the flow behind the airfoil becomes turbulent, much like the flow behind a rock in a fast-flowing river. This aerodynamic model does not account for the effects of viscosity at high angles of attack.
Basic Concepts

Terms

Some of the terms and concepts from aerodynamics are defined to aid in understanding the model that is developed. These are adapted from Li and Lam, 1964, and from Mathews, 1988. The model used assumes a steady, inviscid, incompressible fluid flow. A steady flow means that at any given point in the fluid, the flow velocity and fluid parameters, such as temperature and mass density, do not change with time. Next, an inviscid flow neglects the effects of friction between the fluid particles and the airfoil’s surface. Finally, an incompressible flow means that the mass density of the fluid does not change.

Another part of assembling the model is to establish the way particles are described. There are two ways of describing fluid motion. One is called the Lagrangian system; the other, the Eulerian system. In fluid mechanics the Eulerian system is more commonly used while the Lagrangian system is widely used in solid mechanics. Simply stated, the Lagrangian system traces the motion of individual fluid particles as they move. The motion of each particle is described by a trajectory, which is called a pathline. On the other hand, the Eulerian system focuses on a fixed point in space and describes the motion of the various particles as they move past the point.

From the Eulerian perspective, a flow field is defined by the velocity of the fluid particles at each point in the space through which the fluid passes. A streamline through the flow field is a smooth curve whose tangent at any point is in the direction of the velocity vector at that point. In a steady flow, the streamlines indicate the path a particle
follows. In addition streamlines in a steady flow, by definition, cannot intersect because this condition would imply that there would be two velocities at the intersection point, which contradicts the steady-flow assumption.

In most situations the flow around an airfoil can be accurately modeled by two-dimensional flow fields. In a two-dimensional field the flow characteristics can vary from point to point in a plane, but remain constant in a direction perpendicular to the plane. Thus, all of the flows considered in this project are two-dimensional or planar.

However, even though the flow around the airfoil has been simplified to a two-dimensional flow field, it is still complex and intricate; fortunately, it can be decomposed into simpler flow patterns, which, when combined with each other, form the more complex flow field. The ability to combine the flows is the basic principle used in developing the model in this paper. These elementary flows can be best described by pictures of their streamlines.

**Concepts**

The first flow field is the uniform flow. Its streamlines are shown here as going from left to right. However, they can also go form right to left as horizontal lines across the plane and at any angle of tilt, as long as all the streamlines are still parallel (see Figure 2).
The second flow field is the source flow. To visualize this field, imagine a hole in a perfectly level table with a hose connected to the hole. Water comes out of the hole and spreads out over the surface of the table as shown in Figure 3.

Figure 2 A uniform flow field.

Figure 3 A source flow field.
The next flow field is the sink. Imagine the same table as before, except that instead of water going out of the hole, it is being drawn in, as in Figure 4.

![Figure 4](image)

Figure 4  A sink flow field.

The last flow field is generated by a vortex line, which is a line of purely angular flow. The fluid flows in circular paths around the line. There is also a sign convention, indicating the rotation of the vortex flow. A vortex of positive strength is one that rotates clockwise while a vortex of negative strength rotates counterclockwise (see Figure 5).
Simple or elementary flow fields, when added together in a variety of combinations, can model more complex and intricate fields, like the one developed around an airfoil. A simpler illustration of this technique is the combination of a source and a sink flow field, one on top of the other, in the presence of a uniform flow. This combination gives a flow field which replicates the air flow around and past a non-rotating cylinder (see Figure 6).
When a vortex is superimposed on the flow in Figure 6, the model simulates a rotating cylinder in a uniform flow (see Figure 7).

Figure 6  A cylinder in a uniform flow.

Figure 7  A rotating cylinder in a uniform flow.
It is interesting to contrast the situations represented in Figures 6 and 7. In Figure 6, the flow above and below the cylinder is the same, so there is no pressure difference and thus no lift on the cylinder. However, in Figure 7, the streamlines on the top of the cylinder are closer together than the ones on the bottom. This means that the velocity of the flow on top is faster. Then, from Bernoulli's principle, the pressure on top of the cylinder must be less than the pressure on the bottom, creating a net upward force or lift. This simple demonstration is important for two reasons. First, it shows how combinations of simple flows can model more complex situations. Second, these combinations can model flow patterns which generate lift forces. The examples above provide simple demonstrations of the basic principles behind a technique used to model the complex flow past an airfoil. This technique is called the panel method.

Panel Method

The following summary of the panel method is adapted from Kuethe and Chow, 1986, and the complex variable formulations from Mathews, 1988, and from Milne-Thompson, 1958.

Although it is not an exact method, the panel method provides good approximations of the characteristics of the flow around and past an airfoil.

This method starts by approximating the airfoil by a number of flat panels (see Figure 8).
The panels are connected on the surface of the airfoil, thus forming a polygon which approximates the airfoil surface. The analyst employing the method decides on the number of panels to use. A better approximation of the flow characteristics is obtained when more panels are used up to the limit of accuracy of the computer; however, the trade off is that more computations are required.

The next step in assembling the model for the panel method is based on combining a number of elementary flows to model the actual flow pattern. Vortex filaments, which vary in intensity, are applied at every point on the perimeter of the polygon that approximates the surface of the airfoil. The vortex intensity per unit length is assumed to vary linearly across each panel of the polygon. If $\gamma(S)$ is the vortex intensity per unit length, then the linear equation which describes this variation is
The strengths at the ends of the panel are $\gamma_k$ and $\gamma_{k+1}$; $S_k$ is the length of the panel; and $S$ is the distance along the panel (see Figure 9).

\[ \gamma(s) = \gamma_k + \frac{s}{S_k} (\gamma_{k+1} - \gamma_k) \]  

(1)

The vortex strength per unit length at the end of one panel must be the same as the vortex strength at the start of next adjacent panel. Therefore, there will be no discontinuity in vortex strength where the panels meet.

A flow field is assembled mathematically by adding the contributions from the vortex sheets and the uniform flow past the airfoil. The velocity of the flow at any point is calculated by adding the contributions from the vortex sheets and the uniform flow. The distribution of the vortex intensity around the surface of the airfoil is selected so that the resulting flow field accurately portrays the characteristics of the flow around the
airfoil. The theoretical condition imposed is that the surface of the airfoil must be a streamline of the flow. This condition is imposed by requiring the velocity of the flow to be tangent to the surface of the airfoil at the midpoint in each panel. Physically this stipulation makes sense because the condition does not allow the flow to go through the airfoil. If the airfoil is approximated by $N$ panels, then there will be $N+1$ unknown vortex strengths, $\gamma_k$ (see Figure 6). The streamline condition gives $N$ equations in $N+1$ unknowns.

However, another equation is still needed to solve the system of equations. The additional equation comes from the empirical observation that at lower angles of attack, the flow separates from the surface of the airfoil at the sharp trailing edge. This observation implies that there can be no circulation at the end of the airfoil and that the net vortex strength there has to be zero, that is, $\gamma_1 + \gamma_{N+1} = 0$. The equation that accounts for this observed condition is called the Kutta Condition.

To describe the flow field, a relationship between the vortices and the streamlines they cause is required. The use of complex variables simplifies the analysis. The first step is to describe the velocity of the fluid flow at each point in the complex plane with a velocity vector, $V(x,y) = V_x(x,y) + iV_y(x,y)$, where $V_x$ is the $x$-component and $V_y$ is the $y$-component. Because the flow is steady and inviscid, the flow field can be described by a complex potential function, $W(z)$, where $z = x + iy$. More specifically, the velocity is the conjugate of the derivative of the complex potential function, $V(z) = \overline{W'(z)}$. The complex potential for a vortex is
where $\Gamma$ is the strength and $z_0$ is the location of the vortex in the complex plane. The velocity potential for a uniform flow then is

$$V_\omega e^{i\alpha} z$$

where $V_\omega$ is the magnitude of the velocity and $\alpha$ is the angle of the flow, relative to the airfoil, at points far away from the airfoil.

At any point in the plane specified by $z$, the velocity can be calculated by summing the contributions of the vortex sheets on the panels and the uniform flow. At the point $z$, then the effect of each panel of vortices is calculated by integrating over the vortex distribution. To do this the vortex sheet is divided into elements of length $ds$, and the vortex strength of each element is $d\Gamma = \gamma (s) ds$. There is an infinite number of vortices on each panel. Therefore, each panel must be integrated over its length to find the panel's total contribution to the flow at the point $z$ from the $k^{th}$ panel. The equation is

$$W_{V_k} = \frac{\Gamma}{2\pi i} \frac{\gamma (s)}{\ln (z - z_0)} ds$$  \hspace{1cm} (4)$$

where $\gamma (s)$ is the vortex strength per unit length at each point $s$ on the panel as given by equation (1) and $W_{V_k}$ is the total contribution to the flow form that panel. Also, $z_0$ is the location of the vortex element, $d\Gamma$, and can be written as a linear function of $s$ as follows
\[ Z_o = Z_k + S e^{i\theta_k} \]  \hspace{1cm} (5)

where \( \theta_k \) is the angle the \( k^{th} \) panel makes with respect to the positive real axis (see Figure 9). Therefore, combining all of these equations (1-5), we arrive at the total potential at point \( z \) caused by the \( k^{th} \) panel:

\[ W_{z_k}(z) = \frac{1}{2\pi i} \int_0^{S_k} \left( \gamma_k + \frac{S}{S_k} (\gamma_{k+1} - \gamma_k) \right) \ln \left( z - z_k - se^{i\theta_k} \right) ds \]  \hspace{1cm} (6)

Next, the flow potentials for all the panels can be added to give the total potential at point \( z \) because of the airfoil:

\[ W_{\text{airfoil}}(z) = \sum_{k=1}^{N} F_k(z) \]  \hspace{1cm} (7)

Finally, the potential at point \( z \) is obtained by adding the potential for the uniform flow field to the total potential due to the airfoil:

\[ W_{\text{r}}(z) = W_{\text{airfoil}}(z) + V_{\infty} e^{i\alpha} z \]  \hspace{1cm} (8)

Now that the potential function has been assembled, the system of equations can be formulated by imposing the requirement that velocities are tangent to the surface of the airfoil at the center of each panel. This means that the component of the velocity perpendicular to the surface must be zero or, equivalently, that the flow cannot penetrate the surface. The equation expressing this requirement is as follows:
\[ V_n(z_c) = \text{real}(z_n \cdot W'(z_c)) = 0 \] (9)

where \( V_n(z_c) \) is the perpendicular component of velocity, \( z_c = \frac{z_k + z_{k+1}}{2} \) is the center of the panel, and \( z_n = \frac{z_{k+1} - z_k}{S_k} \) is the unit vector perpendicular to the panel.

When the process just described is applied to each panel, it gives an equation in the \( N+1 \) unknown \( \gamma_k \)'s. The equation for the \( i^{th} \) panel is of the form

\[ a_{i,1} \gamma_1 + a_{i,2} \gamma_2 + \ldots + a_{i,N+1} \gamma_{N+1} - b_i = 0 \] (10)

Therefore, \( N \) equations in the \( N+1 \) unknown \( \gamma_k \)'s will result, after the process is applied to all \( N \) panels. There is one more unknown than there are equations. The additional equation needed to solve the system is obtained by applying the Kutta condition,

\[ \gamma_1 + \gamma_{N+1} = 0. \]

Now there are \( N+1 \) equations in \( N+1 \) unknowns which can be solved for the \( \gamma_k \)'s. These vortex strengths are the "keys to the kingdom," as Dr. Glenn Gebert says. From the vortex strengths the velocity at each point in the flow field can be determined, and then from the velocities, the pressures can be calculated, using Bernoulli's equation. In addition, the \( \gamma_k \)'s can also be substituted back into the complex potential function, \( W_v(z) \), which completely characterizes the flow field around the airfoil. For example, the contours of the imaginary part of the potential function, \( \text{Im}(W_v(z)) \), are the streamlines of the flow and its derivative is the conjugate of the velocity field.
A better approximation of the flow field is obtained as the number of panels increases up to a point. The point being the accuracy that the computer. However, a better approximation also means that the system of equations to solve is larger and more computational effort is required. The system of equations is solved using a digital computer because the number of equations is large. The graphical capabilities of the computer also aid in visualization and interpretation of the results produced by the model. The model was programmed in a computer language called Mathematica. The Mathematica program was verified for accuracy by comparing results to those produced by a FORTRAN program in Foundations of Aerodynamics by Kuethe and Chow.

Mathematica, a mathematics software package developed by Wolfram Research, is an attractive computer language for this application because of its symbolic, numeric, and graphical capabilities. It is designed to handle a wide range of mathematical applications from simple algebra to calculus and beyond. More specifically, Mathematica can process the integrals, summations and dot products, and solve the system of equations necessary for replicating the streamlines around an airfoil pitched at different angles of attack.

Moving Surface Modification of the Panel Method

The purpose of this research is to analyze the effects that a rotating cylinder, on the leading edge of an airfoil, has on the lift produced by the airfoil. To perform the analysis the panel method was modified to accommodate moving surfaces, such as a
rotating cylinder. The lift as expected, improved by the increase in circulation around the airfoil, caused by the moving surfaces.

The net circulation of the flow around the airfoil can thus be changed by providing a mechanism which allows the surface or a portion of the surface to move. For example, this motion could be accomplished with a belt around the airfoil that moves like the sand paper belt on a belt sander or the treads on a tank. Another mechanism that would accomplish this function would be a rotating cylinder on the leading edge. In this case, most of the surface of the airfoil would not move, except for the exposed portion of the rotating cylinder (see Figure 10).

Figure 10  Airfoil with a rotating cylinder.

The moving surface will change the total potential of the flow field around the airfoil. This movement is modeled by superimposing a uniform vortex sheet over the portion of the surface which is moving. The contribution to the complex potential function is

\[ \int_{\text{moving surface}} \frac{\gamma_0}{2\pi i} \left[ \ln(z - z_0) \right] ds \]  

(11)

where \( \gamma_0 \) is the uniform vortex intensity per unit distance along the surface. When (10) is added to \( W_{v_\text{a}}(z) \), we obtain the potential function for an airfoil with moving surfaces in a uniform flow. The equation for the total potential function is
\[ W(z) = W_{v_0}(z) + \int_{\text{moving surface}}^\gamma \frac{\gamma}{2\pi i} \left[ \ln(z - z_0) \right] ds \]

\[ = W_{\text{airfoil}}(z) + V_0 e^{ia} z + \int_{\text{moving surface}}^\gamma \frac{\gamma}{2\pi i} \left[ \ln(z - z_0) \right] ds \]  

Now the analysis proceeds by forming the system of equations as was done in the unmodified panel method by applying the tangency condition in equation (12). Once the equations are solved, the flow field is generated by graphing the contours of the streamline function.

**Results and Conclusions**

First, an airfoil without a moving surface was analyzed to verify the program. The program was used to calculate the flow field around a NACA 2412 airfoil in a steady, uniform flow. The airfoil was presented to the steady flow at different angles of attack. For each angle of attack, the computer program assembled the governing set of equations and solved them to find the vortex distribution around the surface of the airfoil. The vortex distribution was then used to calculate the complex potential function in the space surrounding the airfoil. The results were compared with those generated by a FORTRAN program in a text book by Kuethe and Chow to verify that the program in *Mathematica* works correctly. The results produced by the Mathematica program matched the Kuethe and Chow results exactly for the case considered. The Mathematica program also calculated the lift vector associated with each angle of attack by taking the cross product of the circulation around the airfoil with the uniform-flow velocity. The

22
following figures generated by the program show the airfoil, the flow around it, and the corresponding lift vector at different angles of attack. The vertical scale is exaggerated. Consequently, the airfoil appears twice as thick as it actually is.

\[ \text{Figure 11} \quad \text{Kueth and Chow airfoil: } \alpha = 0^\circ \]

\[ \text{Figure 12} \quad \text{Kueth and Chow airfoil: } \alpha = 3^\circ \]
The panel method gives good results as demonstrated by the figures above. Twelve panels were used to generate the pictures. The number of panels used is relatively small, but even so the visual representations are quite good. The NACA 2412 airfoil was analyzed to verify that the program developed for this project gives the same results as the FORTRAN program in the Kuethe and Chow text. Consequently, the program provided a base for modifications to accommodate moving surfaces.

Next, the program was modified to analyze an airfoil with a rotating cylinder placed at the leading edge. The vortex strengths were calculated and the resulting streamlines and lift are shown in Figures 14-16. Again, the vertical scale is exaggerated. Consequently, the airfoil appears twice as thick as it actually is. The airfoil’s angle of attack is 4°.
Figure 14  Symmetric airfoil with rotating cylinder: $\gamma_0 = 0$

Figure 15  Symmetric airfoil with rotating cylinder: $\gamma_0 = 1.5$
Figure 16  Symmetric airfoil with rotating cylinder: $\gamma_0 = 2.5$

The figures show how the stream flow is modified by the rotation of the cylinder. A dynamic animation of a series of figures like the ones above shows a marked increase in the downward momentum of the fluid after it passes the airfoil, although the downward momentum increase is not as noticeable on the static pictures above. The change in momentum is caused by the airfoil and rotating cylinder exerting a net force on the air flow, which deflects it downward. Consequently, the air flow must exert a equal and opposite force on the airfoil, which is the lift. As the rotational speed of the cylinder increases, so does the downward momentum of the fluid and the resulting lift on the airfoil.

The rotating-cylinder model gives results that appear to be reasonable. However, the results were not compared with experimental measurements. A continuation of this project will include these comparisons.
The most important thing to note is that lift performance is enhanced and safety is improved by providing moving surfaces, such as a rotating cylinder on airfoils. Figure 17 shows results produced by the panel method program which has been modified to accommodate a rotating cylinder on the leading edge. This graph shows that for a given rotational speed and air speed, lift can be substantially increased at all angles of attack within the range of values considered. The graph also shows that to achieve the same amount of lift, the airfoil without a rotating cylinder must be at a higher angle of attack than the airfoil with a rotating cylinder. At a higher angle of attack, the airfoil is closer to the stall condition. Thus safety performance is improved with the rotating cylinder.

![Graph showing lift of rotating cylinder vs. non-rotating.](image)

**Figure 17** Lift of rotating cylinder vs. non-rotating.

The results indicate that a rotating cylinder does promote the increase in circulation, thus improving the lift that the airfoil produces. The increase in lift means that airfoils would be less likely to stall. Therefore, the planes would be safer to fly because pilots would have a larger margin for error.
The power of the panel method lies in its ability to calculate flow characteristics for problems that have no exact solutions. The panel method can be used to analyze general airfoils and very elaborate wing structures with multiple elements. With the modification to accommodate moving surfaces, the panel method may prove useful for analyzing airfoils with moving surfaces located at different parts of the airfoil.

The program also demonstrates the utility of Mathematica. In this application there are two distinct advantages that Mathematica has to offer. The first is that the mathematical formulations for a problem can be stated directly in Mathematica code, whereas considerable translation is required when using more primitive programming languages. The second virtue of Mathematica is its ability to produce graphic representations directly from generated data for visualization and interpretation of results.
References


