1st
Edition

Intermediate Fluid Mechanics

James Liburdy

Open Educational Resources

Oregon State University

Intermediate Fluid Mechanics

INTERMEDIATE FLUID MECHANICS

JAMES LIBURDY

Oregon State University
Corvallis, OR

Publication and ongoing maintenance of this textbook is possible due to grant support from Oregon State University Ecampus.

OERU Illustration support: Daniel Thompson

Suggest a correction (bit.ly/33cz3Q1)

Privacy (open.oregonstate.education/privacy)

CONTENTS

I. INTRODUCTION

This book is meant to be a second course in fluid mechanics that stresses applications dealing with external potential flows and intermediate viscous flows. Students are expected to have some background in some of the fundamental concepts of the definition of a fluid, hydrostatics, use of control volume conservation principles, initial exposure to the Navier-Stokes equations, and some elements of flow kinematics, such as streamlines and vorticity. It is not meant to be an in-depth study of potential flow or viscous flow, but is meant to expose students to additional analysis techniques for both of these categories of flows. We will see applications to aerodynamics, with analysis methods able to determine forces on arbitrary bodies. We will also examine some of the exact solutions of the Navier-Stokes equations based on classical fluid mechanics. Finally we will explore the complexities of turbulent flows and how for boundary layer flows one can predict drag forces. This compilation is drafted from notes used in the course Intermediate Fluid Mechanics, offered to seniors and first year graduate students who have a background in mechanical engineering or a closely related area.

In developing some of these more advanced topics there will be a number of mathematical tools developed and applied to specific flow situations. An early introduction to some of the basic concepts is presented in Chapter 2. But other mathematical tools and manipulations are introduced later as the topics require. Much of fluid mechanics can be developed from a mathematical point of view and students should realize that much of the early development was from mathematicians, such as Bernoulli, Euler, Navier, Stokes, whose names should sound familiar, and many others. However, as presented here the physical interpretations and applications are important and an attempt is made to develop analysis methods with an understanding of the physical consequences along side of all of the mathematical constraints and requirements of a problem or situation. It is hoped that the student not only learns the equations and how to manipulate them but also to understand the physical situations and how the physical flow phenomena are interpreted.

Fluid mechanics, as its name implies, is a subset of the larger field of mechanics. Mechanics is a branch of science dealing with forces and motion, and their relationships. Mechanics has static and dynamic elements. Forces may exist without motion and/or with motion and forces may initiate or change motion. Since fluid properties are significantly different than solids, fluid response to applied forces can be much more complex and difficult to describe. Due to fluid deformation rates (yielding to forces over time) fluids have complex distributions of pressure and velocity and acceleration. This spatial distribution of fluid motion is an important part of the understanding of how forces are transmitted within fluids. A major area of study in fluid mechanics is the kinematic motion of the fluid and how this is described. The dynamic flow of fluids is governed by Newton's law of momentum conservation whereby forces are required to accompany a rate change of momentum.

Since forces and motion are all around us and influence much of what we do in our everyday lives it is not surprising that the origins of mechanics, and fluid mechanics, dates back to the ancient Greeks. Archimedes was instrumental in developing the concepts of hydrostatics which are used to understand forces by fluids on its surroundings as well as how fluid pressure changes due to gravitational forces. Much of the interesting applications of fluid motion follow the formulations of the conservation of momentum. In this sense one is interested in how fluid motion is altered by the existence of imposed forces on the fluid. This is important in the applications of fluid transport (pipelines, biological systems, lubrication, chemical reactions and a host of other applications). It is also important when objects move within fluids. Since we are surrounded by fluids in our living environment, by either air or water, any motion of an object must deal with the fact that fluids must be "pushed out of the way". That is to say, object motion translates into fluid motion. And based on Newton's Law of reaction, forces acting on objects by fluids are related to forces by objects acting on fluids. So the guiding principles of fluid motion analysis comes from conservation principles of momentum when tied to other constraints of conservation of mass and energy.

The conservation equations illustrate the physical relationship governing fluid flow and the resultant forces on and by fluid flow. Flows are often classified by the inclusion of certain forces and/or certain effects. The largest classification is most likely between inviscid and viscous flows. This is a major division used in the development here. We first present inviscid flows where we analyze forces either on or by fluids and how they affect fluid motion. The primary forces are caused by pressure distributions and gravity. As we will see the pressure field or distribution in space, is influenced by the fluid motion. Consequently, the resulting forces can become rather complicated as pressure and velocity are intertwined. In contrast, gravity represents a constant force proportional to the mass of fluid as it plays a role in affecting fluid motion. After examining some of the inviscid flow situations, we introduce viscous effects and discuss how viscous dominated flows are of importance and how they may be analyzed. As we shall see the approach to these flows is very different because of the physical conditions, boundary conditions and resultant analysis methods that are used. The coupling of viscous effects, pressure and velocity create complex flow dynamics.

Potential flows are irrotational and allow for the determination of the flow field and pressure field based on the use of a scalar velocity potential. Potential flows ignore frictional effects. This has many advantages mathematically as well as allows for a good physical interpretation of the flow. However, there is lost information concerning any viscous effects that provide additional forces that may alter the flow field and pressure distribution. The underlying assumption is that these effects are minor for certain types of flows. One may hear different classifications of flows such as potential flow, ideal flow, incompressible or constant property flows. Potential flow, as mentioned, allows the replacement of the velocity vector with a velocity potential, which is a scalar and proves mathematically useful for many situations. We will deal with potential flows as inviscid, incompressible and also irrotational. The latter condition is that the vorticity is zero throughout the flow (except maybe at some singularity points within the flow). The definition of the velocity potential mathematically requires flows to be irrotational, as we shall see. The consequence of this is that the viscous terms vanish and the expression used for the acceleration of the fluid can be simplified. The solution to the resultant governing equations becomes very much simplified.

Another class of flows, known as ideal flows, are inviscid and incompressible. Typically, incompressible flows are those that do not have (significant) changes in density with changes of pressure. Liquids tend to be incompressible except under extreme conditions of high pressure changes. Gases may or may not be compressible depending on how large any pressure changes may be within the flow. Since pressure changes are linked to velocity changes within a flow it is possible to classify compressible effects through the flow conditions. Note that variations of density within a fluid flow can be caused by temperature effects, or if the fluid has a variation in concentration of some solute, but the flow still be treated as incompressible.

When including viscous effects the additional force may work with or against other forces, such as those due to pressure variations or body forces such as gravity. Interestingly the inclusion of viscous forces requires a model that depends on the properties of the fluid. This model relates how friction is measured within a moving, deforming fluid. It is not a universal law, say like conservation of mass or energy, it is mostly an empirical relationship that has been shown to be valid for a wide range of fluids and conditions. However, one can expect exceptions to this for some "exotic" fluids. We introduce the viscosity as a property of the fluid which defines the needed viscous force that results in a specified fluid deformation rate. This type of model is based on the work in the 1900s by Navier and Stokes who, separately, developed a brilliant formulation to account for viscous effects. This formulation is developed from the Cauchy equation which itself includes viscous effects in an overall identification of forces, or stresses, acting within the fluid flow field. The model allows the evaluation of stress terms based on deformation experienced by the fluid caused by viscous forces. In our applications we will restrict analyses to incompressible, constant property flows to be able to assess the contributions of the various terms. This has wide reaching applications, but does not delve into the realm of gas dynamics which combines certain equations of states with the conservation of mass, momentum and energy to evaluate compressible flows.

Another very important aspect of viscous forces is the fluid-surface boundary condition. Fluids, in contact with a solid surface, will tend to have the "no-slip" boundary condition. This is stated mathematically as the velocity at the interface is equal to the velocity of the surface, in both magnitude and direction. Moving away from this boundary the fluid velocity will change and the rate of change is found to be related to the frictional force on the surface caused by the fluid. By Newton's third law there is an equal and opposite force by the surface on the fluid. One must treat this boundary condition based on the view of the fluid as a "continuum". That is to say one does not get down to the molecular level, since at that scale molecules of fluid are bouncing off and on to the surface. The scale at which a fluid can be thought of as a continuum is larger than the mean free path of the molecules of the fluid, which is an average distance molecules travel before they collide with other molecules. For a continuum, the fluid velocity at the surface takes on the value of the surface velocity. There are applications, such in very small scale flows (sub-micron scale) where one can not treat the fluid as a continuum and in these situations "slip" may occur. Also, for rarified gases, under very low pressure, molecules can be very far apart and a continuum is not a valid approach. This transitions into the realm of the kinetic theory of gases. It is beyond the scope of this course to deal with these conditions.

Fluid properties are an important part of fluid flow analyses. Numerical results obviously greatly depend on values of say, density, viscosity, and other properties like surface tension or compressibility. However, here we do not go into these effects specifically. We do note that density is important in the relationship between pressure and velocity. This is easily noted by placing your hand out of the window of a car and comparing the resultant pressure on your hand with that of sticking it out of a boat into the water below when moving at the same speed as in the car. Since water density is nearly one thousand times greater for water compared with air, so is the resultant pressure. Also, a highly viscous fluid like honey will have different flow characteristics than a fluid which a much lower viscosity like water, as seen when pouring honey versus water from a jar under the action of the gravitational force. We will attempt to retain fluid properties in problems that are discussed so these types of distinctions become obvious. However, we will not go into the details of conditions of highly variable properties and the resultant change in flow and forces caused by this variability.

The rest of this book is organized as follows. Chapter 2 develops most of the common mathematical tools required for the rest of the book, although not exclusively. This is to provide a common level for mathematical notation and some manipulation. Chapter 3 develops a generalized Bernoulli equation, useful for later sections in the book. It also helps to explain the less restrictive conditions on the equation compared to that experienced by most first course students. Chapters 4 and 5 develop potential flow methods and solutions and chapter 6 utilizes this approach in developing the Panel Method for solving for pressure and forces on objects in external flow. Chapters 7, 8, 9 and 10 all deal with viscous flows. Chapter 7 develops the Navier-Stokes equation and chapter 8 provides a few classical "exact" solutions. Chapter 8 develops the boundary layer equations, the Blasius solution, and chapter 9 develops the integral solution method. Finally, Chapter 10 explores turbulence, its basic physics, some scaling conditions and a brief application to boundary layer flows. After completion of this book, students should have a better understanding of how to analyze both potential flows and viscous flows for incompressible conditions.

Online Material

- Potential Flow: MIT (web.mit.edu/2.016/www/handouts/2005Reading4.pdf)
- Efluids for general material and examples and images and videos (efluids.com)
- APS gallery of motion (gfm.aps.org)

II. MATHEMATICAL TOOLS

In this chapter we introduce a few mathematical tools that we will use in formulating some of the analysis of fluid flow problems for both inviscid and viscous flows. We introduce the use of tensor notation which is widely used in expressing fluid mechanics governing equations. We also show some mathematical manipulations that will help to provide some physical insight into the governing equations.

Tensor Notation

Most students are very familiar with vector notation (or Gibbs notation) for describing (usually) three component vectors in fluid mechanics. Vector notation implies the existence of components of the vector. A scalar, as is known is fully described by a single number, its magnitude. Of course a vector quantity may have more than three components in general, but here we are using vectors to describe components in three dimensional space. Each component may be a function of both space and time, and as such the entire vector is a function of space and time. The number of components in the vector is determined by the "dimensionality" of the vector. For instance a velocity field may only have two orthogonal directions, say (x,y), and as such the flow becomes two dimensional — the assumption is that there is no velocity variation in the third, z, direction. We can think of the three components as a set of three independent variables that the variable in question is a function of.

Tensor notation is an alternative approach and is a very powerful way of expressing any dimensional vector, as well as what are known as higher order tensors — variables that have several sets of independent variables to be considered. We say that a scalar is a zero order tensor and a vector is a first order tensor, such as velocity. An example of a second order tensor is stress. It requires two sets of indices to define its local value. A third order tensor requires a set of three indices to be fully defined. In our three dimensional world then, a first order tensor has three components. A second order tensor has nine components, determined by the possible combinations of the three elements within each set of indices. To make this clear note that for some vector V in three dimensions:

$$\mathbf{V} = u\mathbf{i} + v\mathbf{j} + w\mathbf{k} \; = u_i$$

Where u,v, and w are the values of the three components each a function of (x,y,z), $\mathbf{i}, \mathbf{j}$ and $\mathbf{k}$ are the unit vectors associated with x,y and z coordinates respectively and u_i is tensor notation for a vector V. The tensor representation uses a subscript which can have three possible values, each representing one of the three components the vector, or $u_1 = u, u_2$ = v and $u_3 = w$. Each component is denoted by a particular subscript that is defined based on the numerical value of the index. Here index number "1" is representative of the x coordinate, etc. But this is not limited to Cartesian coordinates. For example subscripts 1, 2 and 3 could represent r, θ and z in cylindrical coordinates.

Also, we can define a stress tensor as: τ_{ij} and the number of possible combinations of the subscripts i,j are nine. Do not confuse these subscripts with the unit vectors since they can take on one of the possible three directions. For fluid flow problems we will define the stress tensor and use it to arrive at our conservation of momentum equation. A third order tensor could be written as T_{ijk} and it would have 27 elements (3x3x3). As stated above, a single subscript denotes a vector and has 3 possible values, one for each coordinate.

It is possible to write all of our vector equations using tensors instead of the vector notation. This has some advantages and is used widely in fluid mechanics. Table 2.1 illustrates a number of examples of the equivalence of vector and tensor notation. A few of these are worth discussing and we will be using them as we write out the various equations that we will be formulating and using. It is suggested that you examine this notation and look at the various operators listed.

Operations among vectors carry with them some special consideration and rules. The gradient operator is ∇ in vector notation. If the quantity of interest is the gradient of scalar "a", written as $\nabla \mathbf{a}$, then this operation results in a vector whose components are partial derivatives in each of the three coordinates, $\frac{\partial a}{\partial x}$ is the x component, etc. In tensor operation this is expressed as $\frac{\partial a}{\partial x_i}$ which is a vector whose components correspond to each of the three possible values of the index "i". If the gradient is being taken of a vector, say u_j, the the resulting gradient expression is a second order tensor with the two indices, i for the partial derivatives and j for the vector quantity u_j. This is then written as $\frac{\partial u_j}{\partial x_i}$. Notice that the gradient operator index comes first since it operates on the vector u_j. There are nine combinations of i, j for this second order tensor. We will see this tensor in our discussions of viscous flows and vorticity.

Table 2.1 Listing of a few vector, tensors or scalars in vector and tensor notation (examples of vector vs tensor notation)

Quality	**Vector Notation**	**Tensor Notation**
Scalar	a	a
Vector	$\bar{a}$	a_i
2nd order tensor	$\overline{\overline{a}}$	a_{ij}
dot product (scalar)	$\bar{a} \cdot \bar{b}$	$a_i b_i$
cross product (vector)	$\bar{a} \times \bar{b}$	$\varepsilon_{ijk} a_j b_k$
del operator (vector)	$\bigtriangledown$	$\frac{\partial}{\partial x_i}$
gradient (vector; tensor)	$\bigtriangledown a; \bigtriangledown \bar{a}$	$\frac{\partial a}{\partial x_i}; \frac{\partial a_i}{\partial x_j}$
divergence (scalar)	$\bigtriangledown \cdot \bar{a}$	$\frac{\partial a_i}{\partial x_i}$
curl (vector)	$\bigtriangledown \times \bar{a}$	$\varepsilon_{ijk} \frac{\partial a_k}{\partial x_j}$
Laplace operator (scalar)	$\bigtriangledown^2$	$\frac{\partial^2}{\partial x_i \partial x_i}$
divergence of a 2nd order tensor	$\bigtriangledown \cdot \overline{\overline{a}}$	$\frac{\partial a_{ij}}{\partial x_i} (vector)$
dot product: vector & del	$\bar{a} \cdot \bigtriangledown$	$a_j \frac{\partial}{\partial x_j} (scalar)$
dot product: vector & gradient	$\bar{a} \cdot \bigtriangledown \bar{b}$	$a_j \frac{\partial b_i}{\partial x_j} (vector)$
Material Derivative	$\frac{\partial}{\partial t} + \bar{u} \cdot \bigtriangledown or \frac{D}{Dt}$	

Vorticity: (pseudovector)	$\omega_k = \bigtriangledown \mathbf{x} \bar{u} = \left(\frac{\partial u_j}{\partial x_i} - \frac{\partial u_i}{\partial x_j} \right) = \varepsilon_{kij} \frac{\partial u_j}{\partial x_i}$	
Rate of strain tensor: (2nd order tensor)	$\frac{1}{2}\left(\frac{\partial u_i}{\partial u_j} + \frac{\partial u_j}{\partial u_i} \right)$	

Consider now the divergence operation, in vector notation written as $\nabla \cdot \mathbf{V}$. This operation is the dot, or scalar, product between the vector operator ∇ and vector V. The result is a scalar. The tensor notation for this operation is $\frac{\partial u_i}{\partial x_i}$ where the indices for the vector u_i are the same as the index for the partial derivative operator. This implies that the same component of each vector, ∇ and V, should be used simultaneously. The rule in tensor notation then is to add all of these component operations to result in one final scalar. This becomes for the three coordinates, x_1, x_2 and x_3, which in Cartesian coordinates are x, y and z:

$$\frac{\partial u_i}{\partial x_i} = \frac{\partial u_1}{\partial x_1} + \frac{\partial u_2}{\partial x_2} + \frac{\partial u_3}{\partial x_3} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$$

The result is a scalar formed from the sum of each of the partial derivatives. As is apparent from this equation, when a term has a "repeated" index (such as the index "i" in the above equation for both u and x) then the rule is to sum over all values of the index. This applies to a particular term in an equation. As an additional example consider the term $\frac{\partial a_{ij}}{\partial x_i}$ in this case the index "i" is repeated and j is left as a "free index" (which could take on any of the possible three coordinate values.) Consequently this expression is a vector, with one free index, whose components are represented by the value of "j".There is also an operator, ε_{ijk}. This is given the name "permutation operator". Notice in Table 2.1 this is listed as used in the cross product and curl operation (the curl is the cross product of the vector operator ∇, which in tensor notation is written as $\frac{\partial}{\partial x_j}$, and the vector u_k). The permutation operator definition results in the cross product between two vectors. For this each of the indices can take on three different values, and there are 27 different possible combinations. The rules associated with the different values are as follows:

- if any of the three indices have the same value then $\varepsilon_{ijk} = 0$
- if the order of the indices is cyclic (1,2,3 or 2,3,1 or 3,1,2) then $\varepsilon_{ijk} = 1$
- if the order of the indices is anti-cyclic (1,3,2 or 2,1,3 or 3,2,1) then $\varepsilon_{ijk} = -1$

The result is three values for the permutation operator, with values being +1, -1, or 0. Consequently, the following occurs for the cross product: $\boldsymbol{a}_j$ x $\boldsymbol{b}_k$

$$\boldsymbol{a}_j \times \boldsymbol{b}_k = \varepsilon_{ijk} \boldsymbol{a}_j \boldsymbol{b}_k = (a_2 b_3 - a_3 b_2)\, \boldsymbol{i} + (a_3 b_1 - a_1 b_3)\, \boldsymbol{j} + (a_1 b_2 - a_2 b_1)\, \boldsymbol{k}$$

The reader should perform this operation for the curl of a vector where $\boldsymbol{a}_i$ is replaced by $\dfrac{\partial}{\partial \boldsymbol{x}_j}$ and $\boldsymbol{b}_k$ is replaced by $\boldsymbol{u}_k$.

There is another often used operator in tensor notation called the Kronecker delta, $\boldsymbol{\delta}_{ij}$. The rule used is that when $\boldsymbol{i} = \boldsymbol{j}$ the value of $\boldsymbol{\delta}_{ij}$ is one, otherwise it is set equal to zero, that is:

$$\delta_i j = 1 \text{ for } i = j$$

$$\delta_i j = 0 \text{ for } i \neq j$$

This comes in handy when one wants to include terms where there may be a nonzero value only when $\boldsymbol{i} = \boldsymbol{j}$. We will see this in our expression in the Navier-Stokes equation for the pressure contribution to the total force on a fluid element. For example suppose we want to include a gradient of a scalar, $\boldsymbol{S}$, as $\dfrac{\partial S}{\partial x_j}$ in an equation, note that this is a vector quantity. However, if we are evaluating different components of the gradient we only want to include those components that align with the vector component "$\boldsymbol{i}$", then we would write this term $\dfrac{\partial S}{\partial x_j}\delta_{ij}$. In doing this we see that when we evaluate this term we only include the gradient component in the $\boldsymbol{i}$ direction, since when operating on the gradient with $\boldsymbol{\delta}_{ij}$ results in a nonzero possible value only when $\boldsymbol{i} = \boldsymbol{j}$. We will use this for the pressure term in the Navier-Stokes equation as is explained later. We will use gradients, divergence and curl operations in the expressions used in the Navier-Stokes equations. The reader should become familiar with these.

Gradient, Divergence and Curl Operators

The gradient is a vector operator that for instance, when operated on a scalar produces a vector. It takes the partial derivative with respect to each component of the coordinate system. It is denoted as $\bigtriangledown$ and can be expressed in Cartesian coordinates as:

$$\bigtriangledown() = \frac{\partial()}{\partial x} i + \frac{\partial()}{\partial y} j + \frac{\partial()}{\partial z} k$$

Where a scalar of a vector or tensor could be inserted into the parenthesis. A gradient of a vector then has 9 components, where you take x,y,z derivatives of each of the three vector components. This would form the second order tensor:

$$\bigtriangledown V = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial z} \\ \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial x} & \frac{\partial w}{\partial y} & \frac{\partial w}{\partial z} \end{bmatrix}$$

If one takes the dot product of $\bigtriangledown$ with a vector we get the divergence of that vector which is a scaler:

$$\bigtriangledown \cdot V = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$$

Note that the divergence of the second order tensor yields a vector, such as $\bigtriangledown \cdot \bigtriangledown \boldsymbol{V}$. The x component of this vector is:

$$\bigtriangledown \cdot \bigtriangledown u = \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) i$$

The y and z components should be apparent where the x component of velocity is replaced by the y and z components of velocity in the numerators for the y and z components of the resultant vector.

Vector Identities

Vector Identity

Below are shown a few vector identities that are used in fluid mechanics in the manipulation of the governing equations of motion. A,B are vectors, ϕ, is a scalar.

$$\bigtriangledown (A \cdot B) = (A \cdot \bigtriangledown) B + (B \cdot \bigtriangledown) A + A \times (\bigtriangledown \times B) + B \times (\bigtriangledown \times A)$$

$$\frac{1}{2} \bigtriangledown (A \cdot A) = (A \cdot \bigtriangledown) A + A \times (\bigtriangledown \times A)$$

$$\bigtriangledown^2 \phi = \bigtriangledown \cdot (\bigtriangledown \phi)$$

$\nabla \cdot (\nabla \cdot A) = \nabla (\nabla \cdot A) - \nabla^2 A$

Divergence Theorem: $\int\int B \cdot dA = \int\int\int (\nabla \cdot B)\, dV$ (where dA is an elemental area around the boundary of volume V)

Stokes' Theorem: $\oint B \cdot dl = \int\int (\nabla \cdot B) \cdot dA$ (where dl is an element of the line integral around the boundary of area A)

There are a number of vector identities and manipulations that are very useful in fluid mechanics. We list a few of these here for future reference. Unless stated otherwise these are valid for any vector quantity, V.

The reader may want to visit this website (en.wikipedia.org/wiki/Vector_calculus_identities).

Material Derivative

The material derivative is a shorthand notation expressing the rate change of some property or parameter, which could be a scalar or a tensor, such as temperature, pressure or momentum (mass times velocity). This time derivative is based on a Lagrangian frame of reference of a fixed mass of fluid. So in a sense one is following a fixed mass of fluid and describing how the given property changes. Since the fluid mass can move we say that the fixed mass depends on both space and time. For example if we pick the scalar pressure, P, and say that our fixed mass has a pressure that is a function of space and time using Cartesian coordinates we have for the variable pressure:

$$P = f(x, y, z, t)$$

Then if we want its time rate of change for a fixed mass at a given time we use the notation that DP is the change of quantity P and DP/Dt is its change over time accounting for changes in both space and time (note we apply the chain rule for each of the possible coordinate directions:

$$\frac{DP}{Dt} = \frac{\partial P}{\partial t} + \frac{\partial P}{\partial x}\frac{dx}{dt} + \frac{\partial P}{\partial y}\frac{dy}{dt} + \frac{\partial P}{\partial z}\frac{dz}{dt}$$

Noting that $\frac{dx}{dt} = u$ and similarly for each coordinate direction, the equation becomes:

$$\frac{DP}{Dt} = \frac{\partial P}{\partial t} + u\frac{\partial P}{\partial x} + v\frac{\partial P}{\partial y} + w\frac{\partial P}{\partial z} \tag{2.1}$$

Things to pay attention to in this equation are (i) the velocity components u, v and w are not vector quantities, but components of the velocity vector, V; (ii) the time variation is specified by the first term and is evaluated at a particular point in space (as are all of the other terms) but all other terms may in fact be time dependent; this equation is valid for scalars or tensors, and the tensor sense is determined by the quantity being evaluated (in Equation (2.1) it is a scalar since P is a scalar). The material derivative of the velocity vector V, is:

$$\frac{DV}{dt} = \frac{\partial V}{\partial t} + u\frac{\partial V}{\partial x} + v\frac{\partial V}{\partial y} + w\frac{\partial V}{\partial z} \tag{2.2}$$

Since V is a vector then DV/Dt is also a vector, and each term on the right hand side is a vector and the components of each of these vectors is determined by inserting the component for the vector V. In other words the x component is:

$$\frac{Du}{Dt} = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \tag{2.3}$$

The above is for Cartesian coordinates. This can be written for any selected coordinate system. For example the result in cylindrical coordinates in r, θ is shown below.

Material Derivative in Cylindrical Coordinates

We would like to find an expression for DV/DT in cylindrical coordinates that we can use to help interpret streamline coordinates. We will only examine a two dimensional situation, r, θ since z is similar to Cartesian coordinates.

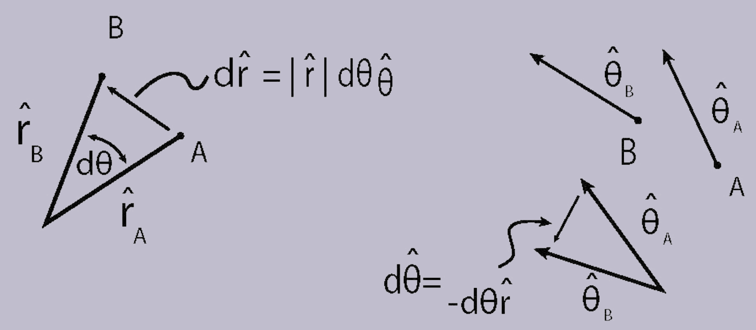
Given the vector $V = u_r\hat{r} + u_\theta\hat{\theta}$ we write this as a material derivative:

$$\frac{DV}{Dt} = \frac{\partial V}{\partial t} + u_r\frac{\partial V}{\partial r} + \frac{u_\theta}{r}\frac{\partial V}{\partial \theta}$$

When the angular position changes then there can be a change in both the unit vectors $\hat{r}$ and $\hat{\theta}$ since their orientation changes. So we can write the following:

$$\frac{\partial V}{\partial \theta} = \frac{\partial\left(u_r\hat{r} + u_\theta\hat{\theta}\right)}{\partial \theta} = \hat{r}\frac{\partial u_r}{\partial \theta} + u_r\frac{\partial \hat{r}}{\partial \theta} + \hat{\theta}\frac{\partial u_\theta}{\partial \theta} + u_\theta\frac{\partial \hat{\theta}}{\partial \theta}$$

Now we must interpret the derivatives of the unit vectors with respect to θ. Refer to the figure below showing a blob of fluid moving $\partial\theta$ that results in a change of the unit vector $\hat{r}$, that we can write as: $d\hat{r} = |\hat{r}|d\theta\hat{\theta} = d\theta\hat{\theta}$, which we can write as: $\frac{\partial \hat{r}}{\partial \theta} = \hat{\theta}$.



Similarly the change in the unit vector, $d\hat{\theta} = -rd\theta$, as illustrated above on the right as well. This is negative because if the change in the unit vector is counterclockwise the change in the r unit vector is in the negative r direction. The result is $\frac{\partial \hat{\theta}}{\partial \theta} = -r$.

All of this can now be combined and used in the material derivative above resulting in:

$$\frac{DV}{Dt} = \frac{\partial V}{\partial t} + u_r\frac{\partial V}{\partial r} + \frac{u_\theta \partial V}{r\partial\theta} = \left(\frac{\partial u_r}{\partial t} + u_r\frac{\partial u_r}{\partial r} + \frac{\partial_\theta \partial u_r}{r\partial\theta} - \frac{u_\theta^2}{r}\right)\hat{r}$$

$$+\left(\frac{\partial u_\theta}{\partial t} + u_r\frac{\partial u_\theta}{\partial r} + \frac{u_\theta \partial u_\theta}{r\partial\theta} + \frac{u_r u_\theta}{r}\right)\hat{\theta}$$

This applies to Cylindrical Coordinates, but as we shall see helps to interpret streamline coordinates when there is curvature to the streamline. The net result is added terms that account for the change of the unit vectors due to the curving nature of the flow.

Acceleration

If one finds the material derivative of the velocity vector of a fixed mass of fluid, namely Eqn. 2.2, the result is the acceleration of that fixed mass of fluid in time and space. The general expression for this vector quantity can be written in tensor notation as:

$$\frac{Du_i}{Dt} = \frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j} \tag{2.4}$$

Note here that the vector, V is replaced with the tensor notation for a vector written as u_i. Also the summation rule has been used in the second term on the right hand side, this implies that there are really three terms being represented by the one term shown. The first term on the right hand side is denoted as the "local acceleration" and physically represents the rate change of velocity at a fixed point in space. The second expression on the right hand side is denoted as the "convective acceleration" and physically represents how the velocity field changes over space. If a flow is steady the local acceleration is by definition zero. However a fluid mass as it moves through space where the velocity changes in space will experience acceleration (or deceleration). An example is steady flow through a nozzle, or diffuser, where the fluid velocity will increase, or decrease, along the flow direction. For these two cases a fixed mass of fluid will accelerate, or decelerate due to moving in space. The reader is reminded that the components of the acceleration depend on the component of velocity in the derivatives.The convective acceleration can be rewritten using a vector identity listed above. Namely, the convective acceleration in vector notation is $(V \cdot \nabla)\, V$ and can be expressed as: $(V \cdot \nabla)\, V = \frac{1}{2}\nabla\,(\mathrm{V{\cdot}V})\, - \mathrm{V}\times(\nabla\times\mathrm{V})$.

In tensor notation this becomes:

$$u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{2}\frac{\partial\,(u_j u_j)}{\partial x_j} - \varepsilon_{ijk}\left(u_j \varepsilon_{klm}\frac{\partial u_m}{\partial x_l}\right) \tag{2.5}$$

Consequently, the convective acceleration can be recast as the expression on the right hand side. Although this may not seem like it is of any benefit and just makes things more complicated we will show that it does indeed have advantages. The reader should pay attention to the subscripts in the above expression. Notice that each term is a vector with the free index of "i", all other indices are repeated and therefore summed. In the summation of vectors we basically need to sum components, this implies that they have the same index, in this case "i".

Streamline Coordinates

Streamline Coordinates define the flow direction of the velocity vector. It is defined as the locus of points tangent to the velocity vector at some instant in time. If the flow is steady these lines do not change. For our purposes we will consider steady flow here, but that is not necessary. Fig. 2.1 illustrates the streamline as well as a streamline coordinate system we will define. Our discussion for simplicity will be two dimensional such that flow is within the $s - n$ plane. We retain an orthogonal coordinate system with "s" along the streamline and "n" normal to the streamline. The third coordinate would be normal to both s and n, into the page.

If we take the cylindrical coordinate system as an example and consider flow along a curve, with some non-infinite radius of curvature, then we expect additional terms to appear in the acceleration. These terms account for the changing unit vectors as discussed in the material derivative in cylindrical coordinates. We make note of the fact that the velocity in the n direction is zero, by definition of the streamline. However, there can still be acceleration in the n direction since the unit vectors are changing. Using the cylindrical coordinate expression as a guide and denoting r as normal to the streamline direction, n, and θ as along the streamline direction, s, then the results for acceleration in the s and n directions are:

$$a_s = \frac{\partial u_s}{\partial t} + u_s \frac{\partial u_s}{\partial s} = \frac{\partial u_s}{\partial t} + \frac{\partial {u_s}^2}{2} \qquad (2.6)$$

$$a_n = \frac{{u_s}^2}{R}$$

Here R is the local value of the radius of curvature of the streamline. The value of R goes to infinity for a straight streamline and there is no acceleration in the n direction. Recall here that u_r (or u_n) is identically zero at the streamline and that the expression for a_n only contains the last term associated with the changing unit vector θ, or in this case s. Also we define in the streamline coordinates the direction n to be inward towards the radius of curvature as shown in Fig. 2.1. This is opposite to the convention used in cylindrical coordinates where r is radially outward, hence the sign is positive for the value of a_n. That is, if there is a curving streamline the acceleration is inward towards the direction of the center of the radius of curvature.

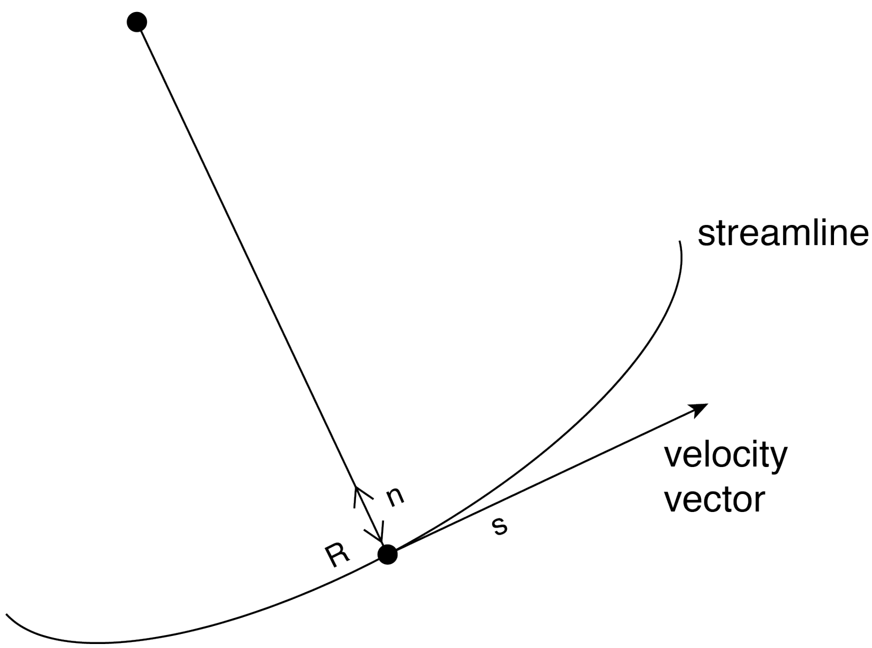


Fig 2.1 Streamline coordinate illustration; s is along the velocity vector direction and tangent to the streamline and n is normal to s while directed towards the center of radius, denoted as R.

Velocity Gradient Tensor

The velocity gradient tensor describes how the velocity varies near a specified location within the flow field. It is represented as $\dfrac{\partial u_j}{\partial x_i}$ which is a second order tensor, and therefor has nine components in three dimensional space. A second order tensor can be decomposed into symmetric and antisymmetric parts given as follows:

$$\frac{\partial u_j}{\partial x_i} = \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) + \frac{1}{2}\left(\frac{\partial u_j}{\partial x_i} - \frac{\partial u_i}{\partial x_j}\right) \tag{2.7}$$

Notice here that the first term adds half of the transpose of the original tensor and the second term subtracts the same quantity completing this identity. The first term is symmetric that is by inverting the i and j subscripts there is no change in the value of the element. The second term is antisymmetric such that inverting the subscripts the value of the element has the opposite sign. The ½ multiplier is included in the definition of the symmetric and antisymmetric parts such that their sum results in the original tensor.

In fluid mechanics applications the velocity gradient tensor is a measure of the velocity changes experienced in the fluid flow as one moves away (infinitesimally) from a given point. That is to say this tensor is a point function, having a value at any given point in the flow. We refer to that as a local function in space. It may be time dependent as well. So we can say that this tensor has a symmetric part and an antisymmetric part. As we will see when we discuss the viscous forces within a fluid the symmetric part is representative of the deformation rate, or strain rate, experienced by a fluid element. The antisymmetric part represents the rotation rate of a fluid element. If one examines the curl operator on u_j given previously as $\varepsilon_{kij}\dfrac{\partial u_j}{\partial x_i}$, where the subscripts indicate that this is the kth element, it can be shown by writing out the terms of the curl operation on u_j that this is equivalent to twice the antisymmetric part of $\dfrac{\partial u_j}{\partial x_i}$ (or the part in parentheses of the second term in Eqn. (2.7)). Therefore the rotation part of the velocity gradient tensor is one half the vorticity of the flow. To be clear, the antisymmetric part of $\dfrac{\partial u_j}{\partial x_i}$ is a second order tensor, but from observation if i=j then the value of this is identically zero (all zero components along the diagonal of the tensor). The off diagonal terms are six in total but those with inverted indices are the negatives of the noninverted components ($\dfrac{\partial u_j}{\partial x_i} = -\dfrac{\partial u_i}{\partial x_j}$). Consequently there are only three independent values, so we identify this second order tensor as a pseudo-vector, having three components. It is a so called pseudo-vector since the sign is arbitrary. In other words do we set $\dfrac{\partial u_1}{\partial x_2}$ as a positive or negative quantity? The sign convention is typically selected so that a counter clockwise rotation is a positive value. This means that for $\frac{\partial u_1}{\partial x_2}$ being positive then as u_1 increases in the positive x_2 direction then the flow will tend to rotate clockwise and is a negative rotation direction. This is illustrated in Fig. 2.2. Note that if one considers a negative value of $\frac{\partial u_1}{\partial x_2}$ coupled with a positive value of $\frac{\partial u_2}{\partial x_1}$ such that

$$\omega_3 = \left(\frac{\partial u_2}{\partial x_1} - \frac{\partial u_1}{\partial x_2} \right) > 0 \tag{2.8}$$

then rotation (or vorticity) about the 3 axis (out of the page in Fig. 2.2) is counterclockwise. Obviously, the signs of the two components of the velocity gradient that go into any of the vorticity components can be positive and/or negative. If in the above example for Eqn. 2.8 the magnitudes of $\frac{\partial u_2}{\partial x_1}$ and $\frac{\partial u_1}{\partial x_2}$ are equal and their signs are the same then the vorticity component ω_3 will be zero. One can think of the net vorticity or rotation rate vector component to be the combined value from both velocity derivative components, which contribute to rotation about an axis. The same interpretation of the combined effects of the two velocity gradient elements can be made for the strain rate, or the symmetric part of the velocity gradient tensor. When the rotation rates of the vertical and horizontal axes are identical (both elements causing, say, counterclockwise rotation at the same rate) then the strain rate will be zero (the two elements in the symmetric part cancel) and the flow is in pure rotation, as a solid body.

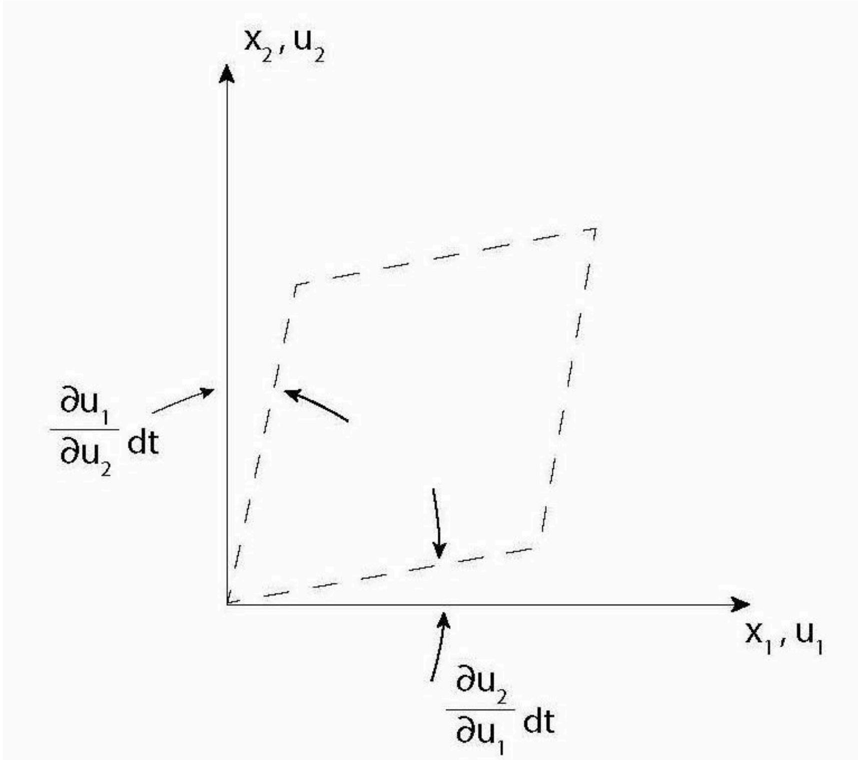


Fig 2.2 Illustration of rotation rate as determined by the velocity gradient tensor components; the original square element has its sides experiencing rotation at a rate of $\dfrac{\partial u_2}{\partial x_1}$ for the horizontal axis and $\dfrac{\partial u_1}{\partial x_2}$ for the vertical axis.

Basic Equations

The basic equations generally used in inviscid fluid mechanics are shown in Fig. 2.3. The viscous terms we will add later. But this chart shows various forms of the governing momentum equation which include pressure and gravitational forces contributing to the overall acceleration of the fluid. At the top is the basic Cauchy form of the momentum equation, where viscous effects would be contained in the stress term, $\boldsymbol{\tau}$. The inclusion of viscous terms will be developed in a later chapter. At this point the stress only contains the pressure forces, which is compressive and normal. The streamline coordinate formulation is shown on the right where integration is along the streamline. The introduction of the variable B is a consequence of using the last of the vector identities listed above as equation (2.5) for the convective acceleration term, and then combining the terms shown in the bottom box on the left. Here B is essentially the Bernoulli constant for steady, compressible flow. This is discussed further in the development of the Bernoulli equation. At this point the equations are presented for reference later on when the various terms are developed.

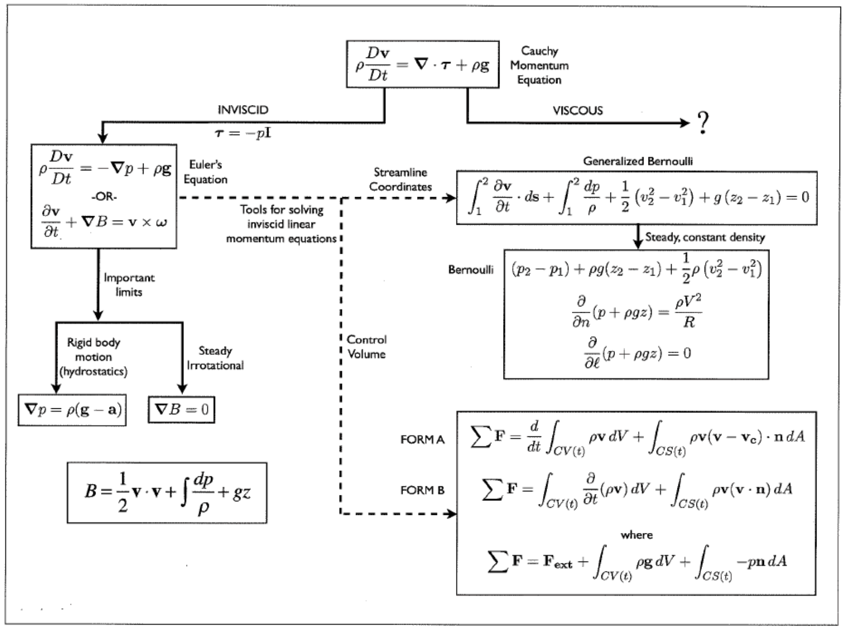


Fig 2.3 Chart illustrating the basic governing equation in inviscid flow.

III. BERNOULLI EQUATION

Generalized Form

The Bernoulli Equation is presented to most all engineering students and even high school students in a simplified form. This allows the development of a basic understanding of fundamental relationships between velocity and pressure within a flow field. It is typically written in the following form:

$$\frac{P}{\rho} + \frac{V^2}{2} + gz = constant \tag{3.1}$$

The restrictions placed on the application of this equation are rather limiting, but still this form of the equation is very powerful and can be applied to a large number of applications. But since it is so restrictive care must be take in its application. The restrictions can be stated as:

- incompressible flow (density is constant)
- viscous forces are assumed to be negligible (no internal fluid friction)
- steady flow (no time dependence)
- flow is along a streamline (apply between two points on the same streamline)

The first condition usually means it can be applied to a liquid or to a gas that has a relatively low velocity such that large changes in pressure do not occur. The second condition is a bit vague at best but assumes other forces, such as caused by pressure changes or body forces are much larger than frictional forces which may be valid when the viscosity is low, and/or when spatial velocity derivatives are not too large. The third term is self explanatory and disallows local acceleration. The last term requires the identification of a streamline and that the evaluation of Eqn. (3.1) occurs between two points along the streamline. A streamline by definition, and as is stated in chapter 2, assures that the velocity vector is tangent to the streamline. The Bernoulli equation can also be expressed by saying that the constant in the equation is the same at the starting and ending point such that the three terms sum to the same value at these two points and as such can be set equal to each other. Most students reading this will have a fairly extensive use of Eqn. (3.1) and there are many examples that can be found on the internet. The derivation of this equation is also available in many introductory fluid mechanics textbooks. The basic relationship stems from applying the momentum equation (without viscous forces included) along a streamline. This derivation will not be repeated here.

We wish to develop a more general form of the Bernoulli Equation that eliminates the restrictions to incompressible, steady flow along a streamline. The only restriction then is that viscous forces are ignored. The starting point for this development is the differential Euler's Equation for the motion of a fluid element that relates the acceleration to the forces caused by pressure and gravity. This can be expressed using the material derivative from chapter 2 as a balance of acceleration with pressure and body forces per unit mass of fluid:

$$\frac{D\boldsymbol{V}}{dt} = -\frac{1}{\rho}\frac{\partial P}{\partial \boldsymbol{x}} + \boldsymbol{g} \tag{3.2}$$

Or in tensor notation as:

$$\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + g_i \tag{3.3}$$

The pressure term is representative of the net force caused by the compressive load of pressure along the direction of the vector component of interest, on a per mass basis. The body force term is again given as the "$\boldsymbol{i}$" vector component of the gravitational vector. The acceleration terms on the left can be recast using the vector identity for the convective acceleration given in chapter 2.

$$u_j\frac{\partial u_i}{\partial x_j} = \frac{1}{2}\frac{\partial\left(u_j u_j\right)}{\partial x_j} - \varepsilon_{ijk}\left(u_j\varepsilon_{klm}\frac{\partial u_m}{\partial x_l}\right) \tag{3.4}$$

The last term introduces the vorticity into the equation since the vorticity is defined as the curl of the velocity vector:

$$\omega_k = \varepsilon_{klm}\frac{\partial u_m}{\partial x_l} \tag{3.5}$$

So the last term becomes the cross product of velocity and vorticity, or:

$$\varepsilon_{ijk}\left(u_j\varepsilon_{klm}\frac{\partial u_m}{\partial x_l}\right) = \varepsilon_{ijk}u_j\omega_k$$

Notice that the result of this operation is a vector in the "$\boldsymbol{i}$" direction, consistent with the other terms in the Euler equation. By introducing the vorticity into the convective acceleration term, this term can now can be considered to have two components. One is half of the gradient of the magnitude of the velocity vector squared (the first term), and the other is the cross product between the velocity vector and vorticity (a vector). This latter term has been identified as the "Lamb vector", after the applied mathematician, Horace Lamb (1849-1934).

Euler's equation can now be rewritten as:

$$\frac{\partial u_i}{\partial t} + \frac{1}{2}\frac{\partial (u_j u_j)}{\partial x_i} - \varepsilon_{ijk} u_j \omega_k = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + g_i \tag{3.6}$$

Next we write the Lamb vector as a gradient of some unknown function:

$$\frac{\partial \pi}{\partial x_i} = \varepsilon_{ijk} u_j \omega_k \tag{3.7}$$

In this expression π must be a scalar quantity such that its gradient is a vector. This is done for convenience so that we can then integrate each term of our equation along any desired path, with elemental distance ds . That is we take the projection of each term along ds (take the dot product of each term with vector ds). However, before we do this we make the following modification to the gravitational term such that we can integrate spatially along ds.

We can write the gravitational term as a "potential" as:

$$g_i = -g\frac{\partial h}{\partial x_i} \tag{3.8}$$

where "h" is a scalar and g is the magnitude acceleration of gravity. The choice of the symbol "h" is because it will have units of length and if gravity is vertically downward then "h" is vertically upward and represents elevation above some arbitrarily chosen datum. Notice that if "i" is horizontal then $\frac{\partial h}{\partial x_i}$ is the change in elevation over a differential change in the horizontal direction, consequently the value of $\frac{\partial h}{\partial x_i}$ is zero. If "i" is in the vertical upward direction then $\frac{\partial h}{\partial x_i} = 1$ and $g_i = -g$. If "i" is at some angle θ to the horizontal, then $\frac{\partial h}{\partial x_i} = \sin\theta$.

We are now able to write the integral along ds of Euler's equation. Below we use tensor notation and write the variable ds as ds_i; this results in a dot product between ds_i and dx_i. The dot product of each term results in a scalar that defines the magnitude of the change of the term along direction ds:

$$\int_s \frac{\partial u_i}{\partial t} ds_i + \int_s \frac{1}{2}\frac{\partial (u_j u_j)}{\partial x_j} ds_i - \int_s \frac{\partial \pi}{\partial x_i} ds_i + \int_s \frac{1}{\rho}\frac{\partial P}{\partial x_i} ds_i + \int_s g\frac{\partial h}{\partial x_i} ds_i = f(t) \tag{3.9}$$

Since the integration is only in space there may be a time dependence of each term that is not accounted for which is why the term on the right hand side appears — a general function of time can be added such that if one takes the spatial derivatives this function will vanish. Note that if there is no time dependence then $f(t) = 0$, as well as the first term on the left.

Performing the integration along the arbitrary line ds (which results in the change, Δ, along line "s"):

$$\int_s \frac{du_i}{dt} ds_i + \Delta\left(\frac{(u_j u_j)}{2}\right) - \Delta\pi + \int_s \frac{1}{\rho}\frac{\partial P}{\partial x_i} ds_i + \Delta\left(gh\right) = f(t) \tag{3.10}$$

As shown we have two integrals that we can not evaluate at this point. The first represents the local acceleration and how it varies along the integration path. The second is the pressure term. Note however if the flow is *incompressible*, ρ is constant, then we can say that:

$$\int_S \frac{1}{\rho}\frac{\partial P}{\partial x_i} ds_i = \frac{1}{\rho}\int_s \frac{\partial P}{\partial x_i} ds_i = \Delta\frac{P}{\rho}$$

Equation (3.10) excludes frictional or viscous forces, but that is about its only limitation. As such it is a general form of the Bernoulli Equation. But considering incompressible and steady flow the result is:

$$\Delta\left(\frac{(u_j u_j)}{2}\right) - \Delta\pi + \Delta\frac{P}{\rho} + \Delta\left(gh\right) = 0 \tag{3.11}$$

Consequently, the sum of these four terms which represent changes along any direction s is zero, or

$$\frac{(u_j u_j)}{2} - \pi + \frac{P}{\rho} + (gh) = constant \tag{3.12}$$

To satisfy some curiosity, one would expect that by applying the four conditions listed above for the specialized form of Eqn. (3.10) will result in Eqn. (3.1). If density is constant the pressure integral term becomes P/ρ as shown. If the flow is steady then the time dependence terms are zero. If the integration is taken along a streamline then we can make the following argument. The velocity vector by definition is aligned with the streamline, therefore the direction of ds and u_i are identical. The vorticity may be in any arbitrary direction, but the cross product of u_i and ω_j must be perpendicular to u_i. Consequently, since the integration is for the projection of each term along ds, and the cross product of u_i and ω_j is normal to ds (since it is normal to u_i) then the net effect of this term is zero (has no component along ds if s is a streamline). So if integration is along a streamline we can delete the Lamb vector effect,$\Delta\pi = 0$. Combining all of these conditions we end up with Eqn (3.1) as we hoped.

Other than along a streamline, another way in which $\Delta\pi = 0$ is if the vorticity is zero along any chosen integration path. That is to say, the vorticity is zero throughout the flow field. Let's examine the consequences of this condition. If each component of vorticity is zero then we can write a set of conditions, one for each component of the vorticity,

$$\omega_i = \left(\frac{\partial u_k}{\partial x_j} - \frac{\partial u_j}{\partial x_k}\right) = 0; \text{where } i \neq j \neq k \tag{3.13}$$

Now we define a "velocity potential" a scalar, ϕ, such that

$$u_j = \frac{\partial \phi}{\partial x_j} \text{ for all values of } j \tag{3.14}$$

This is a rather powerful condition — that a single scalar function,ϕ, can be used to define the velocity field through its partial derivatives. If one takes the derivatives of u_k and u_j as shown in Eqn. (3.13) while rewriting this in terms of the velocity potential, the result is an identity for $j \neq k$. That is to say, it is always = 0 since it is possible to reverse the order of differentiation. Therefore it is concluded that if one can replace the velocity vector with the derivatives of the scalar velocity potential defined in Eqn. (3.14) then the vorticity is zero everywhere. This can also be stated as: a scalar velocity potential exists and can be used to define the velocity field if the flow is vorticity free. Since vorticity can be defined as the degree of local rotation occurring in the flow we say the flow is "irrotational" if the vorticity is zero everywhere.

Using the above definition of a velocity potential, that exists for irrotational flow, it can be said that irrotational flow results in a simplified form of the Bernoulli Equation since π is zero when the vorticity is zero.

Here we summarize the Bernoulli Equation and how it is modified for different conditions.

1. General form
$$\int_s \frac{\partial u_i}{\partial t} ds_i + \left(\frac{(u_j u_j)}{2}\right) - \pi + \int_s \frac{1}{\rho}\frac{\partial P}{\partial x_i} ds_i + (gh) = f(t)$$
2. Incompressible flow form
$$\int_s \frac{\partial u_i}{\partial t} ds_i + \left(\frac{(u_j u_j)}{2}\right) - \pi + \frac{P}{\rho} + (gh) = f(t)$$
3. Steady form
$$\left(\frac{(u_j u_j)}{2}\right) - \pi + \int_s \frac{1}{\rho}\frac{\partial P}{\partial x_i} ds_i + (gh) = C \text{ (a constant)}$$
4. Irrotational, or along a streamline, form
$$\int_s \frac{\partial u_i}{\partial t} ds_i + \left(\frac{(u_j u_j)}{2}\right) + \int_s \frac{1}{\rho}\frac{\partial P}{\partial x_i} ds_i + (gh) = f(t)$$
5. Combination, steady, incompressible along a streamline, or irrotational
$$\left(\frac{(u_j u_j)}{2}\right) + \frac{P}{\rho} + (gh) = C \text{ (a constant)}$$

The last form is that which is most often introduced as a first exposure to the Bernoulli equation, yet it does come with a number of conditions, and one must be reminded that all of the above forms exclude any viscous, or frictional, force effects.

It is important to realize that the Bernoulli equation can be used for rotational or irrotational flow, but the former requires that it be applied along a streamline and viscous forces are not included. As is shown in the chapter of viscous flow, irrotational flow implies inviscid flow, so viscous forces are automatically eliminated. However, inviscid flow does not imply irrotational flow.

In the following chapters dealing with irrotational flow we will apply the Bernoulli equation between points where we know information, like far upstream of some object when there is flow over the object, to some point where we would like to calculate information, like on the surface of the object. However, since viscous forces are not included care must be taken to not apply the typical viscous boundary condition, of no-slip (forcing the fluid velocity to be equal to the surface boundary velocity.) Consequently inviscid flows allow slip, which means that the surface velocity of the fluid is some value that may need to be determined. This determination is the subject of the next two chapters.

We make one more modification to the unsteady form of the Bernoulli equation. We replace the velocity in the unsteady term with the velocity potential gradient using Eqn. (3.14). Then interchange the order of the integration and time derivative. Once we apply the integration along ds_i, the result is:

$$\frac{\partial}{\partial t}\int_s \frac{\partial \phi}{\partial x_i} ds_i = \frac{\partial}{\partial t}\left(\Delta_s \phi\right)$$

Now the time derivative is brought inside the spatial difference to yield the following for unsteady, incompressible flow:

$$\left(\frac{\partial \phi}{\partial t} + \frac{u_j u_j}{2} + \frac{P}{\rho} + gh\right) = f(t)$$

The π term is eliminated because introducing the velocity potential, ϕ, requires irrotational flow since $\nabla \times \nabla \phi = 0$ by a vector identity.

IV. POTENTIAL FLOW BASICS

Potential Flow Basics

Potential flows are those flow situations were the flow is taken to be irrotational, such that the vorticity is zero throughout the flow field (except at possible singularity points). This allows the use of a scalar function, ϕ, to describe the flow field through the definition:

$$\frac{\partial \phi}{\partial x_i} = u_i \tag{4.1}$$

This equation defines each component of the velocity in terms of the local spatial partial derivative in the direction of the velocity component. As stated in chapter 2 this definition when inserted for velocity in the definition of the vorticity results in the identity that the vorticity is zero, hence irrotational flow.

Continuity Equation

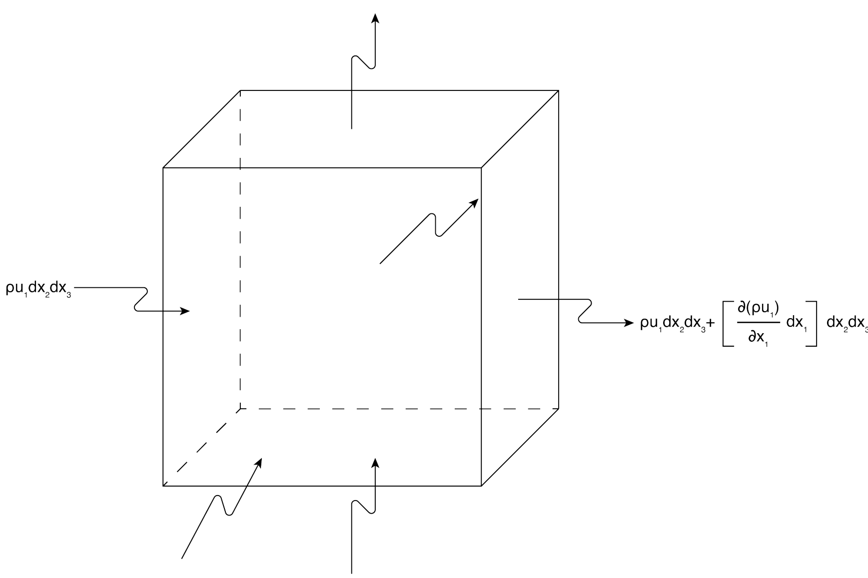


Fig 4.1 Illustration of a cubic element within a flow field with inflow and outflow at each face; the x_1 direction inflow (right) and outflow (left) is indicated where the outflow is expressed in terms of the change of density times velocity between the outflow and inflow.

Before we get into describing flows with the velocity potential we introduce the continuity equation. This equation comes from conservation of mass as applied to a continuum of fluid that may be in motion. The basic derivation of the continuity equation is shown in Fig. (4.1). Imagine a three dimensional volume in space that for convenience is shaped as a cube. Each face can have mass flow across this geometric element. We are interesting in finding the constraints on the flow field that satisfies conservation of mass for flow in/out of this volume. The basic physics of the relationship that we start from is that no mass can be created or destroyed (thus is conserved) over time. So the net flow in minus the net flow out must be equal to the net change in mass within the volume.

Putting this idea in equation form where the mass flow rate across any specified area for a continuum is the density times the velocity normal to the area times the area. We apply this to the faces of the cube:

Net flow in x_1 (outflow minus inflow):

$$\dot{m}_{out,x_1} - \dot{m}_{in,x_1} = \left[\rho u_1 dx_2 dx_3 + \left(\frac{\partial}{\partial x_1}(\rho u_1)\, dx_1\right) dx_2 dx_3\right] - [\rho u_1 dx_2 dx_3] = \left(\frac{\partial}{\partial x_1}(\rho u_1)\, dx_1\right) dx_2 dx_3 \quad (4.2)$$

This is repeated for the x_2 and x_3 direction where changes of density times velocity are with respect to $\partial x_2 \text{and } \partial x_3$ respectively, and u_2 and u_3 are used for the velocity, respectively, while using the area $dx_1 x_3$ and $d_1 x dx_2$, respectively. Summing all three of these net flow rates results in a scalar representation of the difference between the outflow and inflow where all possible flow paths in and out are included. Reversing the sign (to make it inflow minus outflow) this must equal to the change of mass within the volume element, $dx_1 dx_2 dx_3$.

This is expressed as:

$$\frac{\partial(\rho)}{\partial t} dx_1 dx_2 dx_3 = -\frac{\partial}{\partial x_1}(\rho u_1)\, dx_1 dx_2 dx_3 - \frac{\partial}{\partial x_2}(\rho u_2)\, dx_1 dx_2 dx_3 - \frac{\partial}{\partial x_3}(\rho u_3)\, dx_1 dx_2 dx_3$$

or rearranging and dividing each term by $dx_1 dx_2 dx_3$:

$$\frac{\partial(\rho)}{\partial t} + \frac{\partial}{\partial x_1}(\rho u_1) + \frac{\partial}{\partial x_2}(\rho u_2) + \frac{\partial}{\partial x_3}(\rho u_3) = 0 \quad (4.3)$$

This is the continuity equation that must be satisfied to conserve mass. Notice that if the flow is steady the first term is zero. Also if the density is constant (incompressible) then the first term, or the partial derivative with respect to time, is zero, and density can be factored from each of the other terms and divided out of the equation. The result is:

$$\frac{\partial u_1}{\partial x_1} + \frac{\partial u_2}{\partial x_2} + \frac{\partial u_3}{\partial x_3} = 0 = \frac{\partial u_i}{\partial x_i} \quad (4.4)$$

where in the last term there is summation by tensor notation. This is the reduced form of continuity for incompressible flow. Notice that this form does not require the flow be steady (even though the unsteady derivative of density is no longer included). The velocity may in fact vary with time.

If we now insert the definition of the velocity potential from Eqn. (4.1) for each of the velocity components in Eqn. (4.4) we end up with the following equation for ϕ for incompressible flow:

$$\frac{\partial^2 \phi}{\partial {x_1}^2} + \frac{\partial^2 \phi}{\partial {x_2}^2} + \frac{\partial^2 \phi}{\partial {x_3}^2} = 0 \tag{4.5}$$

This equation is the Laplace operation on the scalar velocity potential, ϕ, and represents continuity (or conservation of mass) for an incompressible flow.

Before moving on we write the continuity equation using the Material Derivative from chapter 2. We combine the time derivative of density with the other three terms but notice that there is a difference in the three spatial derivative terms from those found in the Material Derivative, the velocity is included in the derivative in continuity. So if each of these terms is expanded,

$$\frac{\partial\left(\rho u_i\right)}{\partial x_i} = \rho \frac{\partial\left(u_i\right)}{\partial x_i} + u_i \frac{\partial\left(\rho\right)}{\partial x_i}$$

The we see that:

$$\frac{D\rho}{Dt} + \rho \frac{\partial\left(u_i\right)}{\partial x_i} = 0$$

And if the density is y constant we obtain Eqn. (4.4) as expected.

Streamfunction

We now introduce the streamfunction, ψ. This is a scalar quantity as is the velocity potential. For simplicity we will do this in two dimensions, but it is valid in three dimensions as well. Recall we defined a streamline coordinate system (s, n) where s is aligned with the velocity vector. In general, for a time dependent flow the streamlines will be continually changing instant by instant. Within a coordinate system (say Cartesian or cylindrical or spherical) the streamline has an equation that can be written out in the selected coordinate system. The equation of this line can be represented by a streamfunction value. This is done as follows.

Consider a line that represents the instantaneous streamline within a flow. In Cartesian coordinates we can write the following for this line where we assume that there is some constant value ψ associated with the equation for the line:

$$\psi = f(x_1, x_2) = \text{constant}$$

For ψ being a constant small changes along a streamline we can write:

$$d\,\psi = 0 = \frac{\partial \psi}{\partial x_1} dx_1 + \frac{\partial \psi}{\partial x_2} dx_2$$

$$\frac{dx_2}{dx_1} = -\frac{\frac{\partial \psi}{\partial x_1}}{\frac{\partial \psi}{\partial x_2}}$$

Since $\dfrac{dx_2}{dx_1}$ is the slope of the line representing the streamline and since the velocity vector is tangent to the streamline, and the slope of the velocity vector is the ratio of the x_2 to x_1 velocity components we write:

$$\frac{u_2}{u_1} = -\frac{\frac{\partial \psi}{\partial x_1}}{\frac{\partial \psi}{\partial x_2}}$$

or we can write:

$$u_1 = \frac{\partial \psi}{\partial x_2} \text{ and } u_2 = -\frac{\partial \psi}{\partial x_1} \tag{4.6}$$

Eqn. (4.6) represents the definition of the scalar $\psi(x_1, x_2)$ associated with each streamline. If we know expressions for the velocity components as a function of position then we can integrate Eqn. (4.6) to find the value of the streamfunction, ψ. Similarly if we know the equation for the streamfunction then we can calculate the values of each velocity component through partial differentiation using Eqn. (4.6).

Let's assume an incompressible flow so that the flow field follows the continuity equation given by Eqn. (4.4). Now insert for the derivatives of the velocities in terms of the derivatives of the streamfunction. The results is:

$$\frac{\partial^2 \psi}{\partial x_1 \partial x_2} - \frac{\partial^2 \psi}{\partial x_1 \partial x_2} = 0$$

This is an identity, in other words it is automatically true, so the existence of the streamfunction, by the given definition, automatically solves the continuity equation. Said another way, if the streamfunction exists by its definition of Eqn. (4.6) then the flow satisfies continuity for incompressible conditions.

It is possible to take a given velocity field and construct a number of streamlines. At any given point there is a velocity vector and therefor a streamline that passes through it. The only time there can be two or more streamlines passing through a given point (intersecting at some random angle) is if the magnitude of the velocity is zero. Then both partial derivatives of Eqn. (4.6) are zero and the slope is not defined. A stagnation point is such an intersection of streamlines, as shown in Fig. (4.2) for flow over a cylinder. The streamfunction can be continuous up to the stagnation point and beyond, say following the cylinder surface, but it divides at the stagnation point one branch going up and another going down. Stagnation points don't have to be on surfaces they can be distributed within the flow field.

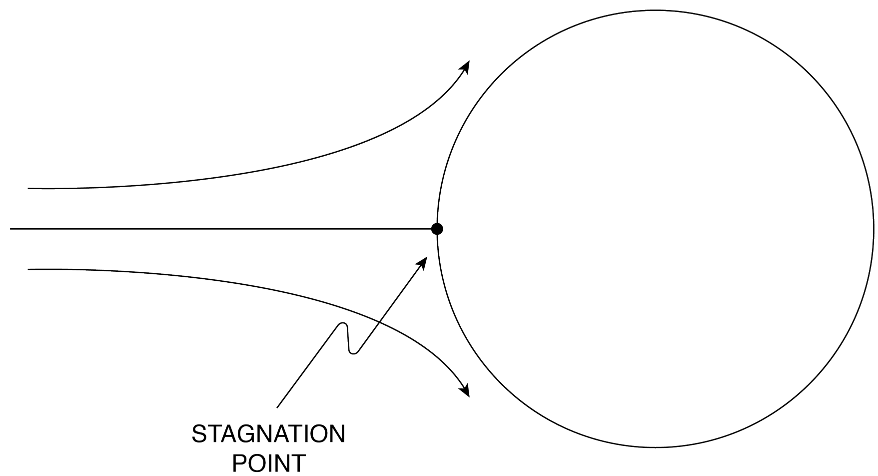


Fig 4.2 Streamlines illustrating a stagnation point streamline where flow separates away from this point, and the velocity at this point is zero.

Streamfunctions are valuable in that they can provide information on local flow rate conditions within a flow field. In general the flow rate (mass or volume) is determined by the velocity vector and an area through which the flow occurs. That is to say, the velocity vector only provides flow rate through an area if there is velocity vector component normal to the area. For a given area we define an outward normal unit vector, $\hat{n}$ as shown in Fig. (4.3). The mass flow rate through the area "A" with this outward normal is given as:

$$\dot{m} = \rho \boldsymbol{V} \bullet \hat{n} A \tag{4.7}$$

The reader should check the units for this equation. Notice that $\boldsymbol{V} \cdot \hat{n}$ is the dot product between the velocity and outward normal that results in a scalar whose value represents the projection of the velocity vector in the $\hat{n}$ direction. To obtain the volume flow rate, $\dot{Q}$ this expression is divided by mass per volume, or the density:

$$\dot{Q} = \boldsymbol{V} \bullet \hat{n} A \tag{4.8}$$

$$\dot{m} = \rho \left(\overline{\boldsymbol{V}} \bullet \hat{n} \right) A$$

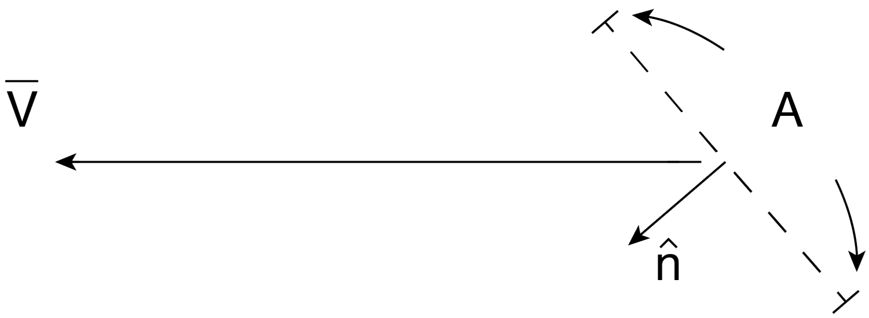


Fig 4.3 Flow through area A with outward normal $\hat{n}$ and velocity vector $\bar{V}$.

Now consider a two dimensional steady flow with a streamline distribution as shown in Fig. (4.4). Since the velocity vector is tangent to each streamline there can be no flow across a streamline. Consequently, the flow that occurs between two streamlines must remain between those two streamlines along the flow direction. In other words, the flow rate between two streamlines remains constant. The value of the flow rate can be interpreted in terms of the change in the streamfunction value between the two streamlines. This is shown as follows.

Consider the two streamfunctions in Fig. (4.4), such that the difference is $\Delta\psi = \psi_2 - \psi_1$. Next draw, a control volume as shown in the figure, where flow can enter through two areas, dx_1 and dx_2 (this is two dimensional representation so there is a unit distance into the page). The volumetric flow rate per unit depth into the control volume must balance the volumetric flow rate per unit depth out of the control volume, $\dot{Q}'$.

$$\dot{Q}' = \int_B^C u_2 dx_1 + \int_C^A u_1 dx_2 = -u_2 \triangle x_1 + u_1 \triangle x_2$$

$$\dot{Q}' = \triangle \psi_{C-B} + \triangle \psi_{A-C} = \triangle \psi_{A-B} = \psi_2 - \psi_1 \tag{4.9}$$

The reason there is a negative sign for $u_2\Delta x_1$ is that in determining the flow rate between points B and C we integrate along negative Δx_1 direction (or the change in Δx_1 is negative). Also we have used the definition of the streamfuction, Eqn. (4.7) to evaluate finite changes, on $u_1 = \triangle \psi / \triangle x_2$ and $u_2 = - \triangle \psi / \triangle x_1$. The interpretation then is that the flow rate $\dot{Q}'$ is equivalent to the change in streamfunction value between two points within the flow.

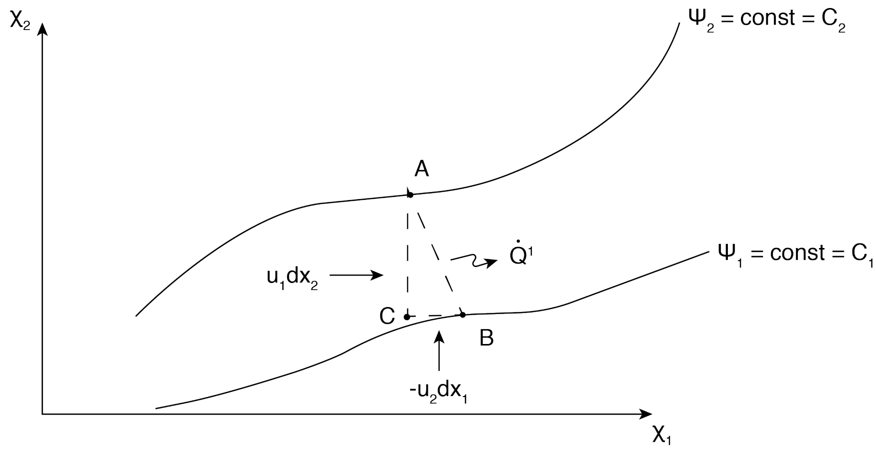


Fig 4.4 Flow between two different streamlines illustrating constant flow rate between the streamlines; control volume ABC is used to show relationships between flow rate and change in streamfunction values, Eqn. 4.9.

Interestingly, one can use streamline maps to qualitatively and quantitatively evaluate the velocity field. Imagine a wind tunnel test in a two dimensional flow over, say, a wing, as in Fig. (4.5). Smoke dye is injected at discrete points upstream separated by some vertical distance between each streamline. The lines of smoke travel downstream and over the wing. As the flow goes over the wing some of the streamlines diverge and some converge (the distance of separation between streamlines changes). Since we have shown that there is constant flow rate between streamlines when the distance between streamlines gets smaller the area of the flow decreases, so the velocity must increase. As streamlines diverge the velocity must decrease. The relationship between cross sectional area and velocity is linear as shown in Eqn. (4.8). Measuring the change of distance between adjacent streamlines provides a measure of the amount of increase or decrease of velocity.

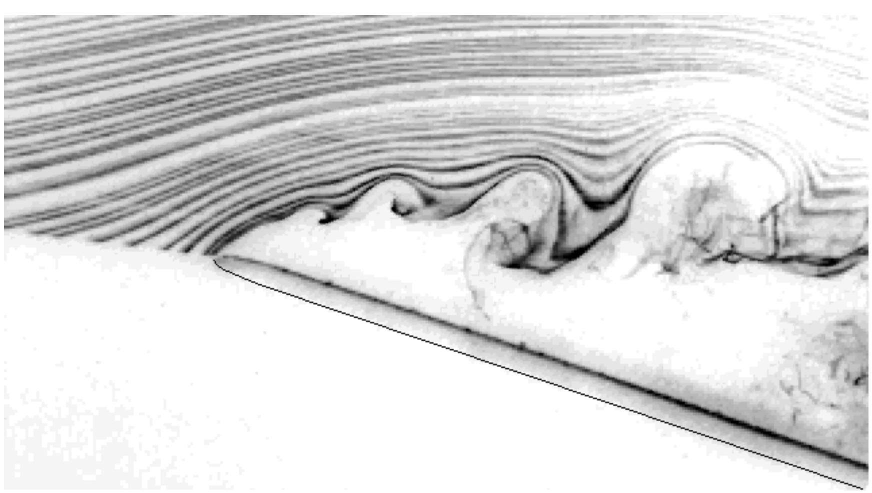


Fig 4.5 Smoke flow visualization of flow over an inclined flat wing.

We now have a physical as well as mathematical interpretation for the streamfunction. Remember that this is a scalar field function representative of the local velocity. If we use the definition of streamfunction, Eqn. (4.7) and insert this into the definition of vorticity for a two dimensional flow in the $x_1 - x_2$ plane we obtain the following:

$$\omega_3 = \left(\frac{\partial u_2}{\partial x_1} - \frac{\partial u_1}{\partial x_2}\right) = \left(-\frac{\partial^2 \psi}{\partial {x_1}^2} - \frac{\partial^2 \psi}{\partial {x_2}^2}\right) = -\left(\frac{\partial^2 \psi}{\partial {x_1}^2} + \frac{\partial^2 \psi}{\partial {x_2}^2}\right) \tag{4.10}$$

We see that the vorticity is equal to the negative of the Laplace of the streamfunction (Shown here in two dimensional flow, but is also the case in three dimensional flow). For $irrotational flow$ the vorticity is identically zero so:

$$\left(\frac{\partial^2 \psi}{\partial {x_1}^2} + \frac{\partial^2 \psi}{\partial {x_2}^2}\right) = 0$$

This shows that the Laplace of the streamfunction is zero for irrotational flow and follows the same results for the velocity potential for incompressible flow. So for irrotational, incompressible (ideal) flow the Laplace of both the velocity potential and streamfunction are equal to zero. This points to the ability to solve the Laplace equation for either of these quantities and from this solution determine the velocity field from the definitions of ψ and ϕ in teams of the velocity. This will be the approach we take in the next chapter.

Since both ϕ and ψ have the same functional form one might think that they are related to each other. We see this in comparing Eqns. (4.1) and (4.7), both are related to velocity derivatives. Notice that:

$$u_1 = \frac{\partial \phi}{\partial x_1} = \frac{\partial \psi}{\partial x_2} \text{ and } u_2 = \frac{\partial \phi}{\partial x_2} = -\frac{\partial \psi}{\partial x_1}$$

The velocity is tangent to the constant streamfunction value, but the velocity is normal to the constant velocity potential value. Consequently lines of constant ϕ are normal to lines of constant ψ. It is straightforward to show that the slope of constant potential lines is $-\frac{u_1}{u_2}$ while the slope of constant streamfunction lines is $\frac{u_2}{u_1}$. We will generate plots for specific flows in the next chapter to illustrate this, but shown in Fig. 4.6 is an illustration of the orthogonality of the velocity potential lines relative to the streamfunction lines. Velocity is always along the streamfunction line and normal to the potential lines.

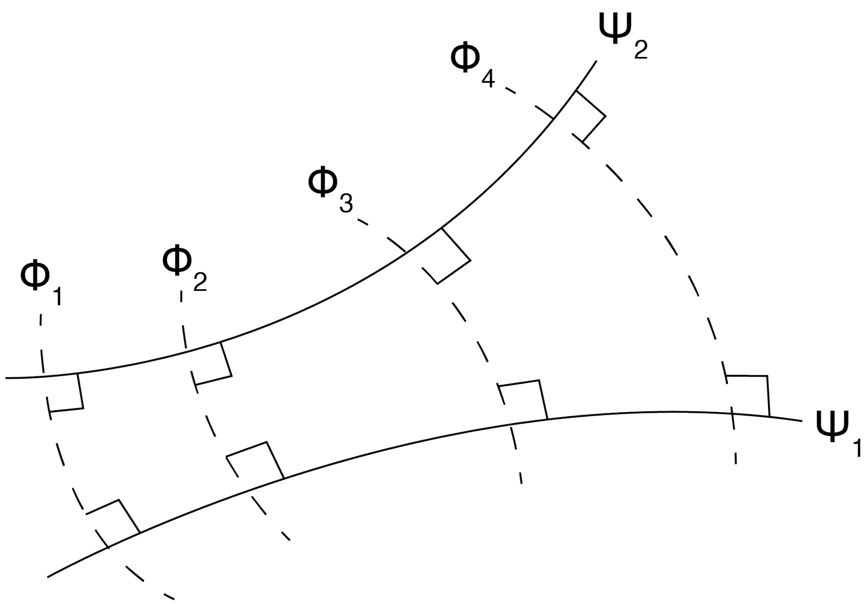


Fig 4.6 Illustration of streamlines (lines of constant ψ) and velocity potential lines, ϕ, indicating the orthogonal condition of the two lines.

In the table below, there are two dimensional expressions in cylindrical coordinates for the various mathematical representations presented here. Note that v_r is the radial velocity component and v_θ is the circumferential velocity component. The vorticity only has a z component, as shown, all others are identically zero for a two dimensional flow in r, Θ.

Cylindrical Coordinate Representation for Incompressible Flow

Continuity: $\dfrac{\partial(rv_r)}{r\partial r} + \dfrac{\partial v_\theta}{r\partial\theta} = 0$

Streamfunction: $v_r = \dfrac{\partial\psi}{r\partial\theta} \qquad v_\theta = -\dfrac{\partial\psi}{\partial r}$

Vorticity: $\omega_z = \left(\dfrac{\partial(rv_\theta)}{r\partial r} - \dfrac{\partial v_r}{r\partial\theta}\right)$

Velocity Potential: $v_r = \dfrac{\partial\phi}{\partial r} v_\theta = \dfrac{\partial\phi}{r\partial\theta}$

V. POTENTIAL FLOWS

In this chapter we will introduce a number of "basic ideal flows". These flows will form a basis from which we can construct more complex flows. The basic principle we are relying on is "superposition". This allows the linear addition of various flows that then result in more complicated flows. This is possible because the basic underlying equations that govern the flows are linear.

We have shown that there is an orthogonal relationship that exists between two variables that describe the flow, streamfunction and velocity potential, for ideal (irrotational and incompressible with zero viscous forces) flows. The orthogonal condition indicates that if we know one of these it is rather straight-forward to determine the other. We will use both of these flow descriptors to some extend, but mostly use the streamfunction representation of the various flows that we will consider. We will also restrict our results to two dimensional cases, although this is not necessary in general. Our basic equations are the Laplace equations we found in the previous chapter for the streamfunction, ψ, and velocity potential, ϕ.

For ideal flows we have the simplified continuity equation that treats the density as a constant, and allows the elimination of the density directly in the equation. This results in a relationship among the velocity components that must hold true to satisfy conservation of mass. Also, the irrotational flow condition requires the vorticity to be zero which leads to additional conditions on velocity derivatives within the flow. The boundary conditions need to be specified for such flows, and are required to solve the governing equations. Since these flows are inviscid we do not have the no-slip boundary condition to help specify the value of the velocity. This implies that the velocity may be some (finite) value at a surface, not equal to the surface velocity). However, we can require that a surface be impermeable, that is no flow crosses the boundary. This then assures that the component of the velocity normal to the surface must be zero. So at least we can say something quantitative about a velocity component.

Basic Flows

In this section we present the governing equations for several basic flows. These equations are solutions of the Laplace equation and are determined through required boundary or imposed flow conditions. We deal with steady two dimensional flows.

Uniform Flow

The most simple flow (other than zero flow) is a steady uniform flow. This condition is a constant velocity in a given direction such that the velocity vector does not vary spatially. We designate this velocity as U. If we align our coordinate system along the direction of U, such that x_1 is the direction of U, then there is only one nonzero velocity component. The streamfunction would be expected to be a straight line along the direction of U as well. Using the definition of ψ we obtain the following:

$$u = U = \frac{\partial \psi}{\partial x_2}$$

Integrating this in x_2 we obtain an expression for the streamfunction:

$$\psi = U x_2 + f(x_1)$$

But since

$$v = 0 = -\frac{\partial \psi}{\partial x_1}$$

then ψ is not a function of x_1 and we obtain

$$\psi = U x_2 + C \tag{5.1}$$

We can arbitrarily set $\psi = 0$ at $x_2 = 0$ so that

$$\psi = U x_2 \tag{5.2}$$

If we proceed along the same line to solve for ϕ from its definition relative to the partial velocity derivative, Eqn. (4.1), the result is:

$$\phi = U x_1 \tag{5.3}$$

where we have set $\phi = 0$ at $x_1 = 0$, other wise there is an additive constant which is equal to the potential at $x_1 = 0$.

For a condition of uniform flow, again with velocity of *U*, but at some angle α to the x_1 axis we have the following (the reader is encouraged to obtain this result by integrating the definition of the streamfunction):

$$\psi = U\left(x_2 cos\alpha - x_1 sin\alpha\right) + C \tag{5.4}$$

The constant C is determined by a “datum” value of ψ that passes through some point. For instance if $\psi = 0$ at $(0, 0)$ then $C = 0$.

Going back to uniform flow (two dimensional) aligned in the x_1 direction we can find the volumetric flow rate per depth through some area say between $x_2 = 0$ and $x_2 = 4$ as:

$$\dot{Q}' = \Delta\psi = U(4 - 0) = 4U$$

Source/Sink Flow

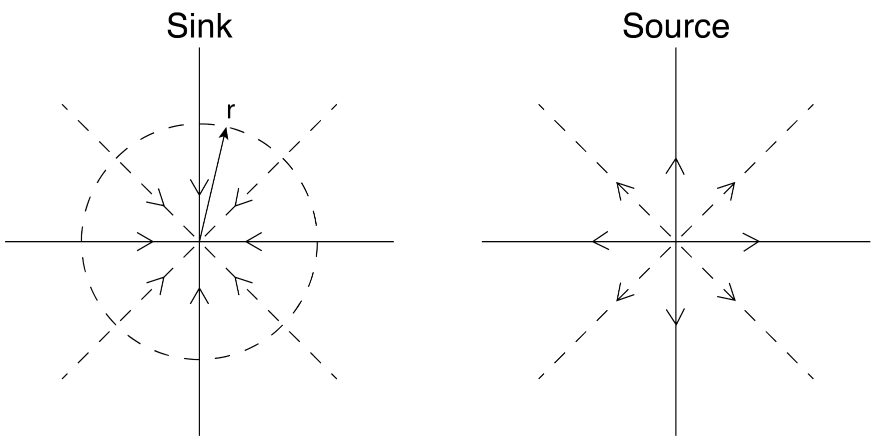


Fig 5.1 Sink and source flows from a point, a sink is the negative of the source and has an inward radial velocity.

In two dimensional flow a source or a sink of flow is possible, since it implies that flow enters or leaves a given two dimensional plane. We treat this as a rate of gain (source) or loss (sink) of mass at some point within the flow. An example of this might be a simulation of a drain say in a tank with a flat bottom. Along the bottom plane flow leaves the plane of the bottom at a small opening. Here we shrink the size of the opening to a point (which is hard to imagine since we have mass flow rate through this point, since the velocity associated with this point is infinite across the plane). Despite these unrealistic characteristics, if we move away from this "singularity" point of infinite velocity and zero area we can assign a flow rate. This is illustrated in Fig. 5.1, which illustrates how flow approaches the sink along the plane with velocity vectors that are inwardly, straight radial streamlines, as shown as vector lines in Fig. 5.1.

By drawing a circle of an arbitrary radius, r, shown as the dashed line in Fig. 5.1, the flow rate is determined by integrating the velocity around the circle. This is because the circle represents the flow area (per unit depth into the page), and the velocity vectors are each normal to the circle based on symmetry. The result for the mass flow rate per unit depth into the page is:

$$\dot{m}' = \rho v_r 2\pi r$$

Note that v_r is negative for a sink and positive for a source.

If a larger circle is drawn the same mass flow rate occurs into the point, but since the area is larger the velocity at the new larger circle is less. One can solve for the velocity, v_r, at any radial position, r, as:

$$v_r = \frac{\dot{m}'}{2\pi\rho}\frac{1}{r} = \frac{\dot{Q}'}{2\pi}\frac{1}{r} \tag{5.5}$$

For a constant steady volumetric flow, $\dot{Q}'$, we see that the velocity decreases linearly as r increases. So this flow has straight streamlines all represented as radial lines from the center point. We see how the velocity increases rapidly in magnitude as r goes to zero, at which point it becomes infinite (or undefined). Based on the orthogonal condition between the velocity potential and streamfunction we see that lines of constant potential are circles (like the dashed line shown in Fig. 5.1.)

We can combine the constants in Eqn. (5.5) $\mu_s = \frac{\dot{Q}'}{2\pi}$, where μ_s is the "source strength" with units of m^2/s in SI units such that:

$$v_{r,source} = \frac{\mu_s}{r} \tag{5.6}$$

Eqn. (5.6) represents the flow from a source with strength μ_s. The source pumps fluid into the plane of flow with streamlines that are radially outward.

The flow can be reversed such that the flow along the radial lines is inward. This implies flow is exiting the plane, and this is a "sink", with velocity inward, which has a negative $r$component:

$$v_{r,sink} = -\frac{\mu_s}{r} \tag{5.7}$$

The streamfunction is found by integrating the velocity based on the definition of the streamfunction using cylindrical coordinates:

$$v_r = \frac{\partial\psi}{r\partial\theta} = \frac{\mu_r}{r}$$

$$\psi = \mu_s\theta \tag{5.8}$$

Velocity potential lines are found from the definition of the potential relative to the velocity:

$$v_r = \frac{\partial\phi}{\partial r} = \frac{\mu_s}{r}$$

Integrating this results in:

$$\phi = \mu_s ln\, r \tag{5.9}$$

Vortex Flow

Now consider a swirling flow such that the streamlines are circles. This implies that there is no radial velocity component, only v_θ. We label this swirling flow as a "vortex", often called a "free vortex" since it is free from external forcing. It is also irrotational. If streamlines are circles as is shown in Fig. 5.2 then velocity potential lines must be straight radial lines from the center of the circle to be orthogonal to the streamlines.

To construct a flow with the above characteristics we examine the possible flow in cylindrical coordinates. Here we use the definition of the streamfunction in cylindrical coordinates:

$$v_\theta = -\frac{\partial \psi}{\partial r}$$

$$\omega_z = 0 = \frac{\partial (r v_\theta)}{\partial r}$$

So if we consider the flow to be given by its velocity $v_\theta = c/r$, where C is a constant, then the vorticity will be zero as required. Inserting this into the definition for the streamfunction above yields:

$$\psi = -Cln\, r \tag{5.10}$$

Next we define the "circulation" Γ, as the line integral of the velocity around a closed line, or a loop, as:

$$\Gamma = \oint v_\theta ds \tag{5.11}$$

This has units of m^2/s in SI units. Inserting the expression for v_θ above and noting that $ds = rd\theta$ with the integration carried out between 0 and 2π, we get:

$$\Gamma = \oint v_\theta ds = 2\pi C \tag{5.12}$$

The result is that the circulation within the vortex is a constant that can be determined knowing C. We can replace C with rv_θ and solve for the velocity as:

$$v_\theta = \frac{\Gamma}{2\pi r} = \frac{\mu_v}{r} \tag{5.13}$$

where we define the "vortex strength" as:

$$\mu_v = \frac{\Gamma}{2\pi} \tag{5.14}$$

And also using the definition of vorticity, which must be zero to be irrotational, as:

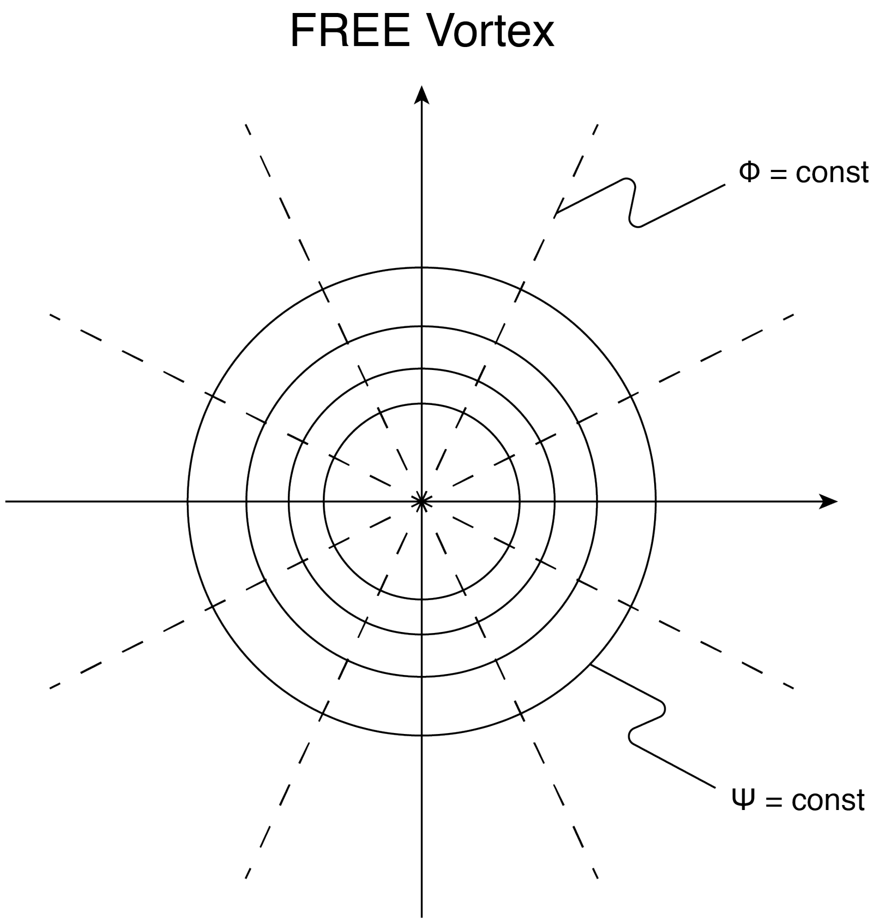


Fig 5.2 Sketch of the streamlines and potential lines for a free vortex; note that the circumferential velocity decreases in the radial direction.

The flow for a free vortex is shown in Fig. 5.2 indicating streamlines as circles and the velocity potential as straight radial lines. Notice that theses lines are just the inverse of what we found for the source.

Since streamlines can have the velocity direction either counterclockwise or clockwise and still have the same general form with the same equations as shown above the sign convention is that a counterclockwise rotation is positive, and a clockwise rotation is negative. This is expressed in the value of Γ, or μ_s. In other words counterclockwise flow has positive circulation.

One last interesting aspect of the free vortex and the circulation is as follows. We use the vector identity:

$$\int (\nabla \times \boldsymbol{V})\, dA = \oint \boldsymbol{V} \cdot \boldsymbol{ds}$$

where $\boldsymbol{V}$ is any vector, dA is an area and $d\mathbf{s}$ is the vector distance along a closed loop integration around the area A. If we let $\boldsymbol{V}$ be the velocity and noting that the vorticity is $\boldsymbol{\omega} = (\nabla \times \boldsymbol{V})$ and that the loop line integral is the circulation given above, then we can see that:

$$\Gamma = \oint v_\theta \boldsymbol{ds} = \int \boldsymbol{\omega} dA \tag{5.15}$$

Consequently the circulation is the area integral of the vorticity in some selected area within the flow. Note that we are dealing with a two dimensional flow and dA is within the plane of the flow so the vorticity is aligned along a vector out of the plane.

Now we have a bit of a dilemma. A free vortex has finite values of velocity v_θ that result is a certain circulation. This circulation is proportional to the vorticity within the flow. But we have said that the flow is irrotational, which means that the vorticity is zero. How can the circulation be nonzero while the vorticity is zero? The answer is that the vorticity (that drives the circular velocity) is concentrated at the center of the circle. Away from the center of the circle we know the vorticity is zero since the velocity we are using, v_θ, was based on the vorticity being zero. If one calculates the circulation about a circular loop of radius r_1 (an area of $\pi {r_1}^2$) and then repeats this calculation for a larger area of radius r_2, the result will be the same. This shows that in the area between r_1 and r_2 there is no added vorticity since the circulation remains the same. This can be done for any arbitrarily small radius r_1, showing that in the limit of r going to zero there is no vorticity within the flow except at r=0. A free vortex has a concentration of vorticity at r = 0. Also, the velocity must go to infinity, a velocity singularity. Interestingly this says that we can have isolated vorticity concentrations within an otherwise irrotational flow field.

Superposition

As stated previously superposition is a powerful tool that allows us to construct more complex flows from several simple flows. This is possible since the governing equation for the streamfunction, that describes streamlines, and the velocity potential are linear, is the Laplace equation.

We begin by examining the flow field in the vicinity of a source and sink. We place the source and sink on the x_1 axis separated by distance 2a, as is shown in Fig 5.3 (a), with the origin mid way between each. The source is on the left (negative x_1, and the sink is on the right (positive x_1). The streamfunction,or the velocity potential, at any point in the flow can be obtained by added together the streamfunction, or velocity potential, from the source plus that for the sink. However, care must be taken into account since our previously obtained equations were written assuming the source/sink were located at the origin of our coordinate system. In this case they are shifted along the x_1 axis. The result for any point P, located at (x_1, x_2) in the flow at distance r_1 and r_2 from the sink and source, respectively is:

$$\psi = \psi_{source} + \psi_{sink}$$

$$\psi = \mu_s \theta_1 - \mu_s \theta_2 \tag{5.16}$$

where it is assumed the source and sink have equal but opposite strength and the angles are shown in Fig. 5.3. Similarly for the velocity potential:

$$\phi = \mu_s ln\, r_2 - \mu_s ln\, r_1 = \mu_s ln\, \frac{r_2}{r_1} \tag{5.17}$$

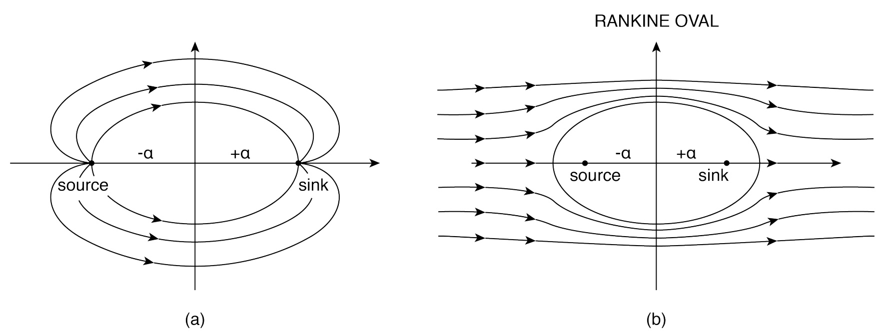


Fig 5.3 (a) Superposition of a source and sink of equal strength both positioned on the x_1 axis distance "a" from the origin, the source on the left and sink is on the right; **(b)** superposition of a uniform flow, a source and a sink which creates a Rankine Oval

In order to arrive at an equation for the streamfunction for the combined flow we use a trig identity:

$$tan(\theta_1 \pm \theta_2) = \frac{tan\theta_1 \pm\ tan\ \theta_2}{1 \mp tan\ \theta_1 tan\ \theta_2}$$

So taking the arctan of both sides and noting that $tan\ \theta_1 = \dfrac{x_2}{x_1 + a}$ and $tan\ \theta_2 = \dfrac{x_2}{x_1 - a}$, so that the right hand side of the above equation becomes: $arctan\left(\dfrac{2ax_2}{{x_1}^2 - a^2 + {x_2}^2}\right)$, the streamfunction is:

$$\psi = -\mu_s\ arctan\left(\frac{2ax_2}{{x_1}^2 - a^2 + {x_2}^2}\right) \tag{5.18}$$

For example, the streamfunction at $P = (3, 4)$ for a = 2 is $\psi = -\mu_s(37.3^o)$.

At this point we have a source and sink separated by a distance of 2a. Extending this we can add a uniform flow in the positive x_1 direction to the combined source sink flow to obtain the following:

$$\psi = \ Ux_2 - \mu_s \ arctan \left(\frac{2ax_2}{{x_1}^2 - a^2 + {x_2}^2} \right) = Ur \ sin \ \theta + \mu_s \left(\theta_1 - \theta_2 \right) \tag{5.19}$$

where the last term is the result written in cylindrical coordinates. If we set the streamfunction equal to some constant we can plot the associated streamline from Eqn. (5.19). In particular setting $\psi = 0$ an oval results as shown in Fig. 5.3 (b). This is known as the *Rankine Oval*. The characteristics of this oval can be adjusted by inserting different values for μ_s for a given U. The characteristics of the oval are:

$$\text{Major axis } (x_2 = 0) = \ 2a \left(1 + \frac{2\mu_s}{a} \right)^{1/2}$$

$$\text{Minor axis } (x_1 = 0) = 2a \left(cot \ \frac{h/2a}{2\mu_s/aU} \right) \ (\text{where } h = \text{minor axis}) \tag{5.20}$$

It is possible then to model the flow over an oval surface with an approach velocity of U by Eqn. (5.19). The velocity is determined by taking the derivatives of ψ relative to x_1, and x_2. The geometry of the oval can be adjusted by varying the strength of the source/sink as well as their locations, a.

Doublet

A doublet is a result of construction of a flow field using the superposition of a source and a sink that are placed very close to each other. The superposition of these two will result in flow leaving the source and entering the sink. The flow lines form circular paths as the flow attempts to leave the source while being drawn in to the sink. By making the strength of the source and sink identical a symmetric flow will result. It will be shown how this flow establishes a streamline that is a circle in the limit of the source and sink approaching each other spatially. This type of flow has powerful applications to simulate more complicated flows as we will see.

To extend the use of a source and sink of equal strength to a doublet we take the limit as their separation distance $a \to 0$. First we note that the arc tan of a small number is equal to the number so $arctan \left(\frac{2ax_2}{{x_1}^2 - a^2 + {x_2}^2} \right) = \frac{2ax_2}{{x_1}^2 - a^2 + {x_2}^2}$ then we have:

$$\psi = -\frac{2a\mu_s x_2}{{x_1}^2 - a^2 + {x_2}^2}$$

Since we want $a \to 0$ we will also let $\mu_s \to \infty$ at the same time and say $a\mu_s = C$, where C is some constant. This may seem arbitrary but it assures us that the streamfunction doesn't vanish to zero and since we can set μ_s to any desired value. The result is that we can rewrite the equation for the streamfunction of a doublet as:

$$\psi = -\mu_d \frac{x_2}{{x_1}^2 + {x_2}^2} \tag{5.21}$$

where $\mu_d = 2C$ a constant that determines the "strength of the doublet". This equation can be rearranged:

$${x_1}^2 + \left(x_2 + \frac{\mu_d}{2\psi}\right)^2 = \left(\frac{\mu_d}{2\psi}\right)^2$$

Noting that for a constant value of ψ the coefficient on the right inside the parenthesis, and the same term added to x_2 is a constant resulting in an equation of a circle. The center of the circle changes with changing values of ψ as does the radius of the circle. The result is a series of streamlines for selected values of ψ, each of which is a circle centered along the x_2 axis as is shown in Fig. 5.4. Notice that the radius of the circle is $\frac{\mu_d}{2\psi}$.

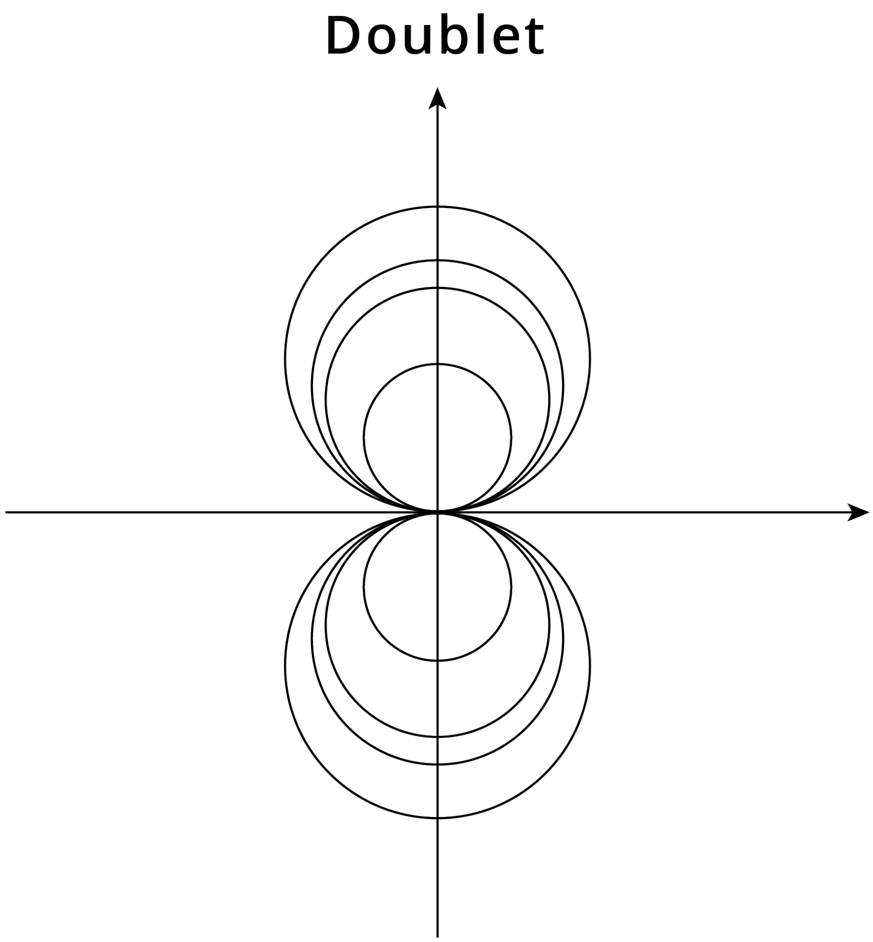


Fig 5.4 Streamlines associated with a doublet – a source and sink with a distance a approaching zero

Uniform Flow over a Cylinder

The doublet can be added to a uniform flow in the positive x_1 direction resulting in a streamfunction given by:

$$\psi = Ux_2 - \frac{\mu_d x_2}{{x_1}^2 - a^2 + {x_2}^2} \tag{5.22}$$

This equation can be recast in cylindrical coordinates where $r^2 = {x_1}^2 + {x_2}^2$, $x_1 = rcos\theta$ and $x_2 = rsin\theta$ and we define $a = \left(\frac{\mu_d}{U}\right)^{1/2}$, a constant for a given value of U, the result is:

$$\psi = Ur \ \ sin\theta - Ua^2 \frac{sin\ \theta}{r} = Ur\ sin\theta \left(1 - \frac{a^2}{r^2}\right) \tag{5.23}$$

It can be seen that in the limit of large values of r the flow reverts back to uniform flow and the contribution for the doublet goes to zero. If we let $\psi = 0$ then $r = a$, a constant. That is to say, the streamfunction of $\psi = 0$ is a circle of radius a. So we have now constructed the flow field for uniform flow over a cylinder of radius a.

From this the velocity components in the r, θ coordinates are found to be:

$$v_r = \frac{\partial \psi}{r \partial \theta} = U\, cos\theta \left(1 - \frac{a^2}{r^2}\right)$$

$$v_\theta = -\frac{\partial \psi}{\partial r} = -U \sin\theta \left(1 + \frac{a^2}{r^2}\right) \tag{5.24}$$

and the velocity vector is $V^2(r, \theta) = {v_r}^2 + {v_\theta}^2$. The streamline distribution is shown in Fig. 5.5. We are only interested in the flow on the outside of the circle, which represents a cylinder. It is important to note that at $r = a$, the velocity is not zero and depends on θ. So at different positions around the cylinder surface the velocity will change. The largest velocity occurs at $\theta = 90^o$ and 270^o, which represents the top and bottom of the cylinder. This is where the streamlines converge the most indicative of high velocity flow. Also at $\theta = 0^o$ and 180^o the velocity is zero. These are stagnation points on the cylinder. Notice also that the streamlines are symmetric about the x_1 and x_2 axes. This has important implications on the forces that exist on the cylinder caused by the flow.

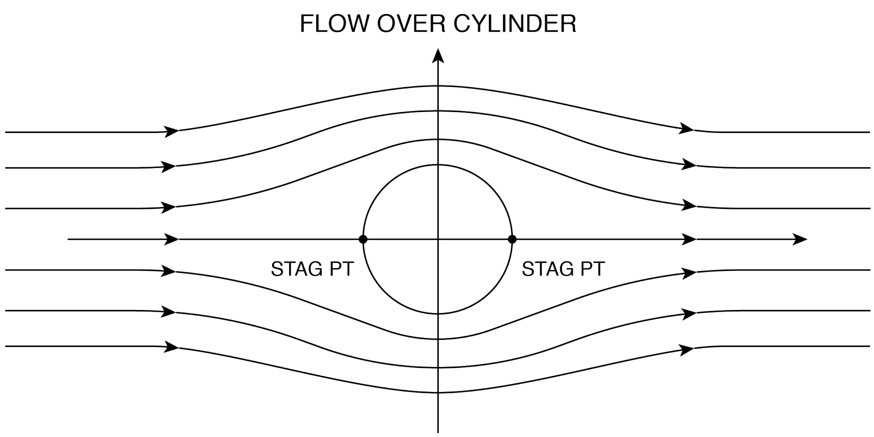


Fig 5.5 Streamline for flow over a cylinder formed from the superposition of a doublet with uniform flow; note stagnation on either side of the cylinder

Imaging

There is a method, call imaging, that allows for the insertion of impervious walls or boundaries within a flow. The basic idea here is that a streamline can be used to simulate a solid boundary since it does not allow flow to cross the streamline location. Consequently, if basic flow elements can be combined to yield a streamline that is of the desired shape of a boundary then it satisfies the required flow. For instance, in the condition of uniform flow over a cylinder described above flow elements are combined (uniform flow and a doublet) that result in a simulation of flow over a cylinder, of radius.

Consider the case of a source flow in the vicinity of a flat wall. This may be a simulation of pumping fluid from a well into the surrounding porous region near a solid boundary, such as a rock formation. This could be reversed and have flow into the well (sink). This is shown in Fig. 5.6 where the source is some distance away from the flat wall. We take the wall to be the x_1 axis and the source to be a distance "l": away from the wall. If we make the strength of a "mirror image" of the source on the other side of the wall the flow becomes symmetric about the wall with the wall being a streamline (no flow crosses it). Note here that the velocity component in the x_2 direction at the wall is zero but not the x_1 component. Without providing further details the superposition results in the following equation for the streamfunction:

$$\psi = \mu_s (\theta_1 + \theta_2)$$

θ_1 and θ_2 are measured from the source and its image, respectively, to any point within the flow. This is similar to what was done when combining a source and sink resulting in the Rankine Oval. In fact the analysis following Eqn. (5.16) can be repeated for this flow but the sign in front of θ_2 replaced with the positive sign shown above for ψ.

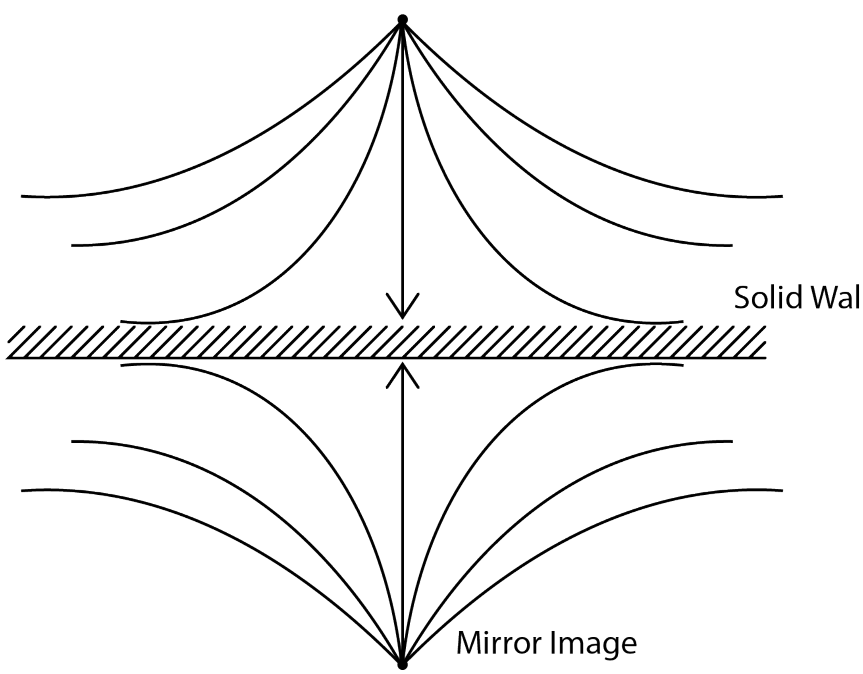


Fig 5.6 Streamlines of a source near a solid impermeable boundary formed using a mirror image, the wall is a streamline from the superposition of an identical source at an equal distance from the wall on the opposite side.

We introduce one more superposition, that of uniform flow, a doublet and a vortex. The combined streamfunction is:

$$\psi = \ Ur\ sin\ \theta\ - \mu_d\ \frac{sin\ \theta}{r} - \mu_v ln\ r\ + C \tag{5.25}$$

where C is a constant that can be determined by setting the value of ψ at some point in the flow, as shown below. The coordinate system used here has its origin at the center of the circle generated by the uniform flow and doublet. The added vortex does not add a radial component of velocity, since its flow streamlines are all circles. The result is adding a θ component of velocity throughout the flow that depends on the radial location. In Eqn. (5.25) the direction of the added circumferential flow is clockwise (negative). To summarize the strengths of the elements we have $U, \mu_d = Ua^2$ and $\mu_v = \ \frac{\Gamma}{2\pi}$. We can think of these as adjustable parameters to the flow field.

Rewriting Eqn. (5.25) by combining the first and second term as was done for flow over a cylinder and then setting $r = a$, inserting the expression for μ_d, as well as setting $\psi = 0$, we have:

$$C = \mu_v ln\ a$$

This then sets $\psi = 0$ on the circle with radius a. Inserting this value of *C* in Eqn. (5.25) we obtain:

$$\psi = \ U\ sin\ \theta\ \left(r - \frac{a^2}{r}\right) - \mu_v ln\ \frac{r}{a} \tag{5.26}$$

This represents uniform flow over a rotating cylinder as shown in Fig. 5.7, where the streamline representing the cylinder has $\psi = 0$. Notice that since we have a clockwise (negative) circulation the cylinder is rotating clockwise. The velocity on the surface of the cylinder now has a contribution caused by the vortex in addition to the velocity found for a non-rotating cylinder. We see that the stagnation points have moved away from along the x_1 axis. Since we know that the velocity is zero at the stagnation points we can solve for their location (note that $v_r\ = 0$ everywhere on the cylinder).

$$v_\theta(r = a) = -\frac{\partial \psi}{\partial r} = -2Usin\ \theta\ + \frac{\Gamma}{2\pi a} \tag{5.27}$$

or

$$sin\theta_{stag} = \frac{\Gamma}{4\pi a V} \text{ for } V_\theta = 0$$

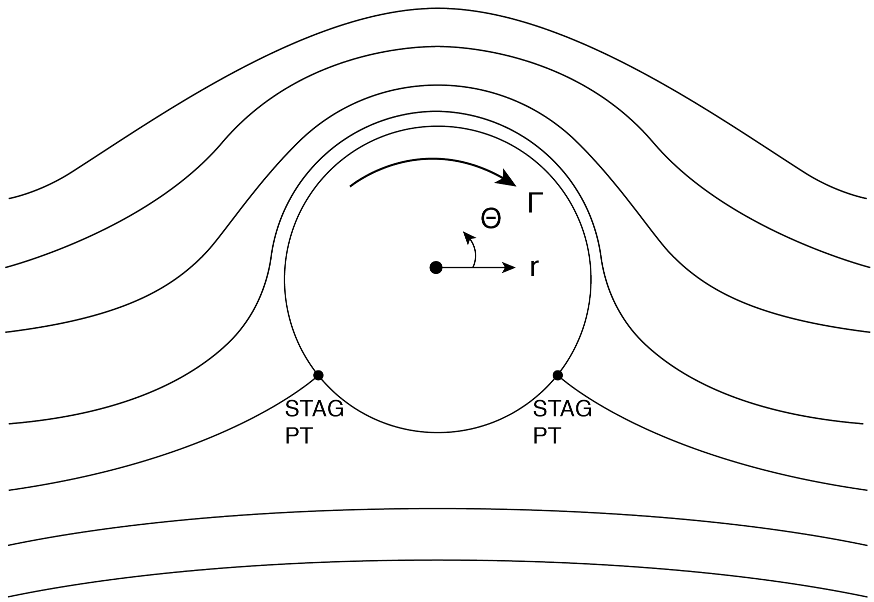


Fig 5.7 Streamlines for flow over a rotating cylinder formed from uniform flow, a doublet and a vortex; note that the rotation direction determines whether flow accelerates over the top or bottom of the cylinder.

We can also determine the pressure distribution around the cylinder surface using Bernoulli's equation:

$$P(r = a) \ = P_\infty + \frac{1}{2}\rho U^2 - \frac{1}{2}\rho {v_\theta}^2 - \rho g h \tag{5.28}$$

where h is the local height on the circle above the centerline (datum)

$$h = a \ sin \ \theta$$

We insert Eqn (5.27) for the velocity on the surface, v_θ, into (5.28) to obtain the expression for the local surface pressure. The integral of the pressure around the circle then determines its net force. First, we find the force component in the x_2 direction, this is denoted as the "lift force" per distance into the page, F'_L:

$$F'_L = \ - \oint P(r = a) \ \ sin \ \theta \ a \ d\theta$$

Inserting the expression for P and noting that integrals of odd powers of the sine functions around the entire circle are zero we are left with:

$$F'_L = \ - \rho U \Gamma \ + \ \rho g \pi a^2 \tag{5.29}$$

The last term represents the net body force by the fluid on the volume of the circle (per distance into the page) or the buoyancy force. The lift force is usually defined without the buoyancy force included so we write:

$$F'_L = \ -\rho U\Gamma \tag{5.30}$$

A few things should be stated about this result. First, recall that the viscous force is not included so this is only due to the pressure distribution. Also, when the circulation is zero there is no lift force (there is still a buoyancy force however). We can conclude that the circulation provides the means to create asymmetric conditions around the circle so that a net pressure force occurs. This expression is often called the Kutta-Joukowsky Law who showed this equation to hold for other shapes as well, and is often used in aerodynamics to determine the lift force on two dimensional airfoils based on the circulation associated with the flow around the wing. The sign convention is such that a counterclockwise rotation results in a downward force, and a clockwise rotation results in a upward force for flow along the positive x_1 axis. It is surprisingly accurate for real flows considering the restrictions on its application. This tends to indicate that viscous forces are small at best. It is only accurate for flows that have not separated from the object surface. We will discuss separation when we get into viscous flow affects.

The Kutta-Joukowsky Law (or theorem) as applied to an airfoil requires what is known as the Kutta condition. This is a condition on the flow at the trailing edge that says that the flow exits the airfoil on the top and bottom surfaces of the airfoil with equal velocity and pressure. This implies that the flow does not tend to wrap around the back end or trailing edge of the airfoil and is the boundary condition that allows the calculation of the lift using Eqn. (5.30).

If interested see this website (en.wikipedia.org/wiki/Kutta_condition).

Complex Variables for Potential Flow Analysis

In this section the analysis methods presented for potential flow are expanded using some additional mathematical tools that allows for complex representation of flows and illustrates how complex flows can be analyzed. The flow itself is restricted to the conditions associated with potential flow which allows flows to be evaluated using superposition of the Laplace equation.

Basic Formulation of Complex Variables

For ideal flows we focus on the use of the velocity potential and streamfunction, both of which adhere to the Laplace equation, the former representing the conservation of mass, and the latter indicating irrotational flow. Both can be expressed as:

$$\nabla^2 f = 0$$

where f represents either the velocity potential, ϕ, or the streamfunction, ψ. The introduction of either of these variables to define the flow field basically replace the velocity vector as the variable of interest. In arriving at a representation of the flow using complex variable notation the variable z is defined as:

$$z = x + iy$$

or in cylindrical coordinates as:

$$z = r\ \left(cos\theta + i\ sin\theta\right) = re^{i\theta}$$

Here, i is the traditional representation of $i = \sqrt{-1}$. Next the Cauchy-Riemann conditions for the two variables ϕ and ψ are introduced, which is predicated on the fact that these two variables satisfy the Laplace eqn. and are thus harmonic functions. The Cauchy-Riemann conditions as:

$$u = \frac{\partial\phi}{\partial x} = \frac{\partial\psi}{\partial y}$$

$$v = \frac{\partial\phi}{\partial y} = -\frac{\partial\psi}{\partial x}$$

It should also be noted that based on this definition the functions ϕ and ψ are orthogonal to each other and it is possible to use one or the other to represent the flow.

A new complex function can be defined, whose real and imaginary parts are based on the velocity potential and streamfunction as:

$$F = \phi + i\psi \tag{5.31}$$

The derivative of F in terms of z is defined as:

$$W\left(z\right) = \frac{dF}{dz} \tag{5.32}$$

Also:

$$\frac{dF}{dz} = \frac{dF}{dz}\frac{\partial z}{\partial x} = \frac{\partial F}{\partial x}$$

Consequently, inserting the definition of F we have

$$W(z) = \frac{\partial \phi}{\partial x} + i\frac{\partial \psi}{\partial x} = u - iv \tag{5.33}$$

This result shows that $W(z)$ represents the "complex velocity" of the flow and is determined by the real velocity components u, v.

It is often advantageous to use cylindrical coordinates, expressed as r, θ in two dimensions. The transformation from x, y to r, θ is:

$$u = u_r cos\theta - u_\theta sin\theta$$

$$v = u_r sin\theta + u_\theta cos\theta$$

The expression for the complex velocity is then (by direct substitution and using the identity of $\cos\theta - isin\ \theta = e^{-i\theta}$) :

$$W = (u_r - iu_\theta)\, e^{-i\theta} \tag{5.34}$$

The use of the complex variable representation in terms of F can then be converted back into the physical space velocity components, u, v, through its derivative relative to the variable z. The magnitude of the local velocity vector is found from W by taking the square root of WW^*, where W^* is the complex conjugate of W. Once the velocity is determined then the pressure field is directly determined from the Bernoulli Equation.

Some examples will help illustrate the use of complex variables to represent rather simple flows.

Uniform Flow

A uniform flow of magnitude U in the x direction becomes:

$$\phi = Ux = Ur\cos\theta \ and\ \psi = Uy = Ur\ sin\theta$$

$$F = Ux + iUy = U(x + iy) = Uz$$

$$W = u - iv = \frac{dF}{dz} = U$$

Note that for a flow, V, in the y direction F is equal to $-iVz$.

By tilting the uniform flow (direction of U) by angle α relative to the x axis the general expression for F in terms of z becomes:

$$F = Uze^{-i\alpha}$$

$$u = U cos\ \alpha; v = U sin\ \alpha$$

Source/Sink Flow

A source and sink flow is radial flow from a point and has a strength proportional to its volumetric flow rate such that for any circle centered about the origin of the source or sink, the line integration about the circle yields the volume flow rate per depth into the plane of the circle (recall that we are only dealing with two dimensional flows). A source has a positive strength with flow outward from the center and a sink has a negative strength (flow is inward towards the center). If the volume flow rate per depth is Q' then the strength is designated as $\mu_s = Q'/2\pi$. The equations governing the velocity potential lines and streamlines are:

$$\phi = \mu_s \ln r\ and\ \psi = \mu_s \theta$$

The resulting expression for the complex potential is:

$$F = \mu_s \ln r + i\ \mu_s\ \theta = \ \mu_s (\ln r + i\ \theta) = \mu_s \ln \left(re^{i\theta}\right) = \mu_s \ln z$$

To locate a source or sink at a point other than the origin, say at location zo, we have:

$$F = \mu_s \ln(z - z_o)$$

Vortex Flow

A vortex flow is one with only a circumferential velocity component about the origin. The velocity decays proportional to (1/r), where r is the radial coordinate. Notice that the resultant streamlines and potential lines for this flow are orthogonal to the streamlines and potential lines for a source flow. The integral of the velocity around a closed path that includes the origin is defined as the circulation, Γ. For convenience we chose a circular path of radius r:

$$\Gamma = \oint u \bullet dl = \oint v_\theta r d\theta = 2\pi C$$

Where $C = v_\theta r$ which is a constant since v_θ varies as 1/r. We define $C = \mu_v$ as a measure of the strength of the vortex as

$$\mu_v = \Gamma/2\pi$$

So, noting that velocity potential lines are radial and streamlines are circular (counterclockwise) as:

$$\phi = \mu_v \theta$$

$$\psi = -\mu_v \ln r$$

$$F = \mu_v \theta - i\,\mu_v \ln r = \mu_v\,(\theta - i\,ln\,r)$$

$$F = \;\; - i\mu_v \ln re^{i\theta}$$

Lastly, the complex velocity can be written as:

$$W\,(z) = -i\frac{\mu_v}{z} = -i\frac{\mu_v}{r}e^{-i\theta}$$

The negative sign for this expression for W results in a rotation in the counterclockwise direction, which is typically selected as "positive" rotation. Notice the resemblance to the formulation for the source flow and vortex flow representation keeping in mind the orthogonal condition of ϕ and ψ. It would be a good exercise to generate plots of the above equations for velocity potential and streamfunction.

There may be some concern that a vortex, which contains vorticity can be considered as irrotational flow. This can be explained as follows. If one were to form a contour anywhere in the flow that does NOT contain the origin (the location of the vortex center) then it can be shown that the circulation is zero. One could conclude that all of the vorticity is located at the center, representing a singularity. The flow driven by this singularity is indeed irrotational. Another way to establish this is to form two contour circles both with centers at the vortex origin but of different radii. It is easily shown then that the difference of the circulation between these two is zero, concluding that for arbitrarily selected contour circles there is no rotational flow between them – so the flow must everywhere be irrotational except at the center.

Flow in a Sector

A sector is defined here to be a region in space near the intersection of two lines, as in a corner with an arbitrary angle. There are general formulations for flows in the region of a sector that can be written in as:

$$F = Az^n = Ar^n\,(\cos n\theta + i\sin n\theta) \tag{5.35}$$

Where "n" represents a parameter to be specified, usually as a constant for a given flow, and "A" is a constant for a specific flow, and shown below to be proportional to the velocity far from the sector, representative of the flow into the sector. An example of this class of flows is uniform flow where $A = U$ and $n = 1$. Note that uniform flow can be thought of as flow over a flat surface that is parallel to the uniform flow direction (there are no viscous forces so the no slip boundary condition does not hold). The flat surface can be thought of as a sector with an intersection of two lines with the lines being parallel. Or, the angle between the two lines is π. The general expression above corresponds to the following expression for velocity potential and streamfunction:

$$\phi = Ar^n \cos(n\theta)$$

$$\psi = Ar^n \sin(n\theta)$$

Using these two expressions we can easily locate lines of constant streamfunction can be found, and in particular when $\psi = 0$: when $\theta = 0$ and π/n. The flow between these two radial lines represents flow in a "sector", as seen in the Fig. 5.8, below.

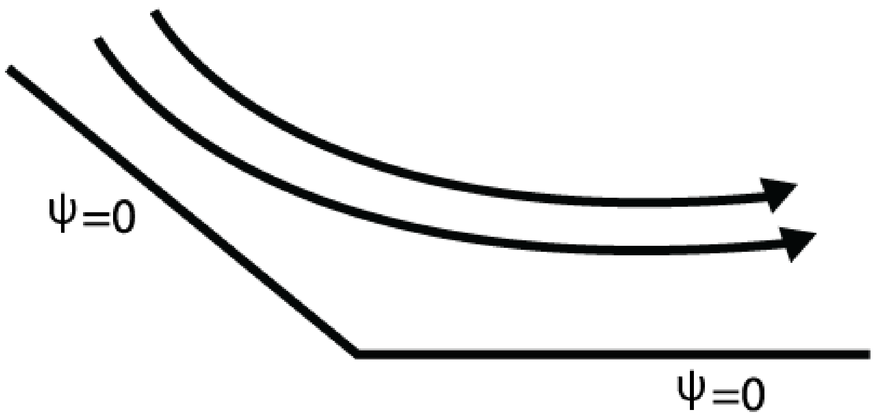


Fig 5.8 Two-dimensional flow in a sector of arbitrary angle between two straight lines; the streamfunction is chosen to be zero along these two lines.

Using the definition of the complex velocity potential, W:

$$W = \frac{dF}{dz} = nAz^{n-1} = nAr^{n-1}e^{i(n-1)\theta}$$

Expanding the exponential reveals that the velocity components in r, θ coordinates are:

$$u_r = nAr^{n-1}\cos(n\theta)$$

$$u_\theta = nAr^{n-1}\sin(n\theta)$$

By inserting different values for the parameter "n" different flows can be simulated. Some examples are given below.

$n = 1:$
$u = A$ and $v = 0$ (uniform flow)
$n = 2:$
$u = 2Ar\cos(\theta)$
$v = 2Ar\sin(\theta)$
or in Cartesian coordinates: $u = 2Ax;\; v = 2Ay$
$\psi = -Ar^2\sin(2\theta) = -Axy$

or for a constant streamfunction (along a streamline) we have:

$$xy = C$$

(where C is a constant determined by the value of ψ at a given location). This latter flow, for a range of values of C yields flow into a right angled corner where the location $(0, 0)$ is the position of the corner. Note that $\psi = 0$ along the axes. The reader is encouraged to plot this function for different values of streamfunction to visualize the flow.

The corresponding flow into a corner of arbitrary angle, α, can be expressed as:

$$F = Az^{\pi/\alpha}$$

Note that Az^n is equivalent to $Ar^n[cos(n\theta) + i\;\; sin(n\theta)]$ which can then be restated in terms of $\pi/\alpha = n$, so that for $\alpha = \pi/3$ we have $n = 3$. This then relates n to the turning angle of the corner flow. For instance, if $n = 3/2$ we have an angle of $2\pi/3$ which is greater than 90°. We can also have flow over a flat plate parallel with the flow if $n = 1$.

A wedge flow for $\alpha > \pi$

Wedge flow resembles the flow that divides at the front side (or leading edge) of an object and is a reasonable model for the flow over the leading portion of certain objects. This is illustrated in Fig. 5.9 where flow impinges on an object with a stagnation point at the wedge vertex and then accelerates along the top and bottom surfaces as the streamlines converge. With $\alpha > \pi$ then $n < 1$.

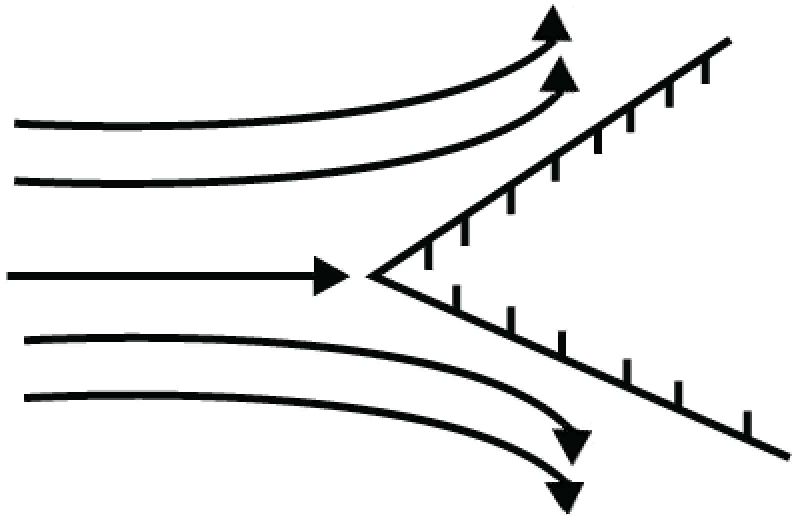


Fig 5.9 Illustration of a wedge flow with a stagnation point at the vertex; along the top and bottom surfaces the flow accelerates resulting in lower pressures on either side of the wedge.

An extreme for wedge flow is flow around a sharp edge, shown in Fig. 5.10 below. Here the flow is moving parallel with a surface, reaches a sharp edge of the surface, and then flows around the edge and then back parallel with the surface. The streamfunction is typically given the value of zero on the surface, and the surface is infinitely thin and flat.

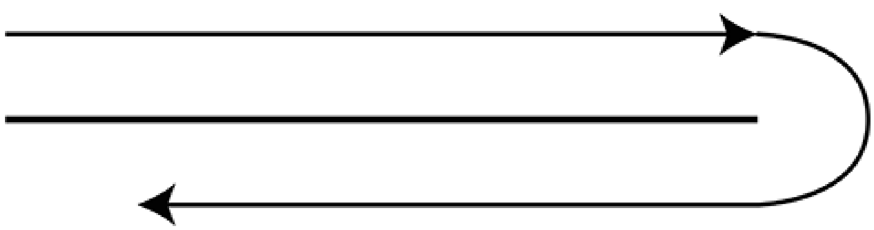


Fig 5.10 Illustration of flow over a flat surface with a trailing edge.

This flow is represented mathematically in terms of a $F(z)$ and constant, $A, n = 1/2$ as:

$$F(z) = A\, z^{1/2}$$

$$F(z) = \; A\, r^{1/2} e^{i\frac{\theta}{2}}$$

$$\psi = Ar^{1/2} \sin \frac{\theta}{2}$$

This expression for the streamfunction can be shown to be equal to zero along the surface for $\theta = 0, 2\pi$. The velocity is obtained from the streamfunction to be:

$$u_r = \frac{1}{r}\frac{\partial\psi}{\partial\theta} = \frac{A}{2r^{1/2}}\cos\frac{\theta}{2} \qquad u_\theta = \ - \frac{A}{2r^{1/2}}\sin\frac{\theta}{2}$$

it can be verified that for $0 < \theta < \pi : u_r > 0$ and $u_\theta < 0;$ and $\pi < \theta < 2\pi : u_r < 0$ and $u_\theta > 0$. This assures the flow reversal around the edge.

Using the flows above written in terms of the constant A we have assumed A to be a real number, but this is not required. We can express this as a complex number by including $e^{-i\beta}$ as a new term: $Ae^{-i\beta}$, where in this expression A is a real number and β is a constant not the same as α used above to define the turning angle of the flow. The result is

$$F = Ae^{-i\beta}z^n = Ar^n e^{i(n\theta-\beta)} \tag{5.36}$$

Noting that the real part of F is the velocity potential and the imaginary part is the streamfunction we have:

$$\phi = Ar^n\cos(n\theta - \beta)$$

$$\psi = Ar^n\sin(n\theta - \beta)$$

This streamfunction expression shows us that when using the complex representation for the constant multiplying z^n there is introduced a rotation of the streamlines through angle β using the same coordinate system when compared to the streamfunction when the coefficient is real (not complex).

Doublet

Next, we introduce the concept of a doublet using complex notation (the superposition of a source and sink brought together in the limit as the separation distance goes to zero – but never actually reaches zero separation). This can be expressed as:

$$F = \frac{\mu_D}{r}e^{-i\theta}$$

$$\phi = \frac{\mu_D}{r}\cos\theta$$

$$\psi = \frac{\mu_D}{r}\sin\theta$$

where μ_D is a constant representing the strength of the doublet. Lines of constant streamfunction $\psi = B$ become:

$$\psi = B = \frac{-\mu_D \ r \ sin\theta}{r^2} = \frac{-\mu_D y}{(x^2 + y^2)}$$

$$x^2 + (y + \frac{\mu_D}{2B})^2 = \left(\frac{\mu_D}{2B}\right)^2$$

This last equation illustrates that we have an equation of a circle with the center located along the y axis with the position changing depending on the value of B, which implies different streamfunction values and therefor different streamlines for each B. The radius of each circle is determined by ($\mu_D/2B$) which depends on the value of the streamfunction. Also, each circle is tangent to the x axis at $x = 0$, or the line with $\theta = 0$ at $r = 0$. See the sketch below in Fig. 5.11 of streamlines.

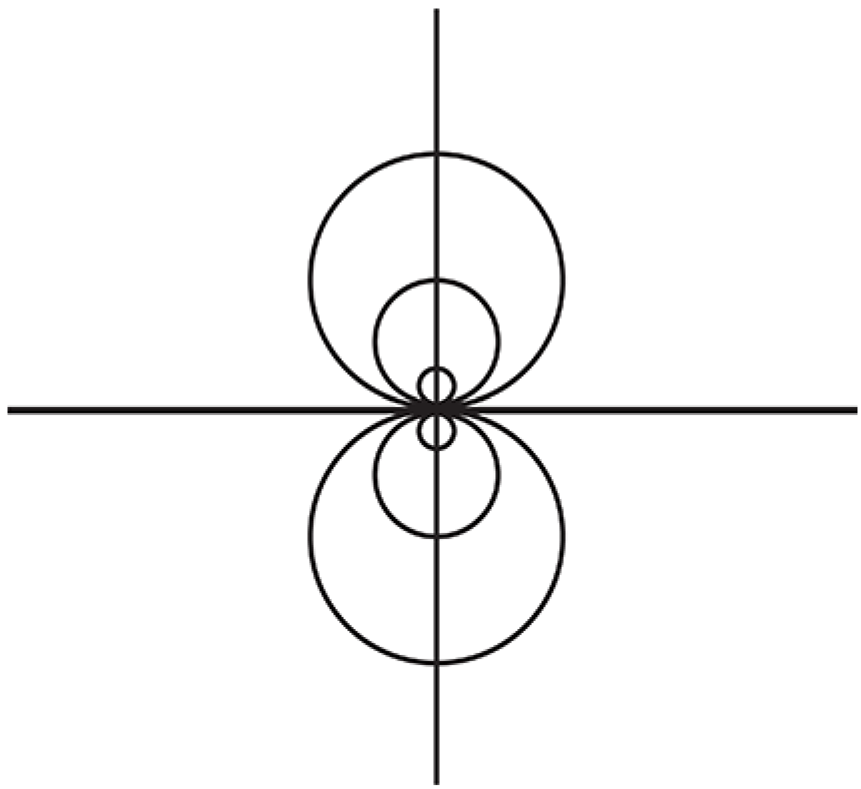


Fig 5.11 Illustration of streamlines associated with a doublet.

Rankine Half-Body

More complicated flows using complex variable representation based on the superposition of simpler flows can be formed with any proper representation of the simpler flows. An example is the Rankine half-body. In some ways this has characteristics similar to the wedge flow identified above in that we are interested in the flow approaching an object and the initial region near the leading edge. However, rather than flat plates forming a corning with a sharp point we have the simulation of flow over a rounded leading-edge surface. In this case we superimpose a uniform flow with a source located at the origin of the coordinate system, both expressed using complex variables. Here we have:

$$F = Uz + \mu_s \ln z$$

To obtain the velocity components in the (r, θ) coordinates we use the definitions of the velocity potential (or we could use the streamfunction):

$$u_r = \frac{\partial \phi}{\partial r} = U cos\,\theta + \frac{\mu_s}{r}$$

$$u_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = -U sin\,\theta$$

At the very front, or leading edge of the body, the velocity should become zero, representative of a stagnation point. So, setting each velocity component equal to zero we can determine where the stagnation point should occur.

$$u_r = 0: \quad U cos\,\theta = -\frac{\mu_s}{r}$$

$$u_\theta = 0: \quad U \sin\theta = 0$$

There are two solutions to the above set of equations: $\theta = 0$ and $\theta = \pi$. However, for $\theta = 0$ the value of r becomes negative from the first equation, which is an invalid condition so the only realistic solution for the location of the stagnation point is $(\mu_s/U, \pi)$. By increasing the source strength, $\mu_s v$, the stagnation point moves further upstream in the flow. The streamfunction that passes through this point is found by noting that $u_\theta = -\dfrac{\partial \psi}{\partial r}$ such that integrating and setting $\theta = \pi$ then $\psi_{stag} = \mu_s \pi$. Since the streamfunction passing through the stagnation point is on the body then a general streamfunction expression for the body becomes

$$\psi_{body} = U r_{body} \sin\theta_{body} + \mu_s \theta_{body}$$

In this expression the subscript "body" represents r, θ coordinates that are on the body which will be a constant value of the streamfunction. A plot of a constant value of $\psi_{body} = \mu_s \pi$ (which is specified by the stagnation point given above) can be used to determine the body surface. In general, we have:

$$r_{body} = \frac{\mu_s(\pi - \theta_{body})}{U \sin\theta_{body}}$$

One can imagine that for increasing r in the first or fourth quadrant the influence of the source will diminish (since we are moving away from the source). Consequently, the flow is expected to eventually become streamlines that are again parallel with the oncoming flow. For the streamline of the body, in the limit of large r we can write an expression for the total width of the body, w_{body} :

$$w_{body} = \frac{2\pi\mu_s}{U}$$

By increasing the source strength for a given freestream velocity U the width of the body increases and it is possible to model flow over the object of any desired width.

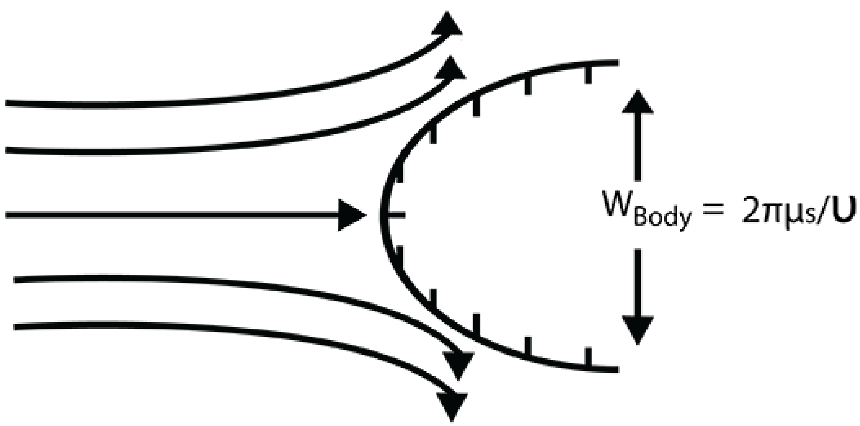


Fig 5.12 Flow over a Rankine half-body.

Enclosed Bodies

It is possible to continue on with the idea of flow over an object and create a completely enclosed object (this requires an enclosing streamline such as a circle or ellipse) to simulate flow over the enclosed object. The streamlines inside the body are of no interest here, just those flowing around the outside of the body and how the velocity and pressure vary. As an example, we simulate the flow over a circular cylinder. This will involve the superposition of a uniform flow and a doublet at the origin. This can be expressed in our complex representation as:

$$F(z) = Uz + \frac{\mu_D}{z}$$

The strength of the doublet for a given flow U will determine the radius of the cylinder. Consider a desired radius a for a given uniform flow U. We can write the complex potential for the circle as:

$$F(z) = Uae^{i\theta} + \frac{\mu_D}{ae^{i\theta}}$$

Noting that the streamfunction is the imaginary part of this expression which yields:

$$\psi = \left(Ua - \frac{\mu_D}{a}\right)\sin\theta$$

We can force the value of the streamfunction to be $\psi = 0$ at $r = a$ by setting the strength of the doublet to $\mu_D = Ua^2$. Consequently, we end up with flow of velocity U over a cylinder of radius a. We insert the strength of the doublet into the expression for $F(z)$:

$$F = U(z + \frac{a^2}{z})$$

Again, we are only interested in the flow outside of the cylinder, as shown in Fig. 5.13.

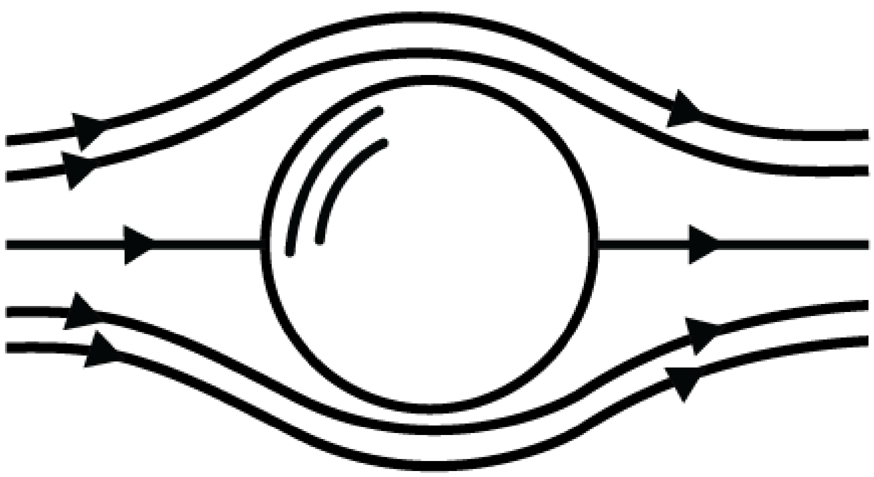


Fig 5.13 Flow over a sphere; note symmetry of streamlines about the x axis and y axis.

As we will see later, it is possible, through a proper transformation of the flow over a cylinder, to obtain the flow over objects, such as an airfoil. However, to make this more physically correct we will want to introduce circulation to the flow to simulate the circulation that occurs for an actual airfoil that provides the lift force experienced by the airfoil. Recall from a vortex flow that circulation is based on circumferential flow, with strength proportional to the u_θ velocity component. Superimposing a uniform flow over a cylinder with radius a, with a vortex of strength μ_v rotating in the clockwise direction, results in the following:

$$F = U\left(z + \frac{a^2}{z}\right) + i\mu_v \ln z + c$$

where we add the constant c, this will allow us to assign the value of the streamfunction to the cylinder surface, at $r = a$. By setting $z = ae^{i\theta}$ to be on the cylinder surface the value of the complex potential can be evaluated and thereby obtaining the streamfunction value in terms of the unknown constant c. By setting $c = \; - i\mu_v \ln a$ it is straightforward to show that the streamfunction is in fact equal to zero for all θ at $r = a$. Then using this value for c and combining c with the second term (combining the ln terms) results in the following expression for the flow over a cylinder with circulation:

$$F = U\left(z + \frac{a^2}{z}\right) + i\mu_v \ln\frac{z}{a}$$

Note that the above expression is for circulation with clockwise rotation, for counterclockwise rotation the ln term is negative.

The velocity field associated with the above expression for $F(z)$ is:

$$u_r = U\left(1 - \frac{a^2}{r^2}\right)\cos\theta$$

$$u_\theta = -U\left(1 + \frac{a^2}{r^2}\right)\sin\theta - \frac{\mu_v}{r}$$

Notice that it is the vorticity that provides the circumferential flow component. Also, along the surface of the cylinder the velocity is not necessarily zero. In fact, since this is a streamline then the value of u_r must be zero so no flow crosses the cylinder, but u_θ is only zero at two points. These points are stagnation points because both velocity components are zero resulting in zero velocity. The locations of these two points are found by setting $u_\theta \; = 0$ and solving for θ:

$$\sin\theta_{stagnation} = \frac{\mu_v}{2Ua} = \frac{\Gamma}{4\pi Ua}$$

If the circulation is zero ($\mu_v \; = 0$) then the stagnation points are at $\theta = 0, 2\pi$, implying symmetric flow both relative to the x and y axis. See Fig. 5.14 for representation of stagnation points for different circulation conditions. For clockwise rotation greater than zero stagnation points will move to the third and fourth quadrants. For counterclockwise rotation, the stagnation points will be in the first and second quadrants. When the strength of the clockwise circulation is large enough to result in $\theta = 3\pi/2$ and the two points coincide on the “bottom” of the cylinder. For larger circulation strengths than this the stagnation point actually moves off of the cylinder.

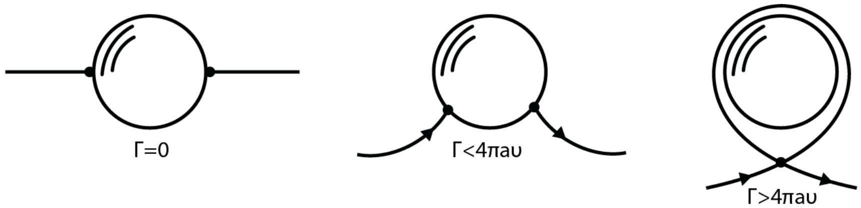


Fig 5.14 Stagnation points on a cylinder with different circulation, [latex]\Gamma[/latex].

At this point it is instructive to evaluate the consequences of the added circulation to flow over a cylinder. As we have seen from above, the circulation removes symmetry across the x axis within the flow. That is the streamline distribution for the third and fourth quadrants is different from the first and second quadrants of the flow. However, this circulation does not change the symmetry that exists across the y axis (the flow in the first and fourth quadrants is symmetric with the second and third quadrants). Recall from the Bernoulli equation (viscous forces are ignored and we have a steady flow) that the pressure field will retain a similar symmetry with the velocity field. The symmetry across the y axis indicates the pressure on the "front" half of the cylinder will be identical to that on the "back" half of the cylinder and the net force in the x direction will be zero. However, across the x axis the loss in velocity symmetry implies a difference in pressure distribution on "top" of the cylinder versus the "bottom". Also, for circulation with a clockwise rotation the velocity on top will be greater than that on the bottom resulting in lower pressure on the top. The net effect is an upward force on the cylinder by the pressure field. If the rotation is reversed there would be a net downward force on the cylinder.

The determination of the force on a body with external flow around it in potential flow can be determined from the Kutta-Joukowski law which relates the net lift on the body to the circulation, Γ, generated by the flow and the magnitude of the freestream velocity, U, as:

$$Lift/span = -\rho U \Gamma$$

This relationship is shown in an earlier part of this chapter. It is also shown below in the context of conformal mapping. In this formulation, if the circulation is negative (clockwise) then the force direction will be positive (upward lift) when there is positive flow over the body (left to right). This can be derived for any shaped two-dimensional body with an associated circulation calculated from the line integral about an enclosed area that includes the body. This law can be shown through the use of Newton's law relating total force (lift plus pressure) to rate change of momentum and the integration of the pressure field obtained from the Bernoulli equation for steady two-dimensional, irrotational inviscid flow. We show this also in the next section. The interested student can review the derivation of this from many references or online (en.wikipedia.org/wiki/Kutta–Joukowski_theorem) using the basic tools of complex representation of the velocity field.

For the circular cylinder case the circulation is specified within the strength of the vortex and is used in conjunction with the freestream velocity to specify the flow conditions associated with a given freestream velocity and cylinder radius.

Conformal Transformations

As mentioned previously it is possible to transform the results for flow over a cylinder with circulation to that of flow over an airfoil, allowing the determination of the lift force through the solution to the magnitude of the lift force for flow over the cylinder. The idea of a transformation in this sense is that by altering the coordinate system it is possible to change from a circular geometry to a different geometry. Once this transformation is specified it is possible to calculate the flow at certain points within the circular geometry and assign the flow results to points in the new, different geometry.

Consider a mapping function $\zeta = \zeta(z)$ which establishes a relationship between coordinates ζ and z (where again $z = x + iy$). Consequently, a known solution expressed in ζ can then be transformed into the z coordinate. The condition for conformal mapping in this sense is that the velocity potential (which satisfied the Laplace equation in the original coordinate must also be satisfied in the new coordinate. The other condition is that new coordinate must satisfy the Cauchy-Riemann equations. These state the following where the new coordinates are, say, (ξ, η) and the original coordinates are (x, y):

$$\frac{\partial \xi}{\partial x} = \frac{\partial \eta}{\partial y} \; and \; \frac{\partial \xi}{\partial y} = -\frac{\partial \eta}{\partial x}$$

Under these conditions it can be shown that the complex potential, $F(z)$, in a given coordinate which determines the velocity potential and streamfunction, can be shown to satisfy the Laplace equations for the velocity potential and streamfunction in the ζ plane. The local solutions in the z plane are found from the solutions in the ζ plane by using the relationship given by $\zeta = \zeta(z)$. In other words, a solution at a point in ζ has the same value at the corresponding point in z. In additional to this, if we take our previous definitions for $W(z)$ we can write:

$$W\left(z\right) = \frac{dF(z)}{dz} = \frac{d\zeta}{dz}\frac{dF(\zeta)}{d\zeta} = \frac{d\zeta}{dz}W(\zeta)$$

This says that the mapping function derivative with respect to z defines the relationship between the complex velocity solutions in each coordinate frame. Lastly, it can also be shown that the strengths of the various flow elements used within one frame are not altered in the transformed coordinate frame. These relationships now allow one to solve a given problem, such as flow over a cylinder with circulation, and interpret the results in another coordinate frame which may change the geometry of the object.

As an example, consider using the Joukowski transformation:

$$z = \zeta + \frac{C^2}{\zeta}$$

with C being a constant that determines the shape of the boundary in transformation. For instance a circle of radius a given by $\zeta = ae^{i\theta^*}$ while setting $C = a$ results in:

$$z = a\left(e^{i\theta^*} + e^{-i\theta^*}\right) = 2a\cos\theta^*$$

and the resultant z plane shape will be a flat plate extending from $-2a$ to $+2a$ along the x axis in the z plane. Note that here, $\theta*$ is used to represent the angle coordinate in the ζ plane (not z plane). This illustrates a transformation between a circle, in ζ, to a flat plate in z. Here we are interested in points outside of the radius of the circle and how they transform to points in the z plane.

Continuing this, by noting that the stagnation points in the ζ plane are determined by the circulation, as shown previously for flow over a cylinder, then when the circulation is zero the stagnation points are at $\theta* = 0, \pi$. The stagnation points in the z plane then are found through the transformations as at $x = \pm 2a$, or at the ends of the flat plate.

The more interesting case is when we have a flat plate with the freestream flow at some angle, a, to the x axis. This is usually defined to be the angle of attack. When this occurs a stagnation point will form on the "bottom" side of the plate when the freestream velocity has a positive angle relative to the x axis (which forms the freestream velocity direction). This is shown in fig. 5.15, below. At this stagnation point the flow separates, with part of the flow moving forward in the positive x direction and some of the flow moves in the negative x direction. The latter flow then must turn around the corner or edge of the flat plate as shown in the figure. Also, the trailing edge will have a relative movement of the stagnation point away from the trailing edge of the plate, resulting in flow having to go around the trailing edge and up along the surface. Since this flow is inviscid the potential flow velocity around an infinitely thin plate must go to infinity since the radius of curvature is zero. This does not happen in a real flow due to viscous effects slows the fluid and resulting in a finite radius of curvature and flow separation at the edge. In a real airfoil the leading edge has a finite thickness and can potentially eliminate flow separation unless the angle of attack is too large. But the trailing edge is typically very thin. The resulting flow condition preventing separation at the trailing edge is the Kutta condition imposed at the trailing edge. See the earlier discussion of the Kutta condition. The idea is that the flow adjusts itself to prevent this separation by imposing local circulation near the trailing edge to offset the separation flow. If circulation is added to the flow to move the stagnation point to the trailing-edge then the flow at the trailing edge will have equal velocity on the top and bottom surfaces of the plate (and by Bernoulli equation equal pressures) such that the flow will leave the trailing edge smoothly and not wrap around the trailing edge.

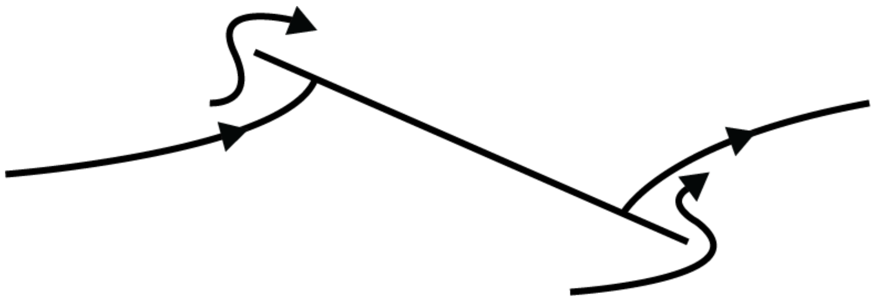


Fig 5.15 Stagnation points for flow over a flat plate at a nonzero angle from the freestream flow.

Considering flow around a cylinder in the ζ plane at an angle of attack of α relative to the x axis, or $\theta = 0$. The uniform freestream must be rotated by angle α, and this results in the following representation of the complex potential:

$$F(\zeta) = U\left(\zeta e^{-i\alpha} + \frac{a^2}{\zeta e^{-i\alpha}}\right) + i\mu_v\ ln\frac{\zeta}{a}$$

However, we wish to impose the Kutta condition as well. To do this we must add circulation equivalent to moving the stagnation point to the trailing edge, or $z = 2a$. For this value of z the corresponding value of ζ is a, which is on the cylinder at $(a, 0)$. To achieve this additional rotation of the flow by angle α in the ζ plane is required. We use the equation for the stagnation point position, $\theta_s tag$, to see that the required circulation is $\Gamma = 4\pi U a sin\ \alpha$.

So inserting this value of circulation into the vortex strength above we obtain:

$$F(\zeta) = U\left(\zeta e^{-i\alpha} + \frac{a^2}{\zeta e^{-i\alpha}}\right) + i2Ua\ sin\alpha\ ln\frac{\zeta}{a}$$

This can be written in terms of the z plane with $\zeta = \frac{z}{2} + \sqrt{\left(\frac{z}{2}\right)^2 - a^2}$

as:

$$F(z) = U\left(\left(\frac{z}{2} + \sqrt{\left(\frac{z}{2}\right)^2 - a^2}\right)e^{i\alpha} + \frac{a^2 e^{i\alpha}}{\left(\frac{z}{2} + \sqrt{\left(\frac{z}{2}\right)^2 - a^2}\right)} + i2a\ sin\alpha\ ln\left(\frac{1}{a}\left(\frac{z}{2} + \sqrt{\left(\frac{z}{2}\right)^2 - a^2}\right)\right)\right)$$

Noting that the imaginary part of $F(z)$ represents the streamfunction. Fig. 5.16 below represents the flow in both the z and ζ planes.

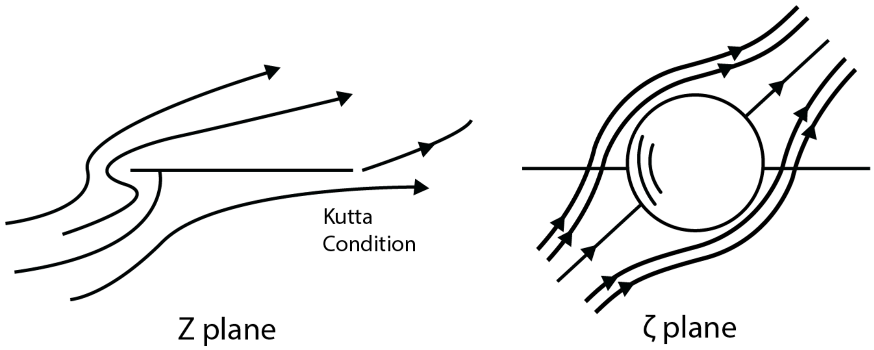


Fig 5.16 Transformation between flow over a flat plate and flow over a cylinder.

We can easily determine the lift force with the known circulation to be:

$$L = \rho U \Gamma = 4\pi\rho U^2 a \sin\alpha$$

Expressing this nondimensionally, as a lift coefficient by dividing by ($1/2\rho U^2 c$) where c is the length of the plate (the chord length) which is equal to $4a$ we obtain:

$$C_L = 2\pi\sin\alpha$$

This is the theoretical lift coefficient for a thin (flat) airfoil at an angle of attack of α, as was shown previously in this chapter. For small angles of attack, we have $C_L = 2\pi\alpha$.

Blasius Theorem and Lift Force for an Arbitrary Body

The lift force for a rotating cylinder in a uniform inviscid flow is given by $L = -\rho U \Gamma$ (where the circulation is positive if counterclockwise and flow is from left to right). What is rather amazing is that the object shape is not critical in the use of this same equation as long as the value of the generated circulation is determined properly.

The Blasius Theorem (also referred to as the Blasius Integral Laws), is a means to obtain the total force on the object within a flow. Ideally the surface velocity distribution would be found, then using the Bernoulli equation the pressure along the surface is found, which could be integrated to find the net force on the object. However, this procedure can be circumvented through the use of the Blasius Theorem.

At any location on the object surface the expression for the local force, dF, in terms of the velocity potential, can be found. Specifically, the force is decomposed into drag and lift components, noting that drag is in the negative x direction, as:

$$dF = dF_D - idF_L = -Pdy - iPdx = -iP(dx - idy)$$

Now to extend this around the entire object it is needed to integrate around the closed path forming the object (by sign convention we go counterclockwise) as:

$$F_D - iF_L = -i\oint Pdz^*$$

$$z^* = x - iy$$

Where $z*$ is the complex conjugate of z. The pressure is determined from the Bernoulli equation with the surface velocity written as

$${U_s}^2 = [(u+iv)(u-iv)]_s$$

$$u + iv = ({u_r}^2 + {u_\theta}^2)^{1/2}e^{i\theta}$$

$$dz = |dz|e^{i\theta}$$

The last two expressions show that the velocity and surface are parallel with each other, both at the same angle, as must be the case for a solid surface in inviscid flow. So, we can show that $(u + iv)dz*$ is a real number (using the fact that the velocity slope equals the surface slope, or $v/u = y/x$, so the imaginary part goes to zero) and that $(u + iv)dz* = (u - iv)dz$. Finally, the velocity is determined by:

$$W = (u - iv)$$

So, by inserting for the pressure in terms of the velocity from the Bernoulli equation, and using the above express for U_s along with the relationship for W in terms of the velocity components it can be show after a bit of manipulation that:

$$F_x - iF_y = \frac{i\rho}{2}\oint W^2 dz$$

The above expression is valid for steady, potential flow and often called the "Blasius Theorem". Notice that the drag component is the negative of the real part of the right hand side and the lift component is the negative of the imaginary part.

In potential flow the integration around any closed contour (say a contour around the surface of a body versus a contour around the body far from the body itself) can be shown to be the same. For this to be true there can be no singularities that occur between the contours. So, in this case no sources or sinks or vortices, etc. can exist between the surface contour and a contour far from the surface. Flow around an arbitrarily shaped object may be generated by putting a distribution of singularities, such as vortices, around the shape contour and adjust their strength distribution to form a closed contour of some desired shape. This is discussed in detail in the next chapter. In the flows considered here there are no singularities outside of the body itself. So, we can take a contour far from the body out into the free stream. All of the distributed singularities will appear, from far away, as if they are all located at the coordinate system origin (assumed to be located near or in the object-although this may not be necessary). The superposition representation of the complex potential can be written by eliminating the small distances from singularity location and the origin. One can then insert the complex velocity associated with the superposition into the above equation for force. The unique aspect here is that as the contour moves further and further away from the body the complex velocity can be simplified by dropping terms that are small, such as for a vortex we have terms like $i\Gamma/2\pi z$ and for a doublet we have μ_D/z^2. Note that here a positive circulation is designated as clockwise, as is typically done in aerodynamic applications. If we only retain terms ~$1/z$ and perform a series expansion then by complex variable theory the contour integral is equal to $2\pi i \sum residuals$.

The residuals are determined by taking the square of the complex velocity as shown in the integral above, and dropping terms higher than say $1/z$. Then the residual is the coefficient of the $1/z$ term which is multiplied by $2\pi i$ to obtain the value of the integral. The real part is F_D and the negative of the imaginary part is F_L.

This can be illustrated using the complex potential for a uniform flow, vortex and doublet (flow over a rotating cylinder): $F = Uz + \mu_v \ln z + \frac{\mu_D}{z}$. Taking the z derivative to form the complex velocity, W, and then squaring this and only retaining up to the $1/z$ terms, the coefficient of the $1/z$ term will be $2i\mu_v$ resulting in the following for the force:

$$F_D - iF_L = \frac{i\rho}{2}\left(2\pi i \frac{(i\Gamma U)}{\pi}\right) = 0 - i\left(\rho U \Gamma\right)$$

This states the there is a net lift force dependent of the circulation and zero drag force and is the identical to the result presented previously. The circulation here is that due to the sum total effect of the distribution of vortices around the object. This result is often called the Kutta-Zhukhovsky theorem.

VI. THE PANEL METHOD: AN INTRODUCTION

The panel method is an analysis method that can be used to arrive at an approximate solution for the forces acting on an object in a flow. The method, as we present it here, is based on inviscid flow analysis, so it is limited to the resultant pressure forces over the surface. The panel method is basically a numerical approximation that relies on using discrete elements on the surface of an object and then prescribing a flow element (such as a vortex or doublet or source or sink) on each element that will satisfy certain boundary conditions (like no flow crosses the surface of the object). The interaction of the elements are accounted for and must also satisfy the condition that far from the object the flow should be equal to the free stream velocity approaching the object. There are a number of books and papers written that describe the method in very general terms and even the inclusion of viscous forces to some degree. But here we are just introducing the method to get a feel for its usefulness in external flows, so we will use a simply geometry with a simply distribution of flow elements. More complicated models exist but they all are based on the simplified form presented here.

We will assume that we have potential flow such that the governing equation for the flow field is the Laplace of the velocity potential, $\nabla^2\phi = 0$. The boundary condition at an impermeable surface, where the velocity normal to the surface is zero, is $\nabla\phi \cdot \boldsymbol{n} = 0$. Also, we can put our frame of reference on the object so fluid flow approaches the object. Keep in mind that since it is inviscid there may be a nonzero velocity component tangent to the surface. Also, the goal is to determine the velocity on the surface, and once this is found the Bernoulli equation can be used to find the local pressure distribution. The pressure can then be integrated over the surface to find the force by the fluid flow.

Without deriving this it can be shown that the following defines the velocity potential at any point P in the flow field (using Green's Identify):

$$\phi\left(P\right) = \frac{1}{4\pi}\int\left(\frac{\nabla\phi}{r} - \phi\nabla\frac{1}{r}\right)\cdot ndS \tag{6.1}$$

where the integral is over the surface area of the flow (assuming two dimensional flow), S. This equation indicates that to solve for the velocity potential we must evaluate the integral on the flow boundaries (both the solid surface and infinitely far away).

This can be shown to be a result of the application of the divergence theorem that says:

$$\int n_j u_j dS = \int \frac{\partial u_i}{\partial x_i} d\forall \tag{6.2}$$

where u_i is the velocity vector. This states that the rate of expansion of a volume of fluid, given by the divergence of the velocity vector, within a given volume equals the flux through the volume boundaries. If the fluid is incompressible then the right side is zero and the net flux through the entire boundary is zero (this is the general argument for the conservation of mass of steady flow).

All this is nice and can be a powerful tool to find ϕ and therefore the velocity in the flow. But we really don't need to find the potential of the entire flow, what our goal is, is to set up a flow that we know satisfies the velocity boundary condition at the surface of the object (with no velocity component across the surface) and far from the surface where the velocity is known to be the freestream velocity. Once this is established the force can be found. That is to say we only need to evaluate the surface velocity and then the pressure on the surface.

The general approach is to select a "grid" which is a series of "panels" that form the surface. Here we take the panels as straight flat surfaces arranged over the real surface. In the limit of very small panels the constructed surface will simulate the actual surface. On each panel we place a distribution of flow elements (like sources, sinks, vortices, etc.) that when combined together will result in a flow field that will satisfy the surface boundary condition. There are lots of ways to identify which elements to use and how they may be distributed on the panels. Here we will use vortex elements, with one placed on each panel. The net flow is the result of superposition of the flow set up by each vortex on each panel element. So at each point in the field we add together the flow caused by all of the panel elements using the superposition rule. The panels that are far away from a given point will have less and less influence on the flow because the strength of the flow caused by a flow element decreases with distance from the element origin. For instance for a "source" the velocity decreases with increasing radial position because the flow is spreading out away from the source. But the influence never really goes to zero.

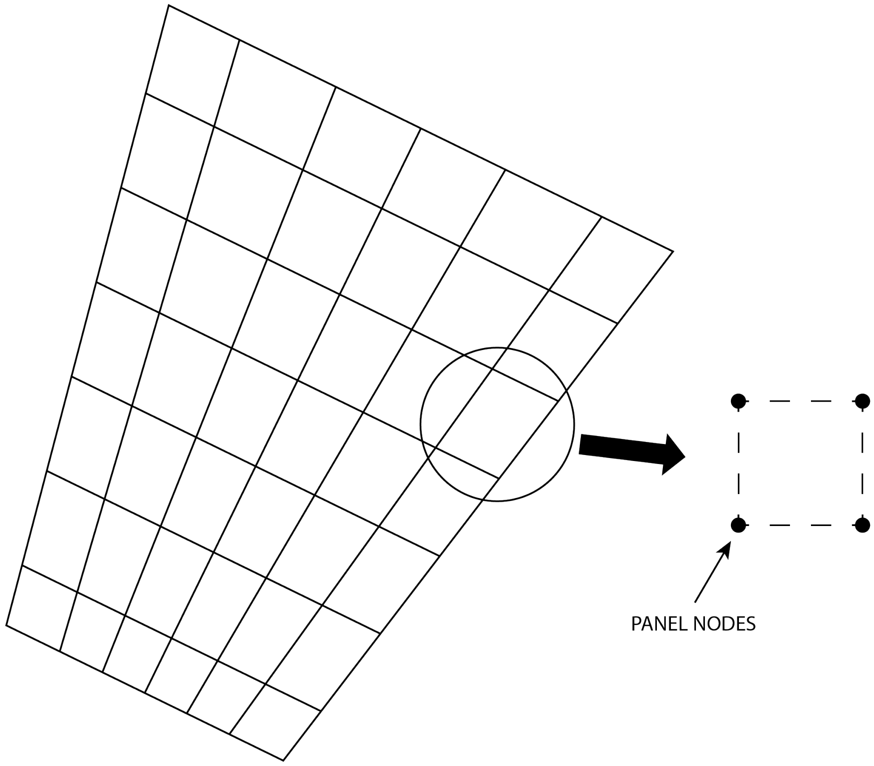


Fig 6.1 Illustration of a panel geometry; any three dimensional shape can be constructed; shown here is a surface of what could be a three dimensional object such as an entire airplane.

In placing a series of panels over the surface we first need to specify the size of each panel. We place a vortex of some strength some where on the panel (whose location is shown below) and we must identify points on the surface where we want to make sure that the velocity is zero across the surface. To be clear, individual points on each panel are used to evaluate the element (vortex) flow field –it needs its own origin, or coordinate system, to write an equation for the flow generated by this element. We also only pick a point on the panel to check to make sure that the net sum of contributions from all elements results in zero flow across the surface. The fact that we only satisfy the condition at one point on each panel will be satisfactory if the panels are made to be reasonably small. These points are called "collocation points" on each panel. In the end each panel will have coordinates that define its location on the surface, coordinates for the element location on the surface and coordinates for the collocation point.

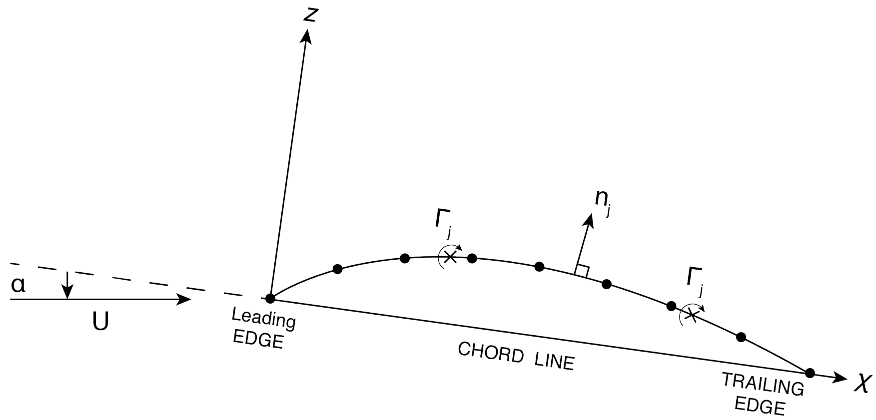


Fig 6.2 Geometry of panels over a two dimensional wing showing individual vortices, Γ_j associated with each panel; n is the outward normal for each panel; the wing is at an angle of attack to the freestream of α.

The method of solution for the force on the object is the determination of the magnitudes or strengths of the elements on each panel. Once we have this distribution of strengths we can calculate the total lift on the surface that results from all of these elements. Recall that the lift experienced by a surface is really the component of the pressure at the surface integrated over the surface area in a direction normal to the approach velocity vector of the flow — it is not necessarily vertical, but normal to the freestream velocity.

For illustration we are going to use flat plate panels with vortex elements, one per panel. To get an idea of how to define the vortex origins on each panel we can examine flow over a single flat plate at some angle of attack to the freestream velocity vector, α.

First we define the "center of pressure", x_{cp}. This is the location on the surface where the resultant lift force caused by the distributed load on the surface acts such that there is no net moment on the surface (see F.M. White, Chapt. 8, for some details on this). In general, if the moment on the surface about the leading edge is Mo, and L is the lift force then using x as the coordinate measured along the flat surface from the leading edge then:

$$x_{cp} = -Mo/L$$

Consider now flow over the same flat surface, the lift we have seen is $L = \rho U \Gamma$ note that we are going to define Γ as positive clockwise — this is opposite to what we have done previously, but it helps get rid of some negative signs, this is not necessary, but convenient. To find out the value of x_{cp} we use the theoretical evaluation of the lift force based on "thin airfoil theory". The interested reader can refer to F.M. White Chapt. 8 for details on this. First, assuming a two dimensional flow, we assume that there is some continuous distribution of vorticity over the surface (in contrast to discrete vortex elements), such that there is a local circulation per unit length of surface, $\gamma(x)$. The surface integral of this distribution results in the net total circulation: $\Gamma = \int_0^c \gamma dx$ where "c" is the length of the flat surface, which we call the "chord".

The Kutta condition is imposed where we want the flow to leave the surface flowing parallel with the back (trailing) edge. If this is a surface like an airfoil we want both the top and bottom flows to leave smoothly from the trailing edge. For this to happen then we don't expect there to be a pressure difference between the top and bottom of the surface at the trailing edge, which would cause the flow to deflect up or down. If there is no velocity or pressure difference across the flow at the trailing edge the local circulation at the trailing edge, $(x = c)$, must be zero. So now we have a boundary condition for the integration of $\gamma(x)$ to get the total circulation Γ, that is $\gamma = 0$ at $x = c$. The needed distribution has been figured out for a single flat surface at an angle of attack of α, the angle between the oncoming flow and the surface. The function that works is: $\gamma = 2U \sin\alpha \, (\frac{c}{x} - 1)^{1/2}$. The integral of this expression for $\gamma(x)$ yields the value of Γ. Once the circulation is known then the lift, L, can be found. Using this lift, and also using the above definition of x_{cp}, it turns out that the "center of pressure occurs at $x_{cp} = c/4$. So by assuming the lift force occurs at x_{cp} then there is no moment caused by the distributed pressure force.

The lift force, L, as calculated as stated above, can be made nondimensional by divided by $1/2\rho U^2 b$, where b is the span of the surface (in and out of the page). The result is an expression for the lift coefficient:

$$C_L = 2\pi \sin\alpha \tag{6.3}$$

This result is the thin airfoil theory result for the lift on a surface at angle of attack, α. So at this point, we take this result for a flat surface and place vortex elements at the center of pressure for each flat element, which will be at a point ¼ of the distance along the panel from the beginning edge of the panel. Although this is not absolutely required for the panel method it is a convenient choice. As panel size is reduced smaller and smaller the impact of this choice becomes small.

The next step is to find the location where we want to impose the condition of zero velocity crossing the surface for each panel. For this we are using a coordinate system for the panel to be (x, z) where x is, again, along the flat surface from the panel leading edge and z is normal (upward) from the surface. To find these collocation points, we note that the the velocity set up by a vortex element at location r from the origin is $v_\theta = -\Gamma/2\pi r$. Writing this in Cartesian coordinates where at the panel surface $v_\theta = \; w$ (where w is the z component of velocity) is:

$$w = -\frac{\Gamma}{2\pi} \frac{(x - x_o)}{(z - z_o)^2 + (x - x_o)^2} \tag{6.4}$$

where (x_o, z_o) is taken as the center of pressure (where the vortex element is located). Also, since the collocation point is on the surface then both z and z_o are zero.

To get the total normal component velocity at the panel surface we add together w being the velocity normal to the surface induced by the vortex, with the component of the freestream flow normal to the surface which is at angle α to the surface, $U \sin \alpha$, and set this to zero:

$$w + U\ sin\ \alpha = 0 \tag{6.5}$$

We next insert Eqn. (6.4) for w, with $z_o = 0$ and $x_0 = c/4$. And we define the location at which we evaluate the zero normal velocity component as the collocation point as: $x = kc$. Finally we solve for "k" to be $k = 3/4$. This says that the x location along any given panel where the boundary condition is to be satisfied is at $x = 3\ c/4$. Recall that the vortex element is located at $x_o = c/4$.

In summary, once the panels are set up on the surface, each has its own angle of attack, α_ρ, depending on the local orientation of the surface. Each panel has a vortex element located at its own center of pressure and the normal velocity must be equal to zero at $x = 3\ c/4$ measured from the front of each panel. An equation can be written for the zero velocity condition at the collocation point of a panel by taking the sum of all of the contributions from all of the panels, each with a different vortex strength Γ_ρ, and setting this sum to zero. This will result in N equations (one for each panel) with N unknowns for Γ_ρ for each panel. We can then solve for all Γ_ρ using this set of equations.

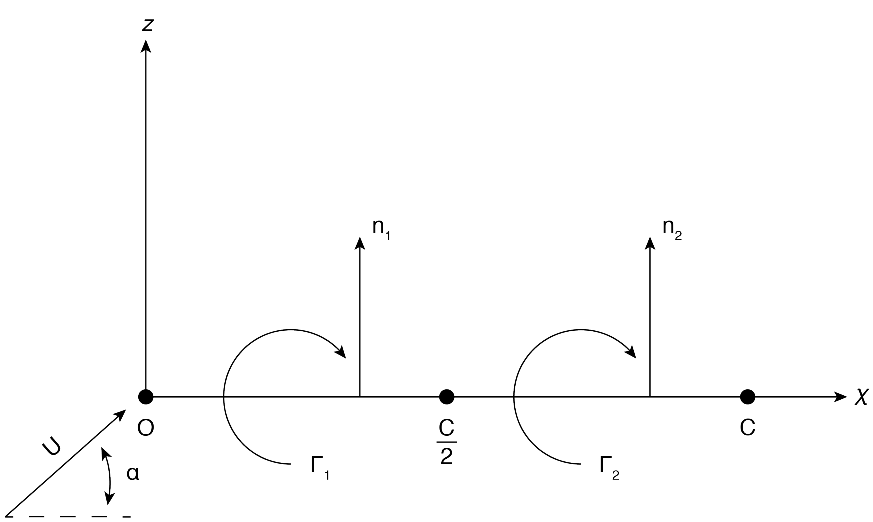


Fig 6.3 A simple example of two panels along a flat surface of length c and angle of attack of α.

Now we can set this up for a simple example to help pull all of this together, see Fig. 6.3. We use a thin flat surface representing a flat airfoil at an angle of attack of α, with a uniform approaching flow of U. We use two panels for illustration. Note that the entire length of the two panels together is "c", NOT the length of each panel. Also, x and z, used previously for a single panel, are along and normal to a line from the start to the end of the surface, respectively. This provides a coordinate system for the "system" of panels. For this simple case the angle of attack for both panels are the same, but we allow individual panel vortex strengths, Γ_1 and Γ_2.

In determining the vortex location for each panel, using the overall coordinate system we get (c/8,0) and (5c/8,0) for panel 1 and panel 2, respectively. The collocation points are (3c/8,0) and (7c/8,0), respectively.

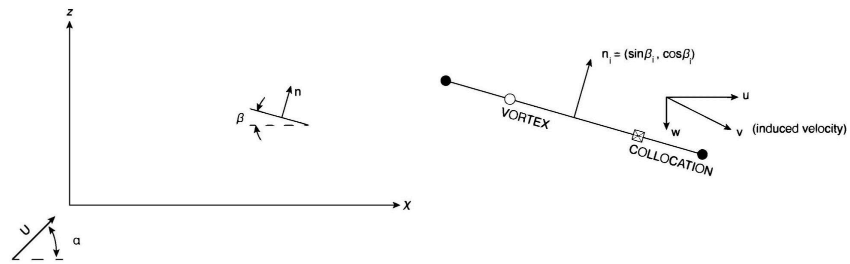


Fig 6.4 Illustration of the coordinate system using the chord as the x axis; this is the coordinate system typically used to define a_{ij}.

Note that the each outward normal, n_i, points in the same direction for this case since both panels have the same orientation, but in general each panel could be at some angle β_i. Fig. 6.4 shows the general definition of the panel angle to the chord, the latter being along the x axis. The outward normal to the panel is given as n_i. For the general case shown in Fig. 6.4 the outward normal is at an angle β_i to the z axis.

The general equation for the boundary condition is Eqn. (6.5), which we write slightly differently as:

$$v_i \cdot n_i = -U \cdot n_i \tag{6.6}$$

Where v_i is the induced velocity by the set of vortices evaluated at each panel j and n_i is the panel outward normal. The left hand side is the dot product of the induced velocity with the outward normal. The right hand side is the component of the freestream velocity aligned with the outward normal. These two balance each other to yield zero velocity crossing the panel. The induced flow from all of the panel elements needs to be added together to get the value of v_i in the above equation.

The general form for the velocity, which has components identified as u, w,at any point in the flow with a vortex element located at (x_o, z_o) is obtained from above and written here as:

$$u = \frac{\Gamma}{2\pi} \frac{(z - z_o)}{(x - x_o)^2 + (z - z_o)^2} \tag{6.7}$$

$$w = \frac{-\Gamma}{2\pi}\frac{(x-x_o)}{(x-x_o)^2+(z-z_o)^2} \tag{6.8}$$

where we have transformed v_θ into the Cartesian components u, w.

Writing this as a matrix we obtain a general expression for the induced velocity anywhere in the flow (x, z) :

$$\begin{pmatrix} u \\ w \end{pmatrix} = \frac{\Gamma}{2\pi r^2}\left(\begin{smallmatrix} 0 & 1 \\ -1 & 0 \end{smallmatrix}\right)\begin{pmatrix} x-x_o \\ z-z_o \end{pmatrix} \tag{6.9}$$

Note here that $x - x_o$ is the distance along the x axis from a vortex element to the point of interest where the velocity is being evaluated. Similarly for $z - z_o$. Since the values for Γ are not known, it is convenient to write this set of two equations for a "unit value" of Γ, that is $\Gamma = 1$. Keep in mind that the flow caused by each vortex when evaluated on a surface results only in a component normal to the surface. We also are going to use values of x_o and x based on the location of the vortices and collocation points, respectively.

Consequently, for our two panel example, where each panel is along the chord line, or the x axis, we can write the velocity components given below, however here these velocities assume $\Gamma = 1$ in the above equations. The notation used below is that the first subscript, i, represents the collocation point of interest at some panel, and the second subscript, j, identifies the vortex element at some panel that induces the velocity at the i collocation point. That is to say these subscripts do NOT represent components of vectors in space in the conventional manner, and here u and w are the x and z component of the velocity at the surface.

$$(u_{11}, w_{11}) = \left(0, -\frac{1}{\frac{2\pi c}{4}}\right) = (0, -\frac{2}{\pi c}) \tag{6.10}$$

$$(u_{21}, w_{21}) = \left(0, -\frac{1}{\frac{2\pi 3c}{4}}\right) = (0, -\frac{2}{3\pi c}) \tag{6.11}$$

$$(u_{12}, w_{12}) = \left(0, \frac{2}{\pi c}\right) \tag{6.12}$$

$$(u_{22}, w_{22}) = \left(0, -\frac{2}{\pi c}\right) \tag{6.13}$$

For instance, u_{12} is the x component of velocity at the collocation point of panel 1 caused by the vortex element on panel 2. Also, w_{12} is the z component of velocity at collacation point 1 caused by the vortex element on panel 2.

We can define, again for $\Gamma_\rho = 1$, for each panel with vortex j, the following matrix a_{ij}:

$$a_{ij} = \boldsymbol{v}_{ij} \cdot \boldsymbol{n} \tag{6.14}$$

This represents the component of the induced velocity in the outward normal direction for a given panel $\Gamma = 1$. We need to be careful here with this notation, v_{ij} in the equation above is the velocity vector caused by the vortices with unit value circulation with the subscripts: i represents the collocation point and j the vortex location inducing that flow. Also, the dot product with n (the outward normal for each panel), gives the projection of v normal to the surface. For example for the above example n is only in the z direction.

$$a_{11} = \left(0, -\frac{2}{\pi c}\right) \cdot (0, 1) = -\frac{2}{\pi c} \tag{6.15}$$

so the elements a_{ij} are quantities representing a measure of the velocity contribution normal to each panel from each vortex j at each collocation point i for a $\Gamma_j = 1$ at each panel.

Once we have identified the components a_{ij}, which, as shown above for the two panel example, are determined only by geometry, it is possible to use the impervious boundary condition to find each of the values for Γ_j. We have a general equation for our two element surface for panels $i = 1, 2$ that says the sum of the outward normal velocity at each panel with the freestream component normal to that panel must be zero. The outward normal velocity due to summation of the induced velocity over both panels is for some unknown distribution of circulations, Γ_j:

$$\sum_{j=1}^{2} a_{ij}\, \Gamma_j \tag{6.16}$$

So this expression is balanced by the contribution from the free stream velocity: $-U \cdot n$ (this is the component of U normal to the surface).

The final system of equations become:

$$\sum_{j=1}^{2} a_{ij}\Gamma_j = -(U_x, U_z) \cdot (n_x,\ n_z) \tag{6.17}$$

Here the right hand side value of U is represented as the vector (U_x, U_z) indicating that U is a vector with x and z components that depends on the angle of attack given by the chord line. The x, z components of the outward normal will depend on the angle β for each panel. The geometry of all of the panels fully determines a_{ij}, as shown above in the simple two panel example. Consequently, once the geometry is known the values of a_{ij} and $(n_x,\ n_z)$ are all determined. Then knowing U the set of Eqns. (6.17) provides the means to find all of the values of Γ_j, where the index represents each of the panel vortices. In the above two panel example there are two values of Γ.

From the solution of Γ_j we can find the lift on each panel, $\Delta L_j = \rho U \Gamma_j$ and then the total lift as the sum over all panels:

$$\sum_{j=1}^{2} \Delta L_j = C_L \left(1/2 \rho U^2 c\right) \tag{6.18}$$

Recall that the lift is normal to the direction of the freestream velocity, so the lift of each panel, even though the panels may all be at different angles of attack, are all in the same direction.

It is also possible at this point to calculate the velocity tangent to the surface at each panel. That is to say, instead of finding the component normal to the surface to satisfy the impermeable boundary condition find the component tangent to the panel, for each panel and combine this with the component of U tangent to the panel. The induced velocity tangent to the panel can be found using the dot product of the vortex induced velocity with the unit vector tangent to the panel. The tangent unit vector is given by its x and z components as $cos\beta_i$, $sin\beta_i$, respectively with β_i being the angle of the panel relative to the chord line which is the x axis, show in Fig 6.4. The freestream contribution is found from the value of U. This combined velocity yields the velocity vector of the flow at each panel and can be used in the Bernoulli equation to find the pressure at each panel. The Bernoulli equation is applied from a known upstream condition (known pressure and freestream velocity) to each panel in order to calculate the panel pressure.

Thin Airfoil Theory Overview

This is an overview of what is known as "Thin Airfoil Theory" used to develop some major results used to analysis flow over airfoils in general, and provide insight into the nature and trends of lift forces generated by flow over airfoils. Airfoils in general come in different geometries and shapes that induce flow perturbations on an approaching flow. These perturbations to the velocity field result in pressure changes that cause a net force on the foil. The force is typically divided into the component aligned with the oncoming flow direction of the fluid (drag force) and the force normal to the oncoming flow direction, (lift force). Viscous effects are not included, but the interested reader may explore some of the options of how certain viscous effects are included by researching on the internet (for instance J.E. Yates, 1980, "Viscous Thin Airfoil Theory", Aeronautical Research Associates of Princeton, Report 413, as well as several more recent developments). In this short description we are limited to mostly two-dimensional steady flow, although the theory can be extended. Thin airfoil theory can also be extended to unsteady flows such as vibrating or oscillating foils or rotating airfoils (see J. Katz and A. Plotkin, 2010, Low Speed Aerodynamics, Cambridge University Press,).

The designation of a "thin" foil provides several mathematical conveniences, which includes the use of "small perturbations" to the freestream flow. This allows some simplification of terms in the governing equations. These mathematical details are not discussed here, but are available in more advanced book on aerodynamics (see J. Katz and A. Plotkin, Low Speed Aerodynamics) The geometric terms used to define the geometry of an airfoil are shown in Fig. 6.5. The assumption is that the foil thickness (its maximum value) is much less than the chord length, c, of the foil. The chord length is measured from the leading edge to the trailing edge (in a straight line called the chord line) of the foil. The leading edge is at the front tip and trailing edge is at the rear tip. This can be seen in the figure below which shows the chord length, thickness and also the "camber line" for a given cross section of a wing. The camber line is a line extending from the leading edge to trailing edge that defines the mid-point between the top and bottom of the foil and its effects are discussed later. The degree to which this is curved (rather than falling directly on top of the chord line) defines the "camber".

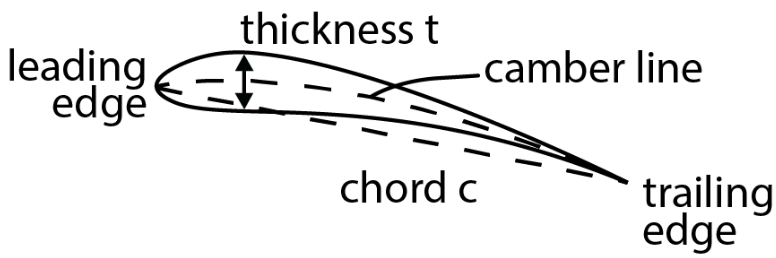


Fig 6.5 Illustration of the basic elements used to define an airfoil, the chord line and camber line and foil thickness; in thin airfoil theory the foil is replaced by the camber line, ignoring the thickness effects.

The general flow situation needed to generate a lift force involves asymmetry between the flow above and below the foil. As flow goes over and under the foil (from the left in the figure), due to asymmetry the flow over the top will be different from that on the bottom. Asymmetry can be set up in two ways, one by foil shape and the other by rotating the foil relative to the freestream flow direction. If the chord line is curved upward providing camber (see figure) then the flow path along the top and bottom will be different and the velocities most likely will differ. The other way to create asymmetry is to tilt the foil relative to the flow of the fluid, U, this tilt is called the angle of attack, α. Specifically, the angle α is between the chord line and the freestream velocity. The velocity difference between the top and bottom flow will imply that the pressure distribution along the top and bottom are also different from each other. A net force results acting on the foil by this pressure difference. Integrating the pressure difference along the entire foil yields the net force.

In thin airfoil theory, typically an inviscid flow is assumed (no consideration of frictional force between the flow and the foil). However, viscous forces occur at the foil surface and are assumed to result in vorticity perturbations along the surface. The foil thickness is taken as thin so the top and bottom surfaces are taken together as a single surface with flow over the top and bottom along the camber line. The flow is analyzed using a distribution of local vorticity placed along the camber line. This is called a vortex sheet (in that it extends into the span of the foil) and is shown in Fig. 6.6. as a series of small vortices distributed along the foil from leading edge to trailing edge. We denote this distribution by $\gamma(x)$ which represents the local circulation per length dx. Recall circulation has units of velocity times distance (m2/s), so $\gamma(x)$ has units of (m/s). Note that $\gamma(x)$ is not a constant along the foil. For this analysis we assume that circulation is positive if rotation is clockwise. This is opposite to the more conventional sign for circulation previously used, but this change is convenient for airfoils since this designation allows for the generation of upward lift for positive circulation.

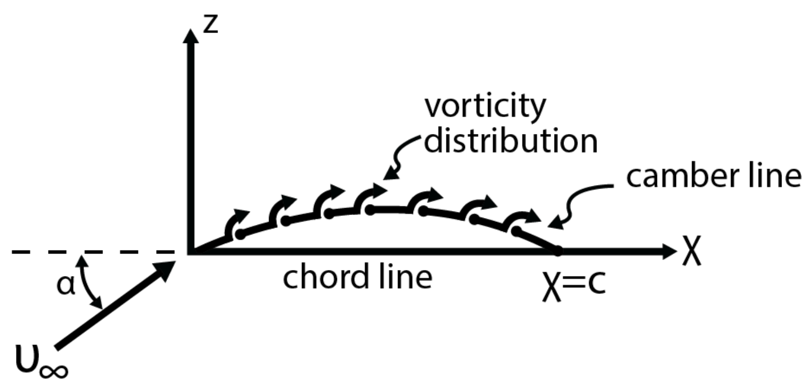


Fig 6.6 Geometry used to define the flow configuration of a thin airfoil where [latex]\alpha[/latex] is the angle of attack (angle between the approaching free stream and the chord line); a distribution of vorticity is assumed along the airfoil shown here as discrete vortex elements but in thin airfoil theory is assumed to be a continuous distribution.

General Formulation of Coefficient of Lift

The flow over the foil in reality is viscous flow and it forms a viscous layer of flow (a boundary layer) along the foil. Within this thin layer is vorticity based on the local magnitude of the velocity derivative, $\partial u/\partial y$, where y is the coordinate normal to the foil surface. As was mentioned, this distribution of vorticity gives rise to the local circulation distribution $\gamma(x)$. Also, the integration of $\gamma(x)$ along the entire foil results in a value of total circulation for the foil, Γ, such that:

$$\Gamma = \int_0^c \gamma(x)\,dx$$