B1 Project
Simulation of free flight data

Luca di Mare and Luke Doherty

1. Introduction

1.1. Purpose of the project

The purpose of this project is to write Matlab programs to analyse free flight data.

Free flight data give the position of the centre of mass and attitude a body gliding through the air. The body is not propelled and moves under the action of gravity and the aerodynamic forces it generates. Free flight data can be used to obtain relation between incidence, lift and drag. This information is valuable when designing aircraft or other vehicles. The programs we will write allow us to predict and analyse the free flight behaviour of lifting bodies.

1.2. Background

The only forces acting on the body are gravity G, lift L and drag D. The pressure distribution on the body surface also gives rise to a pitching moment M, which causes the body to rotate around its centre of mass. The trajectory of the body obeys

$$m\ddot{x} = G + L + D \tag{1.1}$$

$$I\ddot{\theta} = M \tag{1.2}$$

For steady flow, it is customary to write L, D and M in terms of nondimensional lift and force coefficients:

$$L = \frac{1}{2}\rho u_\infty^2 CW\ C_L(\alpha) \tag{1.3}$$

$$D = \frac{1}{2}\rho u_\infty^2 CW\ C_D(\alpha) \tag{1.4}$$

$$M = \frac{1}{2}\rho u_\infty^2 C^2 W\ C_M(\alpha) \tag{1.5}$$

where α is the incidence angle, C is the aerofoil chord and W is its span.

When the flow is unsteady, as is the case in this project, the relation between lift, drag, moment, and incidence angle is more complicated: when the lift changes, vortices are shed from the trailing edge of the body. These vortices induce their own velocity field, which effectively causes the body to meet the air at an incidence different from the one determined from flight attitude and velocity. The influence of unsteady flow can however be modelled by numerical means.

1.3. Objectives

The objective of this project is to develop a progression of models for the behaviour of an aerofoil section in incompressible flow. The models are based on potential flow and boundary layer theory, which you have encountered in the P4 and A4 modules. More precisely, we will write codes for: inviscid, steady thin aerofoil analysis; inviscid, steady thick aerofoil analysis; inviscid, unsteady thick aerofoil analysis; viscous, unsteady thick aerofoil analysis.

1.4. Structure of this document

This document contains information about what is expected of you; information about the panel method, which is the numerical method underpinning the models listed above, together with indications of Matlab functions that can be used as starting points. This document contains grey text boxes, like the one below. These indicate points where we believe you can attempt modifications or developments in the codes.

2. Tasks

Your tasks will be to write Matlab codes implementing the models described in detail in the following sections. You will write four codes,

1. Inviscid, steady thin aerofoil analysis
2. Inviscid, steady thick aerofoil analysis
3. Inviscid, unsteady thick aerofoil analysis
4. Viscous, unsteady thick aerofoil analysis

You will use the last of these to

1. Integrate the viscous, unsteady, thick aerofoil code with Newton's second law for the motion of the aerofoil
2. Determine the values of constants appearing in the friction coefficient correlation for the viscous case resulting in the best fit with trajectory data.

For each code you will produce data proving the correctness of the implementation of the main routines. Boundary conditions and geometry will be provided separately.

3. The panel method

3.1. General information

The models we will develop are based on dividing the flow around the aerofoil in two distinct regions. The first region is a thin layer of fluid near the wall of the aerofoil and subject to the action of viscosity. This layer is the boundary layer. The second region is the flow outside the boundary layer, and is essentially unaffected by viscosity. In this second region, the flow obeys the law of potential flow theory. The two regions interact because the growth of the boundary layer on the body surface effectively modifies the shape "seen" by the flow far away in the potential flow region. The potential component of the flow, in return, determines the streamwise pressure gradient which determines the rate of growth of the boundary layer. We will discuss these interactions more in detail in the second part of this section and we focus first on the potential flow region to start.

3.2. Panel method for potential flow

The panel method seeks to build numerical approximations of potential flows by combining simple flow patterns. You may recall a similar approach in the P4 and A4 modules to construct flow fields such as those around a circular cylinder.

Panel methods are a particular type of *boundary element methods*, which are themselves a type of finite element methods. A boundary element method is a finite element method specialised for mathematical problems where the solution can be constructed directly from its distribution on the boundary. The solutions of Laplace' equation enjoy this property.

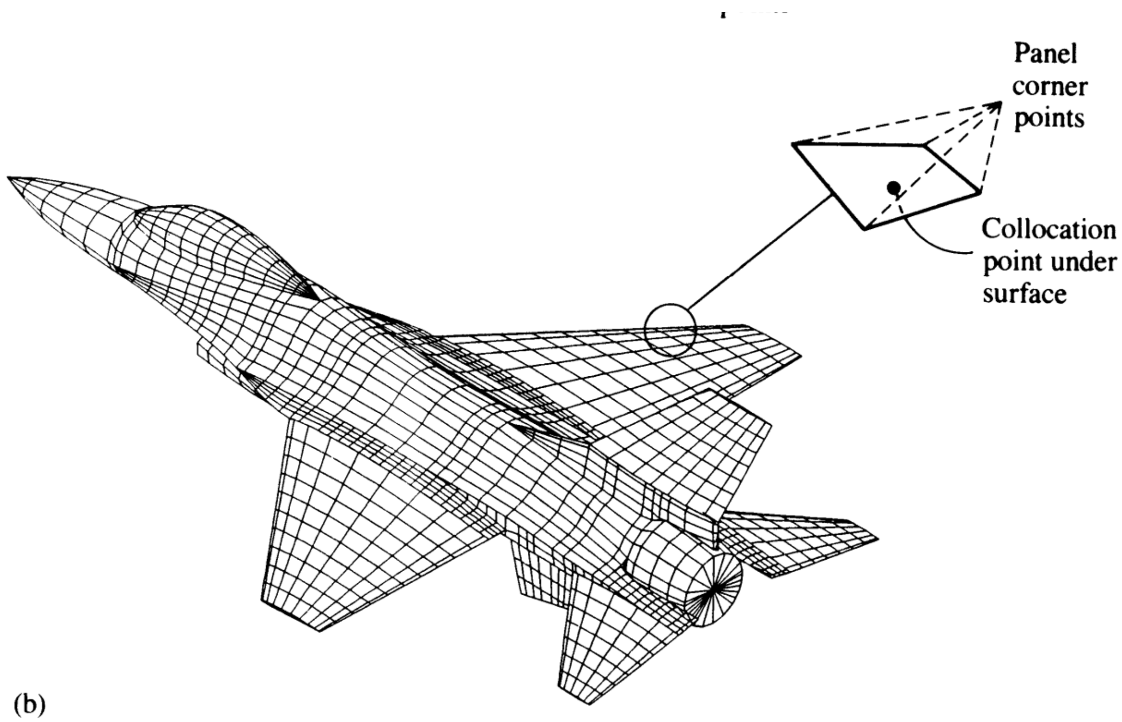

Figure 1: panel representation of a General Dynmics F16 fighter. From Katz, J., & Plotkin, A. (2001). *Low-Speed Aerodynamics* (2nd ed., Cambridge Aerospace Series)

In a typical panel method, the body is divided into panels – hence the name of the method. Each panel generates a simple flow field, as an example a source or a vortex. The strengths of the panels are determined by requiring that the surface of the body is a stream-surface of the flow. This is typically achieved by requiring that the normal velocity at a suitable chosen point on each panel must be nil, as shown in Figure 1. The panel method has therefore three main ingredients

- discretization of the surface into panels;
- simple flow field attached to each panel;
- impermeability condition.

We will now discuss these three elements in relation to the simplest panel method, the lumped vortex method.

3.2.1. The lumped vortex method – thin lifting line aerofoils

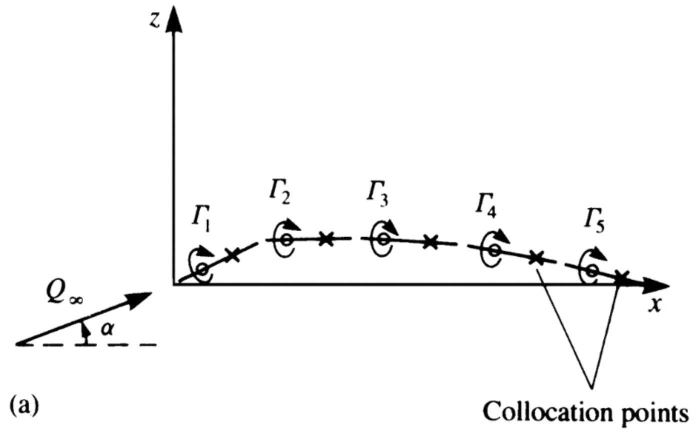

Figure 2:vortices and control points for the lumped vortex method. From Katz, J., & Plotkin, A. (2001). Low-Speed Aerodynamics (2nd ed., Cambridge Aerospace Series).

The lumped vortex method can be used to characterise the behaviour of aerofoils of small thickness compared to their chord, so that the entire geometry of the aerofoil can be identified by its camber line (i.e. the line that runs half way between the top and bottom surface of the aerofoil).

3.2.1.1. **Arrangement of singularities and control points**

In the lumped vortex method, the flow generated by each panel is a potential vortex located at the point at 25% of the chord of the panel. Impermeability is enforced on each panel at a control point located at 75% of chord. See the diagram in Figure 1Figure 2.

3.2.1.2. **Singularity flow field**

If (x_j, y_j) is the position of the vortex attached to the j-th panel and Γ_j is its strength, the velocity field induced by that panel at the point with coordinates (x, y) is

$$\Gamma_j \boldsymbol{v}_j(x,y) = \Gamma_j \frac{1}{(x-x_j)^2 + (y-y_j)^2} \begin{bmatrix} -y + y_j \\ x - x_j \end{bmatrix} \tag{3.1}$$

If the position (x, y) is the control point (x_i, y_i) of panel i, and $\boldsymbol{n}_i$, then the impermeability condition for that panel reads:

$$\boldsymbol{v} \cdot \boldsymbol{n}_i = \boldsymbol{n_i} \cdot \left(\boldsymbol{v}_\infty + \sum_j \Gamma_j \, \boldsymbol{v}_j(x_i, y_i) \right) = 0 \tag{3.2}$$

If the aerofoil is discretised with N panels, we can write N equations of the type above. These N equations form a linear system in the N unknowns Γ_j, that can be solved by standard methods.

3.2.1.3. **The aerodynamic influence matrix**

If the impermeability condition is enforced at all the N panels on the body, then N linear equations in the N unknown circulations can be written:

$$\sum_i a_{ij} \, \Gamma_j + \boldsymbol{n}_i \cdot \boldsymbol{v}_\infty = 0 \tag{3.3}$$

The matrix element a_{ij} is the normal velocity induced at the control point of panel i by the circulation of panel j:

$$a_{ij} = \boldsymbol{n}_i \cdot \boldsymbol{v}_j(x_i, y_i) \tag{3.4}$$

The matrix a_{ij} is called the aerodynamic influence matrix and it features prominently in all panel methods.

3.2.1.4. **Recovery of the pressures**

The evaluation of forces acting on the aerofoil requires a way to estimate pressures. In our calculations we will assume the any boundary layer growing on the surface of the body is sufficiently thin that the pressure at the edge of the boundary layer is an accurate approximation of the pressure on the surface. This is equivalent to saying that the boundary layer is thin compared with the local radius of curvature of the surface.

The problem of evaluating pressures on the surface is therefore reduced to the problem of evaluating pressures at the edge of the boundary layer, where the properties of potential flow can be exploited. It is particularly useful to recall that potential flow obeys Bernoulli's relation:

$$const = p + \frac{1}{2}\rho u^2 \tag{3.5}$$

In the case of the lumped vortex method, we can evaluate velocities at any point on the plane (x, y) by

$$\boldsymbol{v} = \boldsymbol{v}_\infty + \sum_j \Gamma_j \, \boldsymbol{v}_j(x_i, y_i) \tag{3.6}$$

When evaluating the expression above at the control points of the panel, a velocity vector that is tangent to the surface is found; this happens because of the impermeability condition which suppresses the normal component. However, a difficulty arises because the presence of circulation on the panels causes the tangential velocities to have a discontinuity across the panel itself. For a lumped vortex method, the discontinuity is measured by the circulation of the panel or equivalently, the strength of its lumped vortex:

$$\Delta v_{t,j} = \Gamma_j \tag{3.7}$$

There are then two values of tangential velocity, one immediately above, and one immediately below the surface:

$$v_{t,i}^{\pm} = \boldsymbol{t} \cdot \left(\boldsymbol{v}_\infty + \sum_j \Gamma_j \, \boldsymbol{v}_j(x_i, y_i) \right) \pm \, \Gamma_j \tag{3.8}$$

From this estimate, the pressures can be recovered by applying Beroulli's relation

$$p_\infty + \frac{1}{2}\rho v_\infty^2 = p_i^\pm + \frac{1}{2}\rho v_{t,i}^\pm \tag{3.9}$$

<u>Week 5 practical</u>

You can now try to access and modify the Matlab functions in the folder

Inviscid\steady\thin

The functions contain empty lines you can fill in to get started. More details in the file

Inviscid\steady\thin\Readme

3.2.2. The Hess-Smith panel method

The lumped vortex method discussed in the previous section is not very suitable for detailed calculations because it ignores the surface curvature of the aerofoil. There is a large body of literature on high-order panel methods for aerofoil sections. Here, we will report only one of the most commonly applied, the Hess-Smith panel method.

3.2.2.1. **Arrangement of singularities and control points**

The singularities in the Hess-Smith panel method are represented by source distributions uniformly spread over each panel. There is also a uniform vorticity distribution which we will discuss later. The control points are located at the midpoint of each panel.

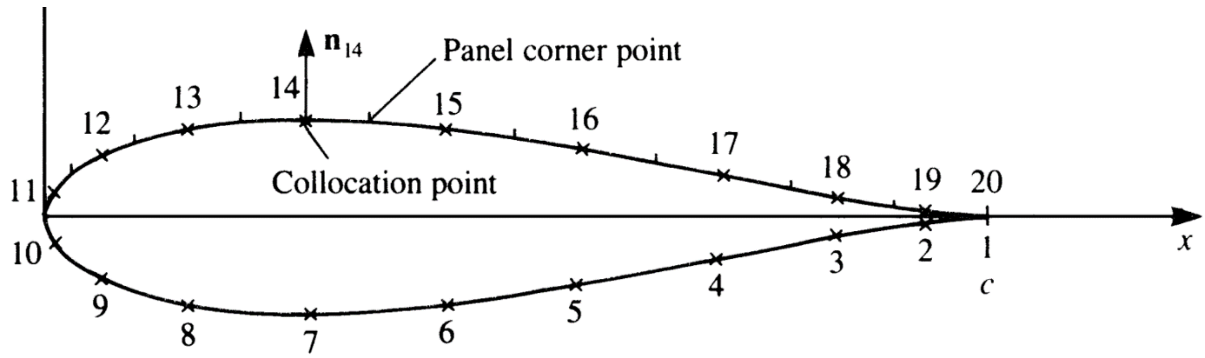

Figure 3: panel corner points and collocation points for the Hess-Smith panel method. From Katz, J., & Plotkin, A. (2001). Low-Speed Aerodynamics (2nd ed., Cambridge Aerospace Series).

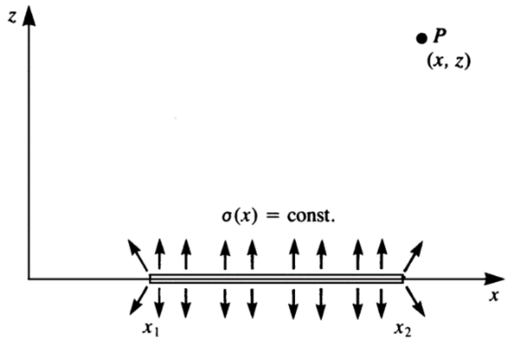

Figure 4: a uniform source distribution along the $[x_1, x_2]$ interval on the x-axis. From Katz, J., & Plotkin, A. (2001). Low-Speed Aerodynamics (2nd ed., Cambridge Aerospace Series).

3.2.2.2. **Singularity flow field**

Each panel carries a uniform source distribution σ_j. It is straightforward to show that the velocity field associated with a uniform source distribution of strength σ placed on the interval $[x_0, x_1]$ on the x −axis is

$$\boldsymbol{w} = \begin{bmatrix} u \\ v \end{bmatrix} = \frac{\sigma}{2\pi} \begin{bmatrix} \log \sqrt{\dfrac{(x-x_0)^2+y^2}{(x-x_1)^2+y^2}} \\ \tan^{-1}\left(\dfrac{y}{x-x_1}\right) - \tan^{-1}\left(\dfrac{y}{x-x_0}\right) \end{bmatrix} \tag{3.10}$$

This flow field can be obtained by integrating flow of a point source along the segment $[x_0, x_1]$. The velocity field of the source distribution has a discontinuity σ in normal velocity. The tangential velocity is continuous across the panel.

The velocity field generated by a uniform vorticity distribution on the same panel is identical, except that its v and u velocity components are swapped and the sign of one of them changed. See Figure 3 and Figure 4.

The flow field of a panel with generic orientation can be obtained as follows. Let $\boldsymbol{x}_0$ and $\boldsymbol{x}_1$ be the end-points of the panel. The tangent vector of the panel is

$$\boldsymbol{t} = \frac{\boldsymbol{x}_1 - \boldsymbol{x}_0}{\|\boldsymbol{x}_1 - \boldsymbol{x}_0\|} \tag{3.11}$$

and the normal vector is

$$\boldsymbol{n} = \boldsymbol{t} \times \widehat{\boldsymbol{k}} \tag{3.12}$$

Then, the velocity field of the panel $(\boldsymbol{x}_0, \boldsymbol{x}_1)$ is

$$\boldsymbol{w} = u\,\boldsymbol{t} + v\,\boldsymbol{n} \tag{3.13}$$

3.2.2.3. The Kutta condition

If we follow the process used for the lumped vortex method in the case of an aerofoil discretised with N panels, we obtain a well-conditioned aerodynamic influence matrix of size $N \times N$. This matrix can be used to determine a source distribution once the far-field velocity $\boldsymbol{v}_\infty$ is known. However, this is not sufficient to represent all possible potential flows around the aerofoil. This problem appears because the potential flow in a multiply connected domain – i.e. one with holes – is not uniquely determined. In order to make the flow uniquely determined it is necessary to specify the circulation around each "hole". You have witnessed this peculiarity of potential flow when studying the flow past rotating cylinders.

On an aerofoil such as the one in Figure 5, there is one and only one value of circulation that gives a finite velocity at the trailing edge. This circulation also make the tangential velocities on the two sides of the aerofoil trailing edge identical. As any physical solution must have finite velocities at the trailing edge, we select this unique solution as the physically valid one.

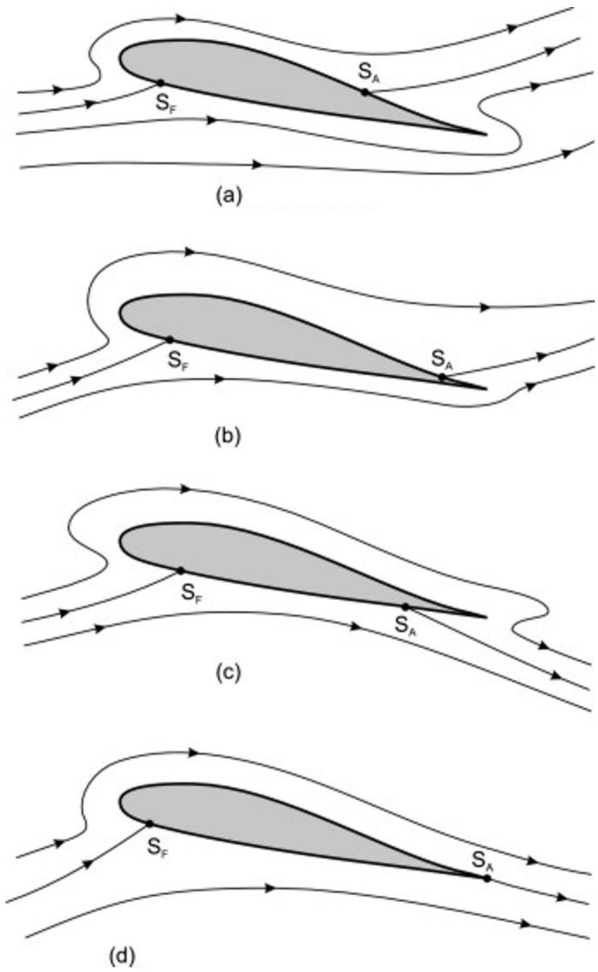

Figure 5: Flows (a)-(d) are all solutions of the potential flow equations and only differ by the circulation around the aerofoil. Flows (a)-(c) have infinite velocity at the sharp trailing edge and are discarded because they are unphysical. From E.L. Houghton, P.W. Carpenter, S.H. Collicott, D.T. Valentine, Chapter 6 - Thin Airfoil Theory, Editor(s): E.L. Houghton, P.W. Carpenter, S.H. Collicott, D.T. Valentine, Aerodynamics for Engineering Students (Seventh Edition), Butterworth-Heinemann, 2017.

This criterion is known as Kutta condition. The Kutta condition provides one additional equation, which can be used to find the circulation around the aerofoil. In the Hess and Smith method, the circulation is obtained by attributing a uniform vorticity distribution to all panels. The value of vorticity is the same around the aerofoil.

Week 6 practical

You can now try to access and modify the Matlab functions in the folder

Inviscid\steady\thick

The folder contains functions implementing the constant source element contained in this section and functions producing the geometry of popular aerofoil sections. You can build functions assembling the residual and the influence matrix similar to those used for the thin aerofoil program. More details in the file

3.2.3. Unsteady flow past a lifting body

One of the main results of the potential flow theory is that vorticity cannot be created or destroyed within the flow field. One consequence of this result is that when the circulation around a lifting body changes in time, e.g. because of variations in incidence, circulation must appear in the wake of the body and the time derivative of the circulation of the flow system formed by the body and its wake must be nought. This result is known as Kelvin's theorem.

Kelvin's theorem explains the phenomenon of the starting vortex, documented by Prandtl and shown in original photographs in Figure 6.

The vorticity shed by the body manifests itself as a tangential velocity discontinuity across its wake. While at first sight arcane, this result is in fact a consequence of the fact that lifting bodies produce lift by turning the fluid around them.

The existence of shed vorticity in the wake means that panels are needed to represent it. As the computation progresses the number of wake panels increases, as shown in Figure 7. If we denote with Γ_j^w the vorticity of the j-th wake panel, then at any point in the flow:

$$\boldsymbol{v} = \boldsymbol{v}_\infty + \sum_j \Gamma_j \, \boldsymbol{v}_j(x_i, y_i) + \sum_j \Gamma_j^w \, \boldsymbol{v}_j(x_i, y_i) \tag{3.14}$$

The vorticity of each wake panel is unchanged in time because Kelvin's theorem requires

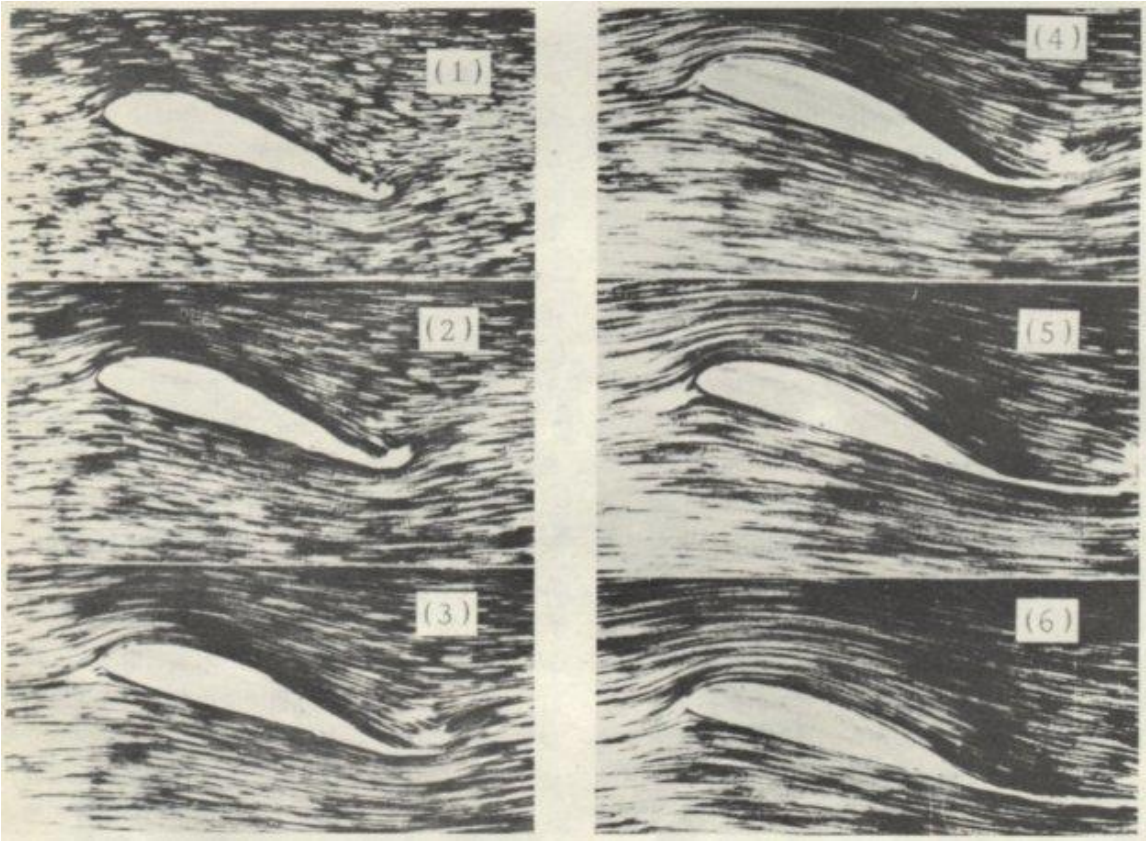

Figure 6: Prandtl's original visualization of starting vortex behind an aerofoil

$$\frac{D\Gamma_{\mathrm{j}}^{\mathrm{w}}}{Dt} = 0 \tag{3.15}$$

The only extra piece of information required is then the strength of the first wake panel, the one closest to the body trailing edge. Kelvin's theorem also helps because the time derivative of the sum of shed and bound vorticity must vanish:

$$\frac{d}{dt}\left(\Gamma_w + \int \gamma ds\right) = 0 \tag{3.16}$$

If a first order integration scheme in time used then

$$\frac{df}{dt} \approx \frac{f^{n+1} - f^n}{\Delta t} \tag{3.17}$$

The time derivative of the wake circulation is the circulation of the first wake panel Γ_1^w

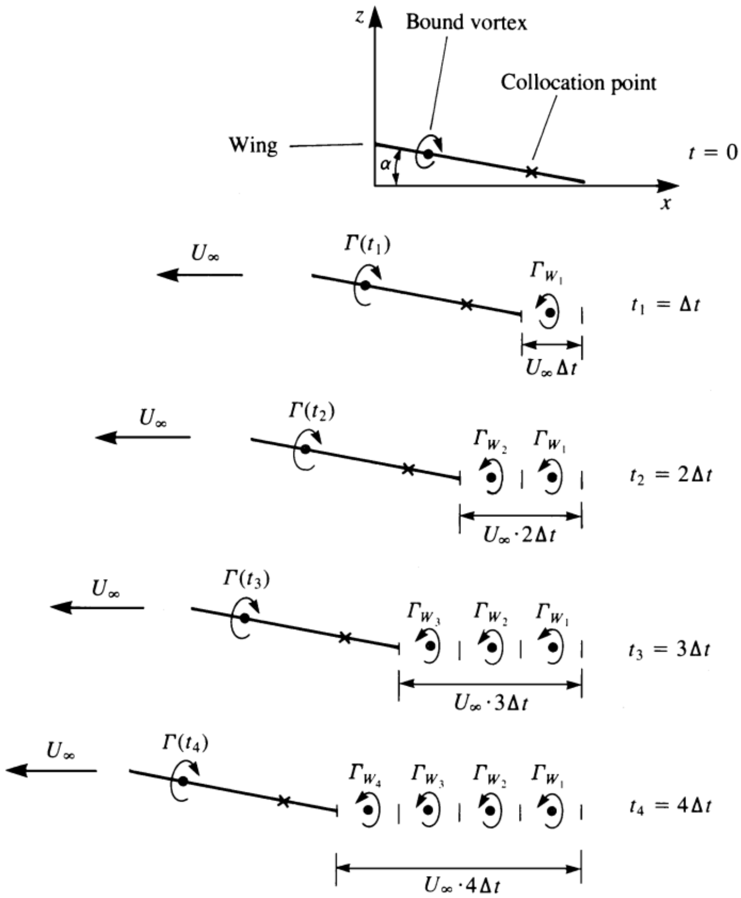

Figure 7: growth of the wake at every time step in an unsteady panel method.

The time derivative of the integral is

$$\frac{d}{dt}\int \gamma ds \approx \frac{1}{\Delta t}\sum_i \Gamma_i^{n+1} - \Gamma_i^n \tag{3.18}$$

The discrete form of the unsteady Kutta condition therefore reads

$$\frac{1}{\Delta t}\sum_i (\Gamma_i^{n+1} - \Gamma_i^n) + \Gamma_1^w = 0 \tag{3.19}$$

We also need a criterion to determine the position of the first wake vortex. A good approximation is to place it on the streamline issuing from the trailing edge at a distance proportional to

$$d = \beta v_\infty \Delta \tag{3.20}$$

downstream of the trailing edge itself. A typical value for β is $\beta = 0.5$.

In the unsteady case the impermeability conditions are modified by the velocities induced by the wake panels:

$$\boldsymbol{v} \cdot \boldsymbol{n}_i = \boldsymbol{n_i} \cdot \left(\boldsymbol{v}_\infty + \sum_j \Gamma_j \, \boldsymbol{v}_j(x_i, y_i) + \sum_j \Gamma_j^w \, \boldsymbol{v}_j(x_i, y_i) - \dot{\boldsymbol{x}}_i \right) = 0 \tag{3.21}$$

$$\sum_i a_{ij} \, \Gamma_j + \sum_j \Gamma_j^w \, \boldsymbol{v}_j(x_i, y_i) \; + \boldsymbol{n}_i \cdot (\boldsymbol{v}_\infty - \dot{\boldsymbol{x}}_\iota) = 0 \tag{3.22}$$

$\dot{\boldsymbol{x}}$ is here the velocity of the surface due to the body motion.

In the discussion above we have not specified how the location of the wake vortices is to be determined. There are various methods, here we will use the simplest:

$$\frac{dx_j^w}{dt} = \boldsymbol{v}_\infty \qquad 3.23$$

<u>Week 7 practical</u>

You can now try modify your functions for the thick steady aerofoil case by adding a wake that is grown by one entry at every time step.

You can start from the sample code in

Inviscid\unsteady\thin

to check how this is done in the thin aerofoil case.

3.3. Unsteady integral boundary layer method

3.3.1. Model equation and definitions

The boundary layer around the body can be represented by an integral equation describing how its thickness grows. The relevant model equation is

$$\frac{1}{u_e^2} \frac{\partial}{\partial t} u_e \delta^* + (2\theta + \delta^*) \frac{1}{u_e} \frac{\partial u_e}{\partial x} + \frac{\partial \theta}{\partial x} = \frac{C_f}{2} \tag{3.24}$$

In this equation u_e is the velocity at the edge of the boundary layer evaluated from the inviscid solution, C_f is the friction coefficient

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho v^2} \tag{3.25}$$

And θ and δ^* are the momentum and displacement thickness, defined as:

$$\delta^* = \int_0^\infty \left(1 - \frac{u}{u_e}\right) dy \tag{3.26}$$

$$\theta = \int_0^\infty \frac{u}{u_e}\left(1 - \frac{u}{u_e}\right) dy \tag{3.27}$$

The momentum integral equation needs to be supplemented by relations between θ and δ^*.

One such relation is given by Green:

$$\frac{1}{u_e}\frac{\partial}{\partial x} u_e \theta G = 0.025H - 0.022 \tag{3.28}$$

$$G = \frac{2H}{H - 1} \tag{3.29}$$

where $H = \delta^*/\theta$ and $G = (\delta - \delta^*)/\theta$. Correlations for the friction coefficient exist and are well validated. For the purpose of this project we will assume that the friction coefficient is a function of the momentum thickness θ:

$$C_f = f\, Re_\theta^\gamma \tag{3.30}$$

The exponent γ and the multiplicative factor f are unknown, for the moment. We will use the data supplied to find best fits.

3.3.2. Interaction with the inviscid flow

As intimated at the start of this section, the boundary layer and the inviscid flow are coupled. The coupling takes place through the displacement of streamlines at the edge of the boundary layer due to its growth. The growth of the boundary layer can be represented in the invisicid solution as a modification to the impermeability condition. Instead of setting the normal velocity to nil, at the control point of each panel we can set

$$u_n = \frac{\partial \delta^*}{\partial t} + \frac{\partial}{\partial x} u_e \delta^* \tag{3.31}$$

<u>Week 8 practical</u>

You can now try to access and modify the Matlab functions in the folder

Inviscid\unsteady\viscous

The folder contains functions implementing integration of the unsteady boundary layer given a distribution of boundary-layer-edge velocity u_e. This function can be integrated in the functions for the unsteady inviscid model of a thick aerofoil.

inviscid\steady\thick\Readme

You will then be able to "fly" the aerofoil following the trajectory contained in the files in

trajectory

and find the best values of f and γ in equation (3.30) to match the measured data.