GENERALISING 2D POTENTIAL FLOW SOLUTION METHODS

AIMS

- To generalize point singularity methods to arbitrary 2D geometries including aerofoils
- To introduce 2D distributed elementary flows
- To introduce 2D panel methods

1 INTRODUCTION

In previous material from Fluids 1 sources, sinks or vortices have been added to a uniform flow and then any stream line of the resulting flow pattern may be taken to represent a solid surface since no flow can cross it. The range of geometries that can be tackled can be extended using conformal transformations e.g. Joukowski aerofoils, however these methods are restricted to a small number of special geometries. To handle an arbitrary 2D geometry more general methods are needed. A number of *numerical* and *simplified analytic methods* that can be sued to extend potential solutions to arbitrary geometries will be presented in the incompressible section of Aerodynamics 2.

2 GENERALISED POINT SINGULARITY METHODS FOR ARBITRARY GEOMETRIES

The simplest approach to generalising potential model generation is introduce a discrete set of singularities (point sources, doublets and/or vortices) of unknown strength at predefined locations to a free stream flow. Due to the linearity of the Laplace equation for the potential $\nabla^2\phi = 0$, the superposition of the solutions is also a valid potential solution. Boundary conditions are then applied to the arbitrary body surface so that it is a stream line. For a finite number of singularities it is only possible to enforce the boundary condition at a finite number of points on the body surface. The boundary conditions lead to a set of equations to solve for the unknown strengths of these singularities. Applying the boundary condition doesn't immediately yield a unique solution because: (1) the singularity type, number and positions can vary (2) physical considerations need to be introduced to fix the correct level of circulation, for example in 2D the Kutta condition is needed (and when extending to 3D wake modelling is required).

The type of singularity depends on the type of flow being modelled. Point sources can only be used to model non-lifting flows whereas point doublets or point vortices can model lifting flows. Combined methods can also used. Here just a point source method is considered for non-lifting flows, since historically it was the first method to be used and is probably the easiest to implement and understand. A point vortex method for thin aerofoils will be developed later in the course, since it uses results from the simplified analytic thin aerofoil theory to implement the Kutta condition.

Note that as indicated in the last handout the 2D flow is assumed to lie in the (x,z) to be consistent with the usual aerofoil notation. This just means y is replaced by z and the velocity v is replaced by w in all formulae previously derived.

2.1 Example: Generalised 2D Point Source Model

2.1.1 Basic Idea

The potential and stream function for a 2D point source at the origin are given by

$$\phi = \frac{\Lambda}{2\pi}\ln r \qquad \psi = \frac{\Lambda}{2\pi}\theta$$

and the corresponding velocity components are given by

$$v_r = \frac{\Lambda}{2\pi r} \quad v_\theta = 0$$

Then N point sources with unknown strengths Λ_i are placed at predefined locations. Since there are N unknown strengths, and there is no need to worry about fixing circulation, then the streamline condition can be enforced at N locations on the body. These locations are called <u>control points</u> or <u>collocation points</u>. The problem can then be solved and velocities, surface pressures and forces calculated as shown pictorially in Figure 1.

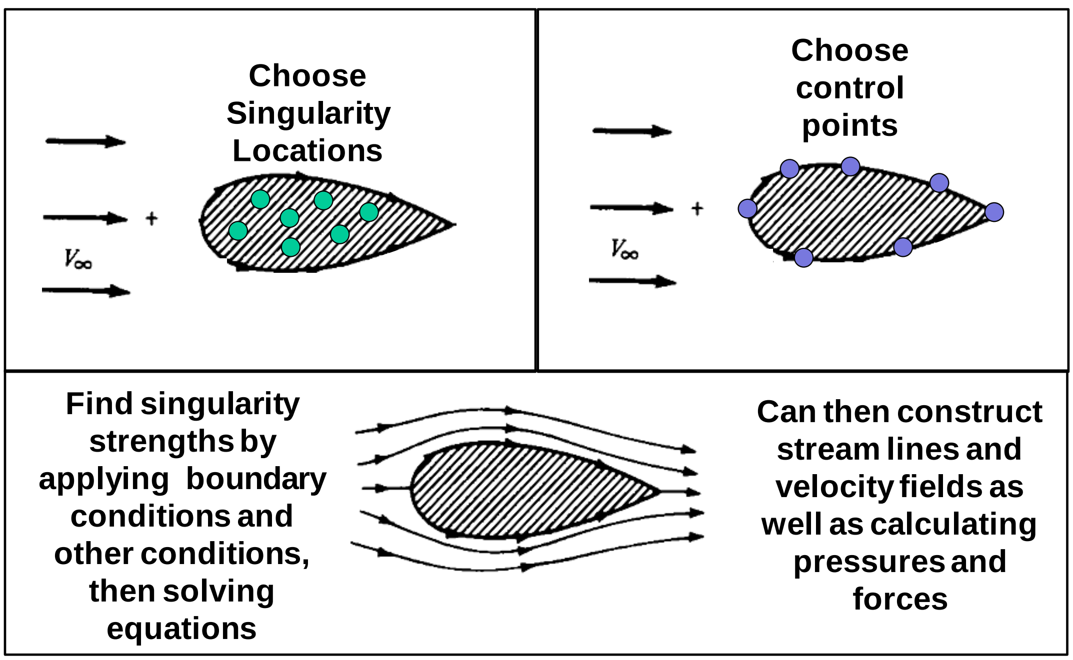


Figure 1 Pictorial representation of solution process

The boundary condition that the body surface is a streamline at the control points can be applied in two ways. The first way is by directly setting the flow velocity normal to the body surface at the control points to be zero. This is called a Neumann condition. An alternative approach specifies the potential on the body surface to make it a streamline. This is called a Dirichlet condition. Here a method that uses a Neumann boundary condition is described in detail.

2.1.2 Details of Method with a Neumann Boundary Condition

Once the locations for the point sources and control points have been selected, the first step is to define the normals $\vec{n}_i$ to the body surface at each of the control points. The surface normal component of the velocity induced by each source at every control point is then evaluated, so for example for source *j* the surface normal component of velocity induced at control point *i* is given by

$$a_{i,j}\Lambda_j$$

where the term $a_{i,j}$, that multiplies the source strength, is called the velocity influence coefficient. The boundary condition of no normal flow is then applied at control point *i* by combining the normal velocities induced by all the sources and the free stream at that control point to give

$$\sum_{j=1}^{N} a_{i,j}\ \Lambda_j + \vec{V}_\infty \bullet \vec{n}_i = 0$$

where $\vec{V}_\infty$ is the free stream velocity vector and $\vec{n}_i$ is the surface normal at control point *i*. Then the equations for all the control points are combined into one matrix equation given by

$$A\ \vec{\Lambda} = \vec{R}$$

where A is the (Velocity) Influence Matrix with entries $a_{i,j}$, $\vec{\Lambda} = [\Lambda_1, \Lambda_2, \ldots, \Lambda_N]^T$ and $\vec{R} = [R_1, R_2, \ldots R_N]^T$ contains terms from the surface normal component of the free stream. This is a linear equation which can be solved for the point source strengths. Velocities, pressures and forces can then be evaluated from the model.

In summary the method requires

- 1. Geometry discretisation
- 2. Calculation of *influence coefficients* and *influence matrix* equation
- 3. Solution of the linear set of equations
- 4. Secondary calculations: pressures, forces, off-body velocities etc.

2.1.3 Example Solutions Demonstrating Effects of Placement and Number of Point Sources

Two cases are considered. The first is flow over a non-lifting cylinder and the second is the flow over a van de Vooren aerofoil.

Non-Lifting Flow Over A Cylinder

An analytic potential solution for this test case is available for comparison with the numerical potential solution. Hundreds of possible solutions could be generated based on different source and control point locations and numbers and it is only possible to give a snapshot of results.

The first set of solutions has sources placed on the centre line of the cylinder. This is a widely used positioning for source. This location guarantees a solution that is symmetric between the upper and lower halves of the cylinder and thus the control points should all be located on one half of the cylinder (note if control points are located symmetrically all round the cylinder then the solution process will fail because matrix A will be singular in this case). The position of each control point is indicated in the stream line pictures by a cross and each source location by a circle.

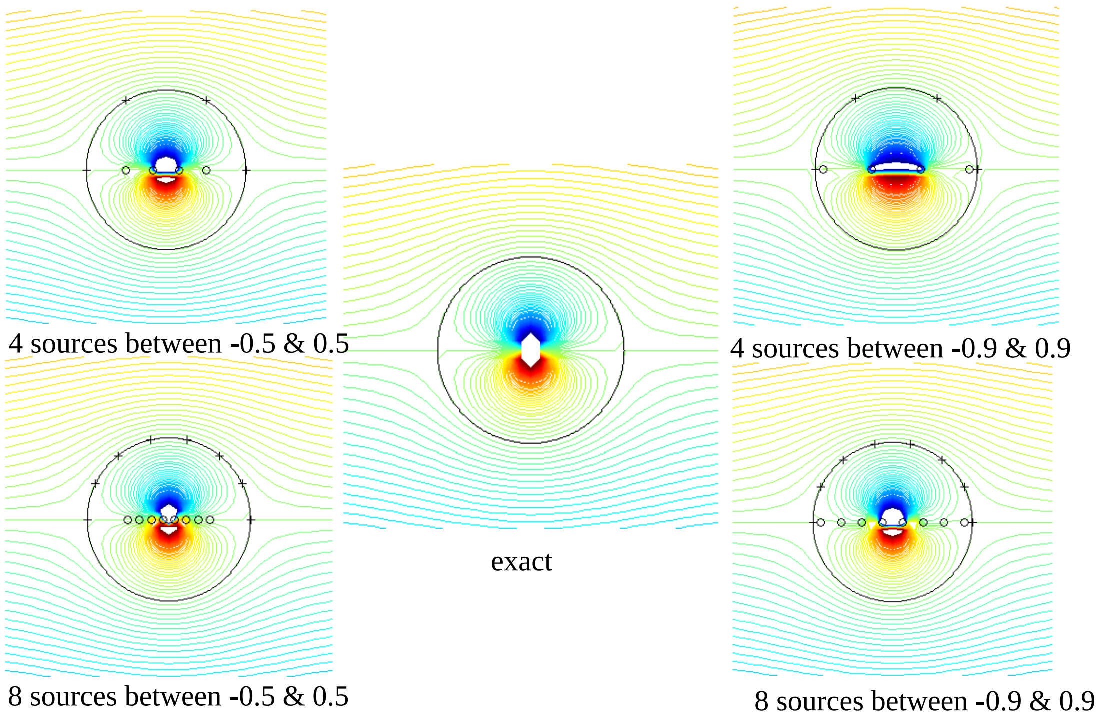


Figure 2 Effect of number and position of point sources on the centre line on stream lines

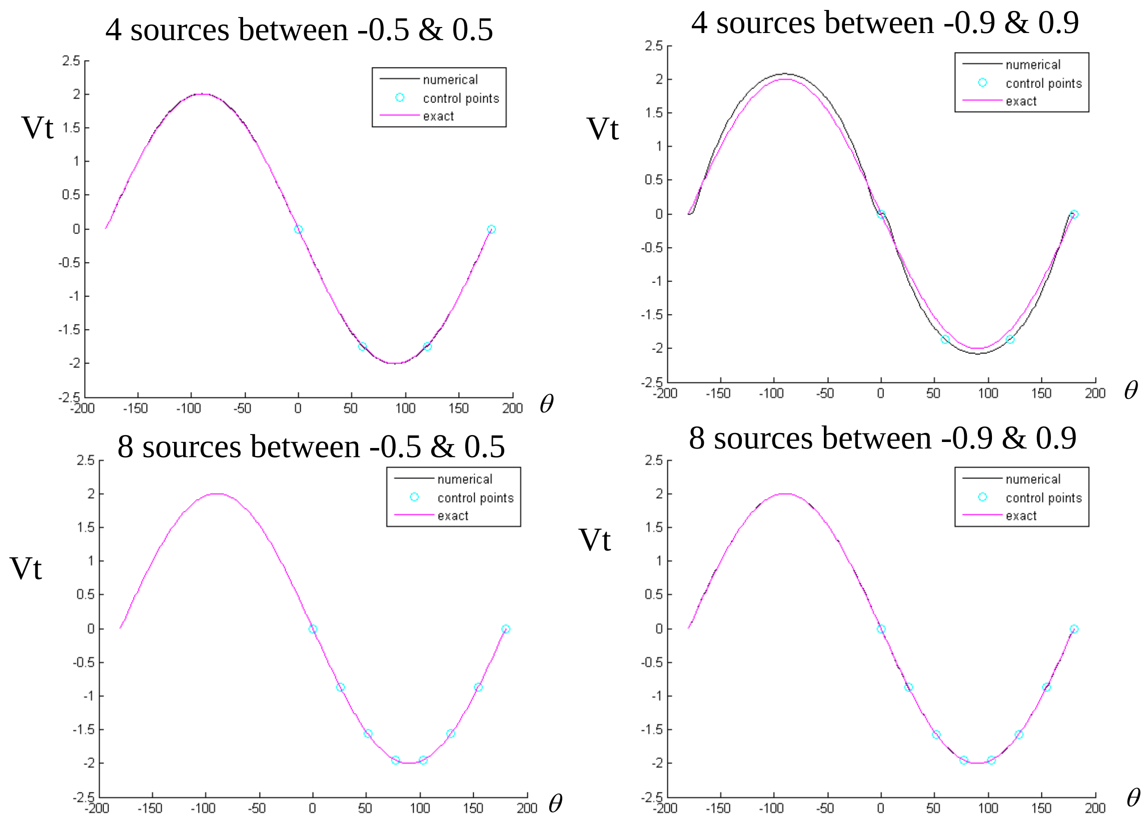


Figure 3 Effect of number and position of point sources on the centre line on tangential velocity

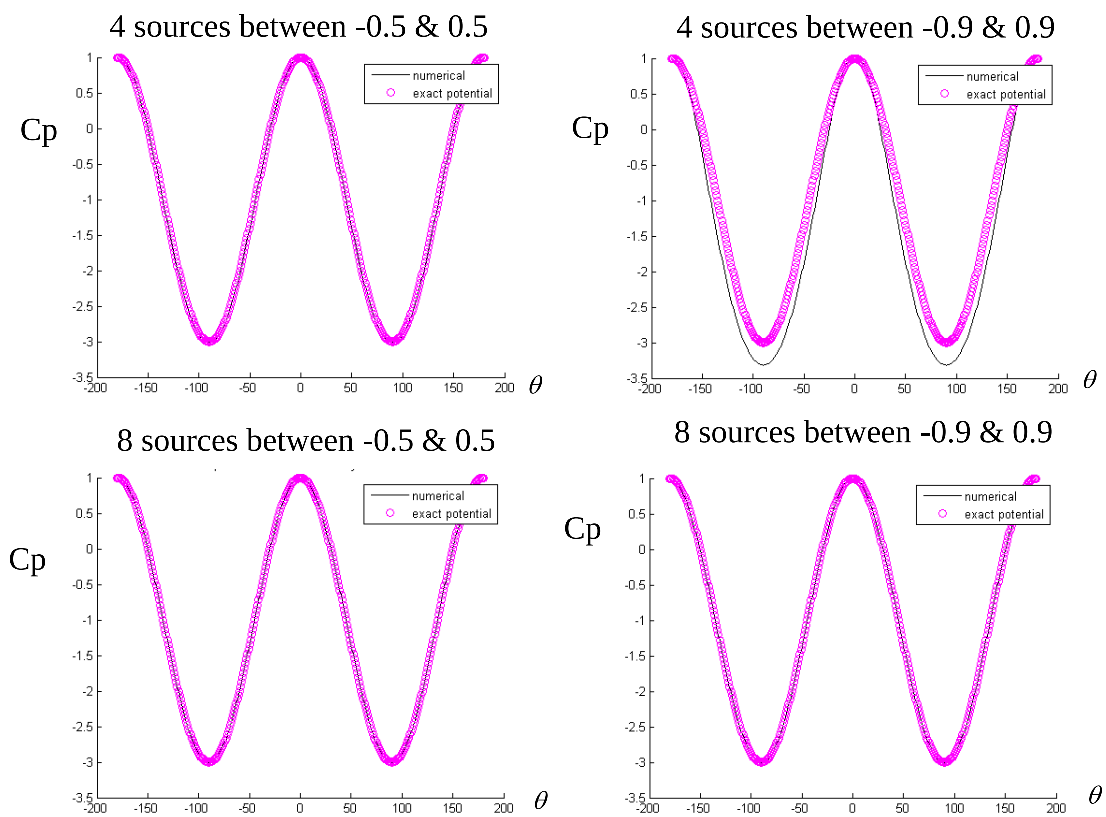


Figure 4 Effect of number and position of point sources on the centre line on Cp distribution

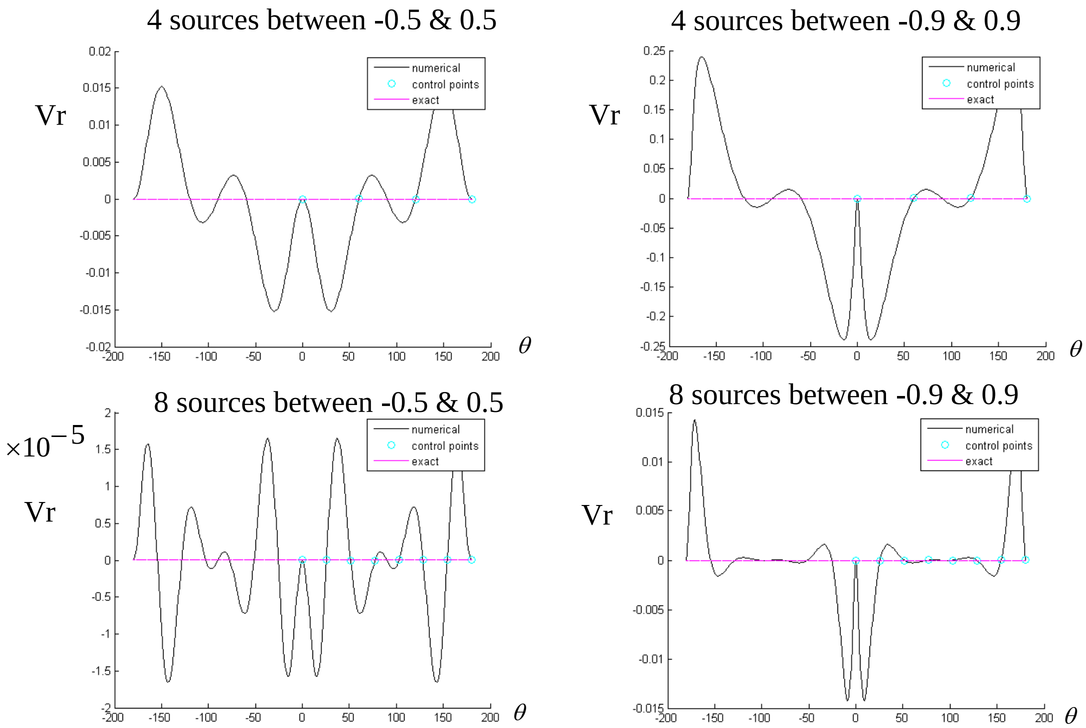


Figure 5 Effect of number and position of point sources on the centre line on radial velocity

It can be seen from Figures 2 to 5 that the numerical solution with 4 sources placed between -0.5 and 0.5 gives a much better solution than 4 sources placed between -0.9 and 0.9. The closeness of the source at -0.9 to the control point at the leading edge, and the source at 0.9 to the control point at the trailing edge, is affecting the solution. The high velocity gradients from the single source in the vicinity are influencing the behaviour in the region of these control points and causing large errors in the solution. Comparing the solutions with the 4 and 8 sources between -0.5 and 0.5 it is clear that the error in the radial velocity which should be zero on the cylinder is reduced significantly if the number of sources is increased i.e. the maximum error is reduced by a factor of 1000.

Another possibility is to place the sources on a circle with a radius smaller than that of the actual cylinder. Then using the cylinder radius equal to 1, two different radii namely R=0.5 and R=0.9 are investigated with two different numbers of points. Note in this case without specifically amending the code, symmetry is not automatically enforced so control point and sources are located on both halves of the cylinder. Figure 6 shows the stream line patterns and it should be noted that, whilst reasonable agreement is obtained except in the case of 8 sources on R=0.9, the stream function has a different value to the analytic function on the stream lines. Remember that the stream function is only determined to within an arbitrary constant and this will not impact on the velocity or pressure fields.

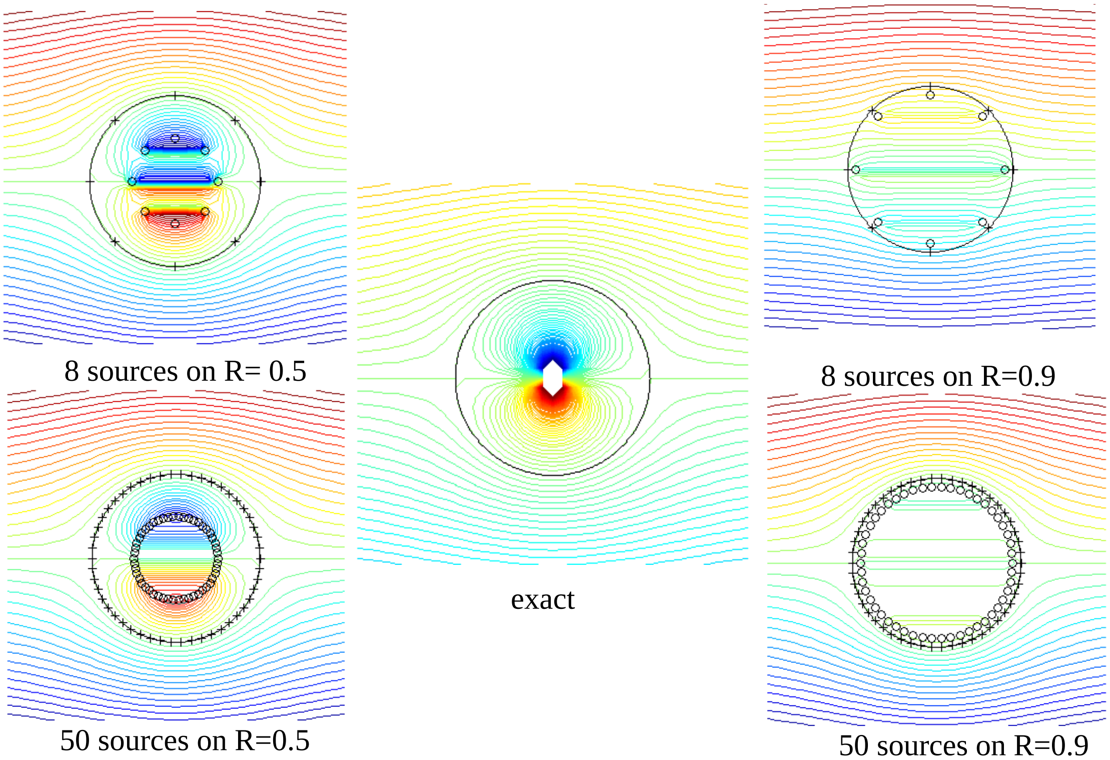


Figure 6 Effect of number and position of point sources on the centre line on stream lines

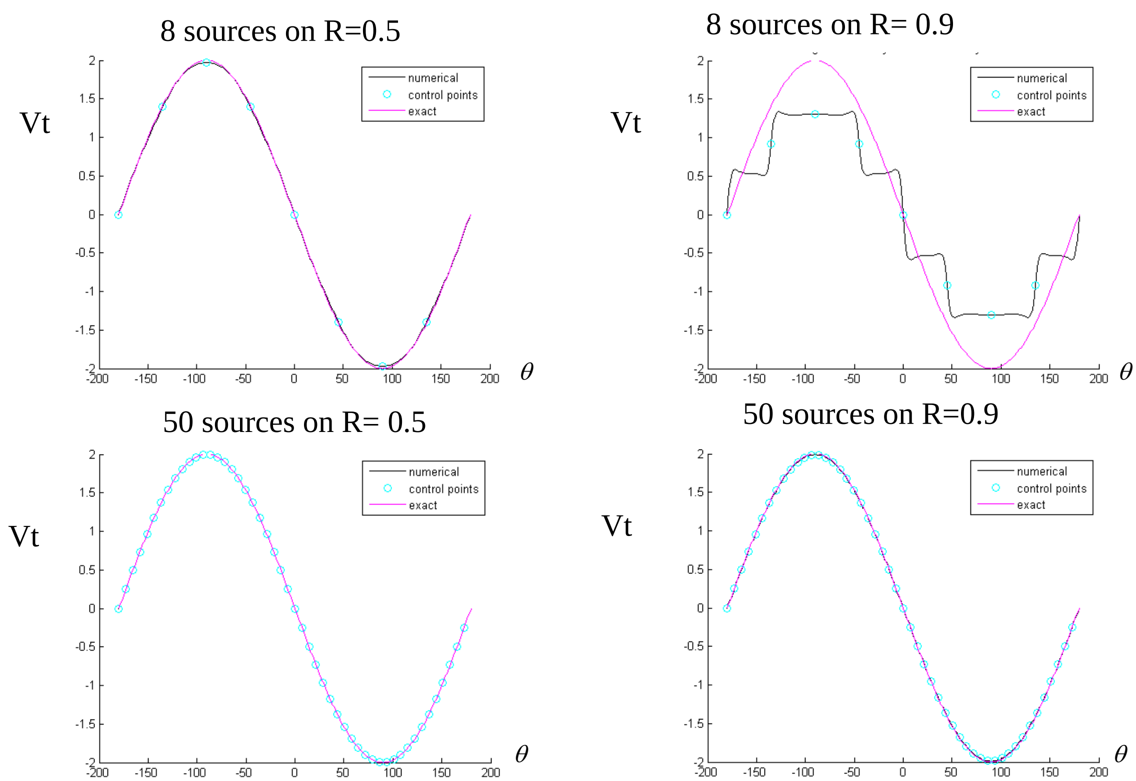


Figure 7 Effect of number and position of point sources on the centre line on tangential velocity

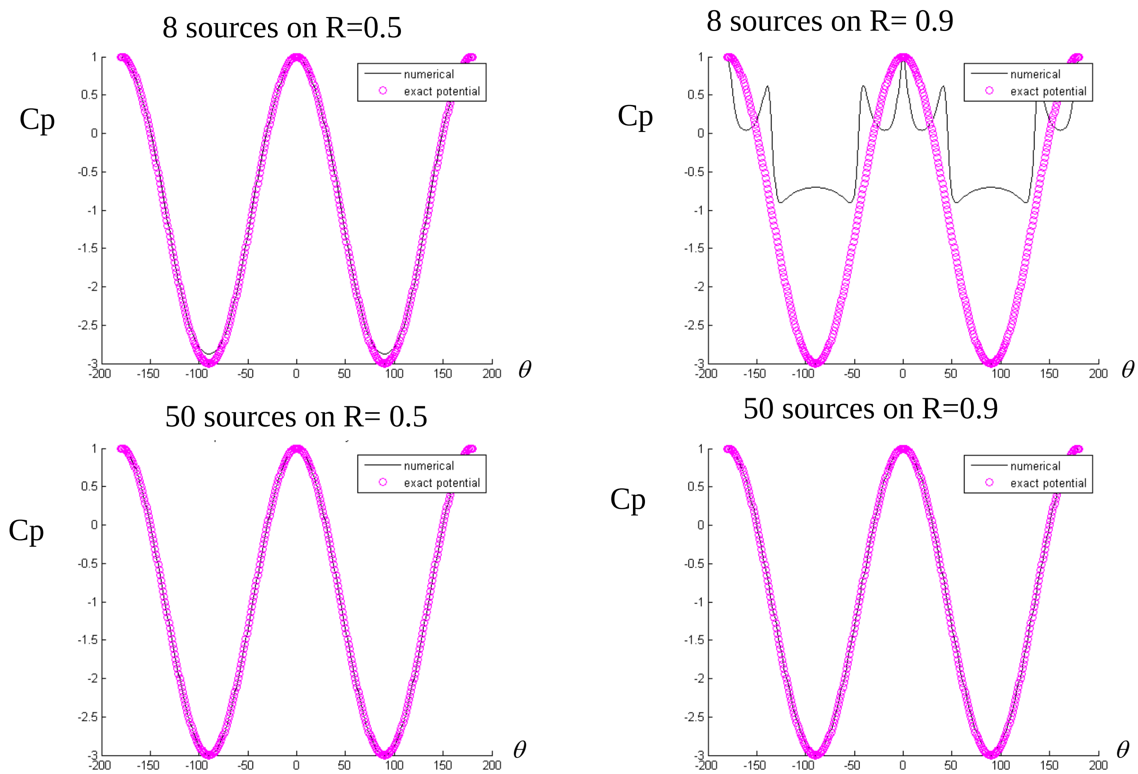


Figure 8 Effect of number and position of point sources on the centre line on Cp distribution

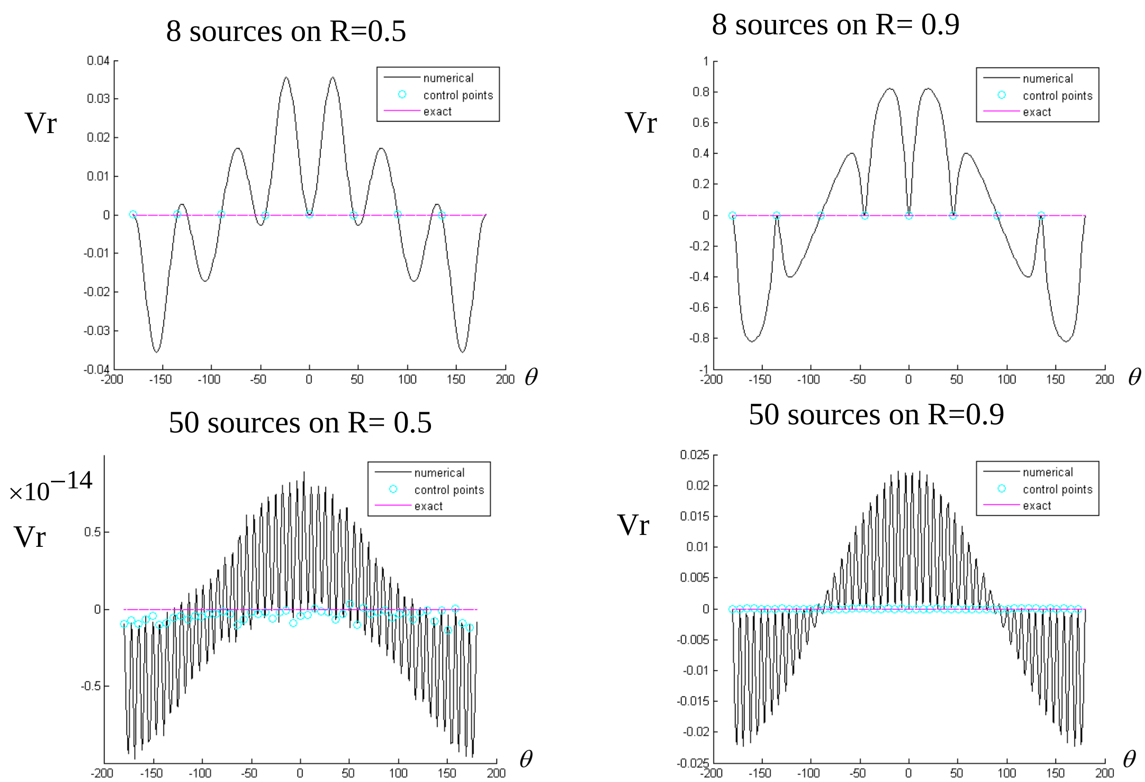


Figure 9 Effect of number and position of point sources on the centre line on radial velocity

It can be concluded from figures 6 to 9 that increasing the number of sources increases accuracy. This is particularly apparent in the radial velocity. For 50 sources the error achieved is exceptionally low –effectively zero to machine precision. Note in this figure that the accuracy with which the computer can enforce the zero radial velocity at control points is revealed.

These results presented represent just a snapshot of the possible solutions. Generally it can be concluded that

- more points leads to greater accuracy, but higher computational cost.
- small numbers of points close to the surface of the cylinder lead to inaccuracy due to the "lumpy" singularities distorting the flow field locally and affecting the physics in regions of the desired flow domain
- discrete sources on the centre line is an efficient method to model body thickness

Non-Lifting Flow Over A van de Vooren Aerofoil

A 15% thick van de Vooren aerofoil is considered. Initially the collocation points are located at the same x locations as the sources as illustrated for a small number of sources in Figure 10. An exact solution can be found for this aerofoil and the solution obtained from the discrete source method is shown by triangles in Figure 11. Clearly the solution is inaccurate compared with the analytic solution. If the first and last control or collocation points are moved closer to the leading and trailing edges then the solution shown as crosses in Figure 11 shows good agreement with the analytic solution. This illustrates the sensitivity of aerofoil solutions to the placement of control points relative to the leading and trailing edges.

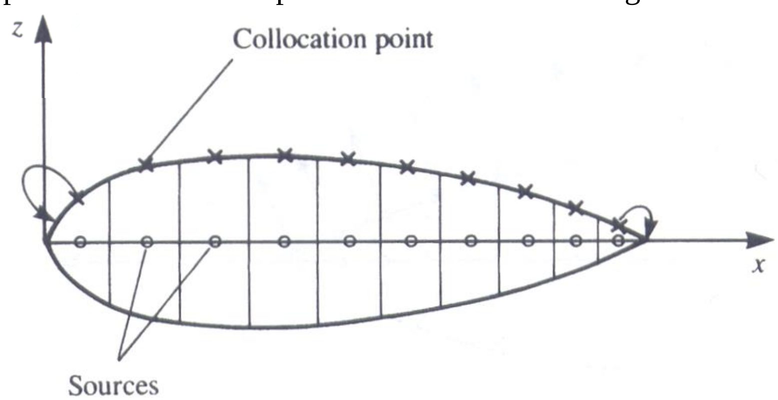


Figure 10 2D illustrative point source distribution on aerofoil

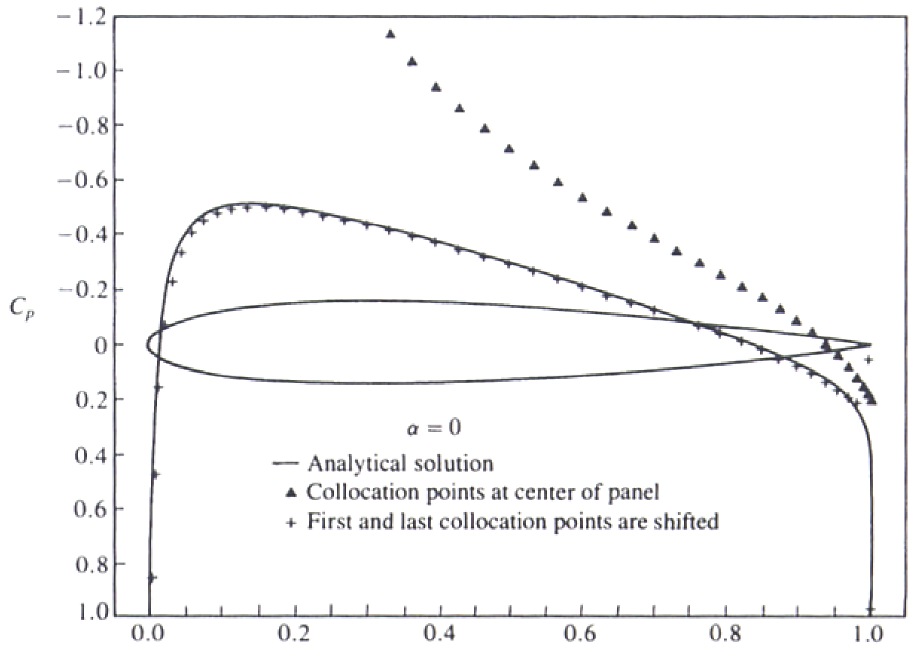


Figure 11 Calculated pressure distributions

3 PANEL METHODS

The 2D generalised models based on point singularities are OK for non-lifting bodies and thin aerofoils, however for thicker aerofoils they may prove inadequate near stagnation points. The 2D generalised models considered thus far have many similarities to 2D 'panel' methods; this is particularly so if the location of the singularities and control points is linked to modelling the geometry with 'panels'. However most methods, referred to in literature as 2D *'panel'* methods, do not use individual discrete point elements. Instead distributed elements are used where the sources, doublets and vortices are distributed continuously along a line. These distributed element *'panel'* models are better able to model realistic aerofoil shapes.

3.1 Background

In the late 1960's, a method was developed which made use of the rapid increases in computational power and speed to provide for the first time a method that genuinely had the capability to analyse flows around realistic geometries, and in many ways marked the birth of modern computational analysis. Although not generally classed as a full Computational Fluid Dynamics (CFD) technique, it shares many similarities including surface meshes and a numerical solution process, although unlike methods that solve the Navier-Stokes or Euler equations numerically the surrounding flow is not meshed. Panel methods utilise linear solutions of the potential function, and hence are limited to subsonic, (strictly incompressible, although compressibility corrections such as the Prandtl – Glauert compressibility correction can be applied to allow velocities up to M_{crit}, where supersonic flow first occurs), inviscid flow. Solutions produced by the panel method can then of course be coupled to a boundary layer solver. They offer a far more rapid solution method than alternative (more flexible) CFD techniques, whilst being considerably more accurate than hand calculations. As such they are still widely used in 'state of the art' versions, particularly where speeds are low (landing configurations, land vehicles).

3.2 Distributed Elements in 2D

These are essentially distributions of sources and vortices along a line in 2D; they are referred to as *'sheet'*, 'surface' or *'panel'* elements because a line source or line vortex in 2D is equivalent to a surface element in 3D (note a point source or point vortex in 2D is equivalent to a line source in 3D).

3.2.1 Distributed Source Elements

Consider an infinite number of sources distributed along a line. Let the distance along the line be s and let $\lambda(s)$ be the source strength per unit length along s. The strength of an infinitesimal element ds of the sheet is then

$$d\Lambda = \lambda(s)ds$$

This small infinitesimal section of the source sheet can be considered as a distinct source when working out the potential and stream function at a point ***P.***

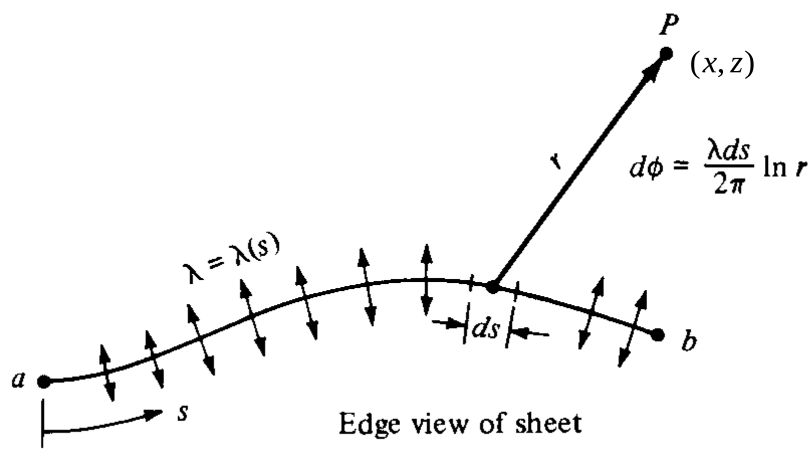


Figure 12 2D source sheet

Then the potential and stream function at ***P*** due to the small section of the sheet are given by

$$d\phi = \frac{\lambda ds}{2\pi} \ln r$$

$$d\psi = \frac{\lambda ds}{2\pi} \theta$$

The potential and stream function at ***P*** due to the whole sheet are then

$$\phi_P(x,z) = \int_a^b \frac{\lambda \ln r}{2\pi} ds \qquad (1)$$

$$\psi_P(x,z) = \int_a^b \frac{\lambda \theta}{2\pi} ds \qquad (2)$$

Analytic solutions can be derived for certain distributions and line shapes, however since the mathematics involved becomes complicated this will not be covered in this course.

3.2.2 Distributed Vortex Elements

Consider an infinite number of point vortices distributed along a line. Let the distance along the line be s and let $\gamma(s)$ be the vortex strength per unit length along s (note that sometimes $\gamma(s)$ is referred to as the vorticity distribution). The strength of an infinitesimal element ds of the sheet is then

$$d\Gamma = \gamma(s)ds$$

This small infinitesimal section of the vortex sheet can be considered as a distinct vortex when working out the potential, and stream function at a point ***P.***

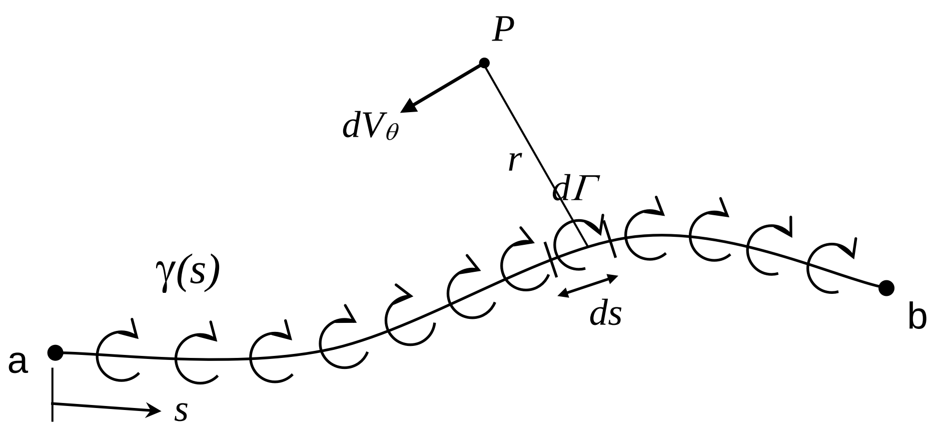


Figure 13 2D vortex sheet

Then the potential, stream function and (tangential) velocity at ***P*** due to the small section of the sheet are given by

$$d\phi = -\frac{\gamma ds}{2\pi}\theta$$

$$d\psi = \frac{\gamma ds}{2\pi}\ln r$$

$$dV_\theta = -\frac{\gamma ds}{2\pi r}$$

Note the velocity is needed for thin aerofoil theory covered later in the course.

The potential and stream function at ***P*** due to the whole sheet are then

$$\phi_p(x,z) = -\int_a^b \frac{\gamma\theta}{2\pi} ds \qquad (3)$$

$$\psi_p(x,z) = \int_a^b \frac{\gamma \ln r}{2\pi} ds \qquad (4)$$

The total circulation around finite length of the sheet is

$$\Gamma = \int_a^b \gamma(s)ds \tag{5}$$

and it can be shown (see for example J. Anderson, Fundamentals of Aerodynamics) that the discontinuity in velocity across the sheet or the local jump in tangential velocity is equal to the local sheet strength

$$v_{t1} - v_{t2} = \gamma(s) \tag{6}$$

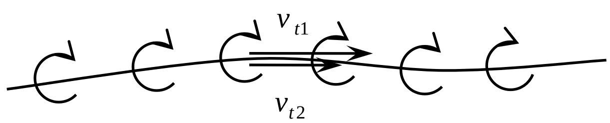


Figure 14 Velocity jump across vortex sheet

This approximates to the velocity distribution in a shear layer (except without viscous dissipation) e.g. wing wake, boundary layer and is used in numerical simulations of separated flow behaviour.

A 2D distributed vortex element can be replaced by a 2D distributed doublet element of appropriate doublet strength distribution, see for example Low Speed Aerodynamics by Katz and Plotkin where a 2D panel with a constant vortex distribution is shown to be equivalent to a 2D panel with a linear doublet distribution. Hence distributed 2D doublet elements, are not considered in detail here.

3.3 Source Panel Methods

Arbitrary, non-lifting, bodies can be modelled by covering the known surface in a source sheet and solving for $\lambda(s)$ so that the combined action of the source sheet and a free stream velocity makes the surface a streamline of the flow. This stream line models the solid surface since no flow can cross it.

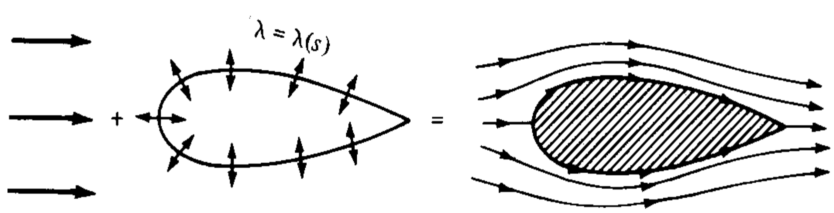


Figure 15 Modelling a 2D non-lifting flow with a source sheet

In practice an analytic solution is impossible and this process has to be carried out numerically by replacing the continuous sheet with a set of (usually) straight panels, where panel i has length Δs_i.

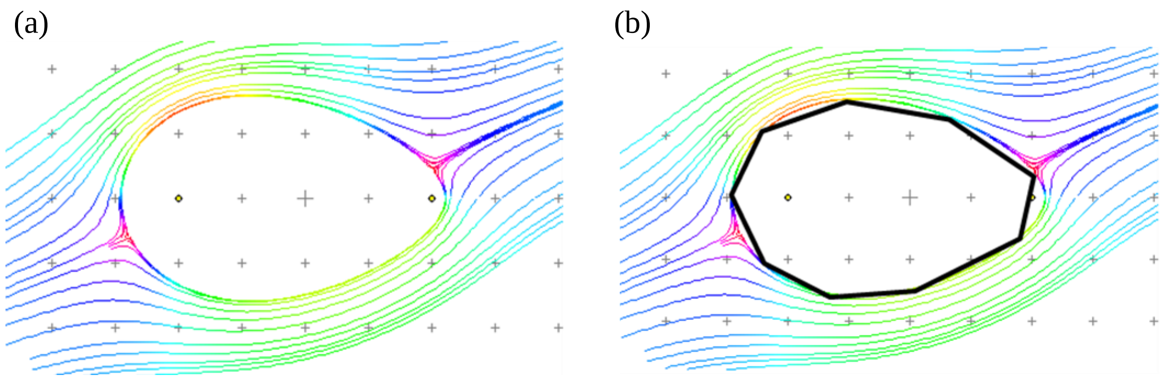


Figure16 (a) Actual body shape and (b) panel representation

The distribution of panels and hence the length of the panels Δs_i must be specified in a suitable way or inaccurate solutions will be obtained. For an aerofoil constant length panels are not suitable. More panels are needed at the leading and trailing edges. An example of a way to get more panels in these regions is the full cosine method. The *x* coordinate of the panel ends is found from the following equation

$$x = \frac{c}{2}(1 - \cos\beta)$$

where angle β takes values between 0 and π and is incremented by a fixed amount Δβ to get the panel locations, see Figure 17

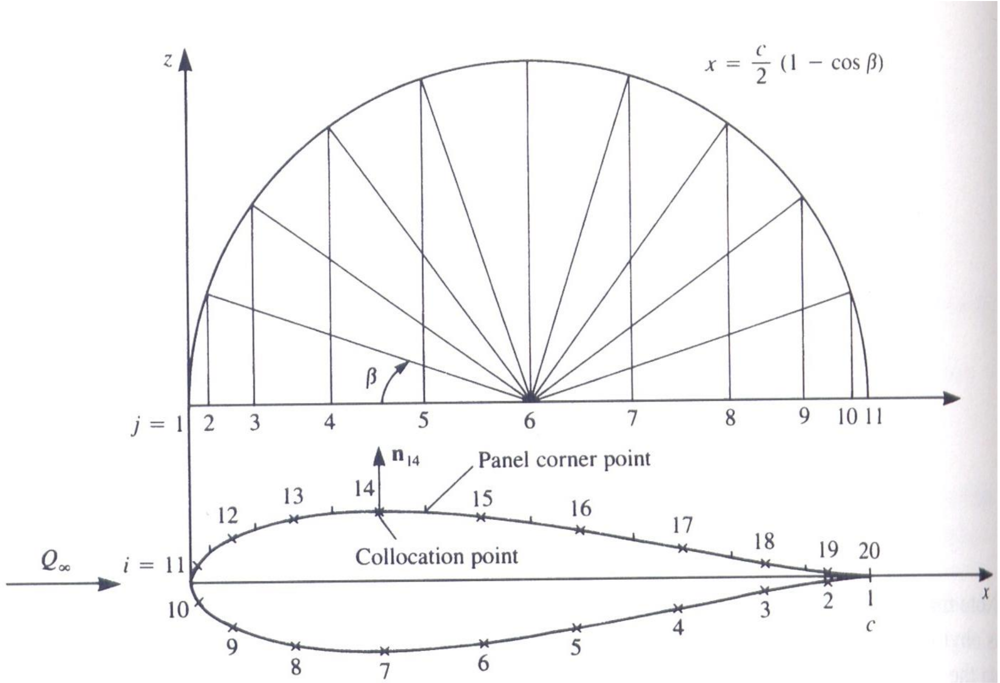


Figure 17 Full cosine method of distributing points on an aerofoil

Once the panel distribution has been established assumptions are made about the form of the source distribution $\lambda(s)$ on each panel to simplify the evaluation of the potential and velocities induced by a panel. Typically constant, linear or quadratic variations are assumed.

Then in order to obtain a set of equations to solve for the panel source strength a set of control points must be selected at which to apply a "surface is a streamline" condition, for example the panel mid points could be used.

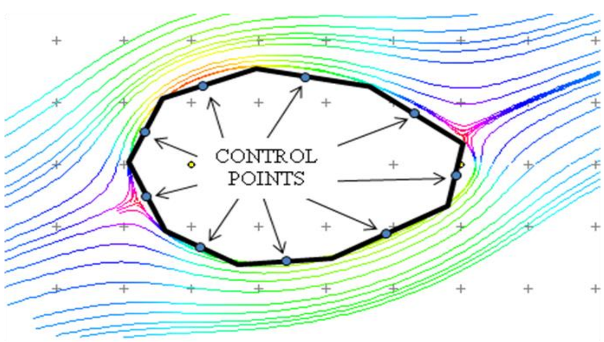


Figure18 control points on panels

3.31 Constant Source Panel Method

The simplest approach is to assume that $\lambda(s)$ is a constant on each of the N panels, but varies from panel to panel. This was one of the first successful panel methods to be developed. The basic method of constructing a solution is essentially the same as for the point source method described above.

First, in order to formulate a solution approach a choice been a Dirichlet or Neumann method must be made. The Dirichlet approach works directly with the potential computing the potential at control point *i* due to panel *j* using equation (1). The Neumann approach works with the velocity and is the approach implemented here. The surface normal component of velocity induced at control point *i* by panel *j* is evaluated and written as

$$a_{i,j}\lambda_j$$

where $a_{i,j}$ is called the velocity influence coefficient and λ_j is the source strength on panel *i*. Then the total normal velocity at control point *i* is found by summing all the contributions of every other panel (remember this is only possible because of solution linearity), i.e. the total normal contribution is

$$\sum_{j=1}^{N} a_{i,j}\ \lambda_j$$

Assuming we have a solid surface, we obviously want this to be equal to zero, i.e. (not forgetting the contribution of the onset flow, as the panel may be at any orientation!)

$$\sum_{j=1}^{N} a_{i,j}\ \lambda_j + \vec{V}_\infty \bullet \vec{n}_i = 0$$

where $\vec{V}_\infty$ is the free stream velocity vector and $\vec{n}_i$ is the normal to panel *i*. In this equation, the only unknowns are λ_i as all the other terms are either calculable from the known geometry ($a_{i,j}$ and $\vec{n}_i$) or specified in advance ($\vec{V}_\infty$). Then for each control point there is a similar equation

$$\sum_{j=1}^{N} a_{1,j}\ \lambda_j = -\vec{V}_\infty \bullet \vec{n}_1$$

$$\sum_{j=1}^{N} a_{2,j}\ \lambda_j = -\vec{V}_\infty \bullet \vec{n}_2$$

$$\vdots$$

$$\sum_{j=1}^{N} a_{n,j}\ \lambda_j = -\vec{V}_\infty \bullet \vec{n}_n$$

Then these equations for all the control points are combined into one matrix equation given by

$$A\ \vec{\lambda} = \vec{R}$$

where A is the (Velocity) Influence Matrix with entries $a_{i,j}$, $\vec{\lambda} = [\lambda_1, \lambda_2, \ldots, \lambda_N]^T$ is the vector of unknown source strengths and $\vec{R} = [R_1, R_2, \ldots R_N]^T$ on the right hand side of the equation contains terms from the surface normal component of the free stream. This is a linear equation which can be solved for the panel source strengths. Velocities, pressures and forces can then be evaluated from the model.

The influence matrix has N^2 entries, none of which are zero. It is generally not symmetric, and combined these two factors mean that this becomes a computationally demanding problem as the number of panels increases. For instance, Gaussian elimination, a matrix solution method with which you should be familiar, requires approximately $O(N^3)$ operations to invert the matrix. This then is not practical for large problems, as it takes too long. Instead of this, iterative techniques are used, and this is one of the areas which distinguish between different versions of the method, clever use of computational algorithms can give one method a speed or stability advantage over its competitors.

In an iterative approach, the equation

$$A\,\vec{\lambda} = \vec{R}$$

is solved by guessing an initial solution for the panel strengths , call it $\vec{\lambda}_1$, then generating an error vector $\vec{E}_1$:

$$\vec{E}_1 = \vec{R} - A\,\vec{\lambda}_1$$

This is used to generate a second guess,

$$\vec{\lambda}_2 = \vec{\lambda}_1 + f(\vec{E}_1)$$

and so on, until the error vector is as small as required. Each iteration takes $O(N^2)$ operations and for a 'good' iterative method, only about 10 iterations will be needed, so we have 10 matrix multiplications, i.e. $\approx 10N^2$ operations, which is smaller than the number of operations used in Gaussian elimination if N is large

3.31 Example Solutions

Non-Lifting Flow Over A Cylinder

A panel method solution for this case is shown in Figure 18. The solution was calculated using 8 equi-sized panels distributed around the entire cylinder.

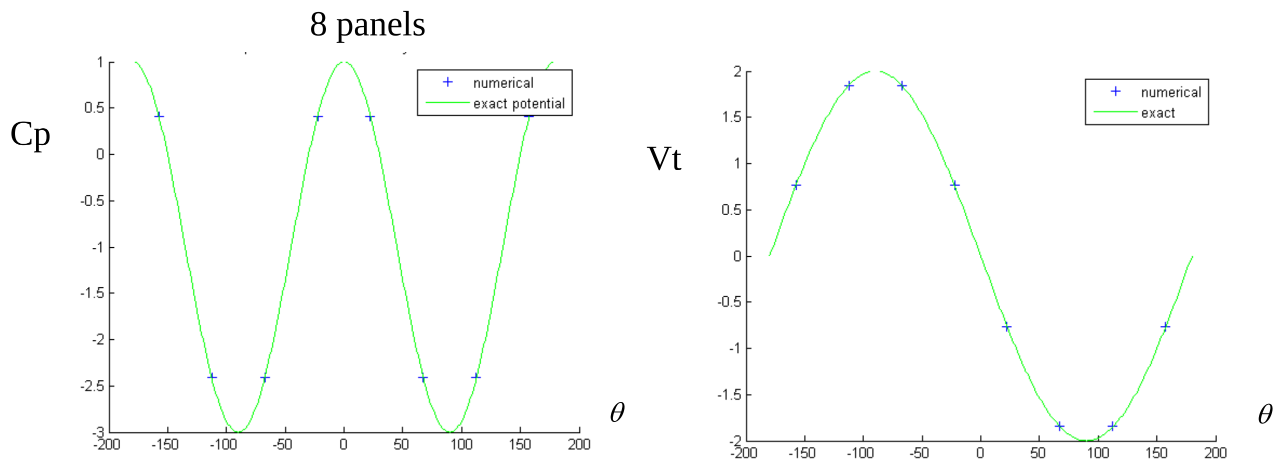


Figure 18 Constant source panel method solutions for surface pressure and tangential velocity.

It can be seen that the above solutions compare favourably to the 8 point source solutions with the sources near the surface shown in Figures 7 and 8. Whilst placing point sources on the centre line was more effective for this case, in general deciding for an arbitrary complex body where to place the sources may not be straightforward. The advantage of the panel method is that only the surface needs to be defined, though there are still issues of numbers of panels required for accurate solutions.

Non-Lifting Flow Over A van de Vooren Aerofoil

The 15% thick van de Vooren aerofoil is again considered. The solution obtained using constant source panels is shown in Figure 19 and agrees well with the exact solution.

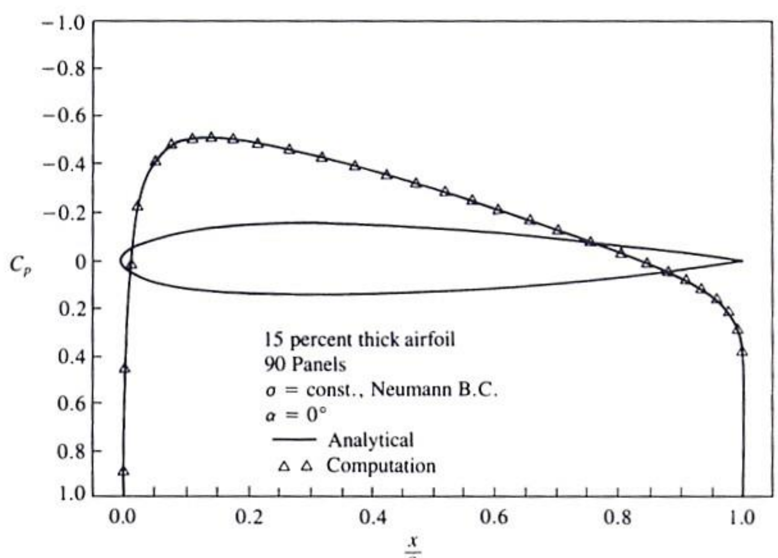


Figure 19 Pressure distribution constant source panels

3.32 Linear Source Panel Method

Another common approach used is to assume that $\lambda(s)$ varies linearly along a panel. If the strength at each end is known so that for panel *i*, the end strengths are λ_i and λ_{i+1} then for this panel

$$\lambda(s) = \lambda_i + \frac{(s - s_i)}{\Delta s_i}(\lambda_{i+1} - \lambda_i)$$

and the velocity induced at any point by panel *i*, is a function of both λ_i and λ_{i+1}. This slightly complicates the construction of the influence matrix *A*, but this is beyond the scope of this course.

3.4 2D Vortex Panel Methods

Arbitrary, lifting, bodies can be modelled by covering the known surface in a vortex sheet and solving for $\gamma(s)$ so that the combined action of the vortex sheet and a free stream velocity makes the surface a streamline of the flow. This stream line models the solid surface since no flow can cross it. For lifting flows over aerofoils it is necessary to apply an additional Kutta condition to fix the overall circulation and hence lift. There is no general analytic solution and so for arbitrary bodies numerical vortex panel methods are used. This is an industry standard method.

3.4.1 2D Constant Vortex Panel Method.

The method to construct a vortex panel method is essentially identical to that for the source panels, except that a Kutta condition must be imposed. If there are *N* panels with constant vortex strength γ_i then the normal velocity induced at each of the *N* control points is

$$a_{i,j} \ \gamma_j$$

Then applying the boundary condition of no normal flow at point *i* yields

$$\sum_{j=1}^{N} a_{i,j} \ \gamma_j + \vec{V}_\infty \bullet \vec{n}_i = 0$$

This gives *N* equations for the *N* unknowns. However the Kutta condition must be enforced, which requires that the upper and lower surface velocities at the trailing edge must be equal in magnitude or zero so that using equation (6)

$$v_{t1} - v_{t2} = \gamma_{TE} = 0$$

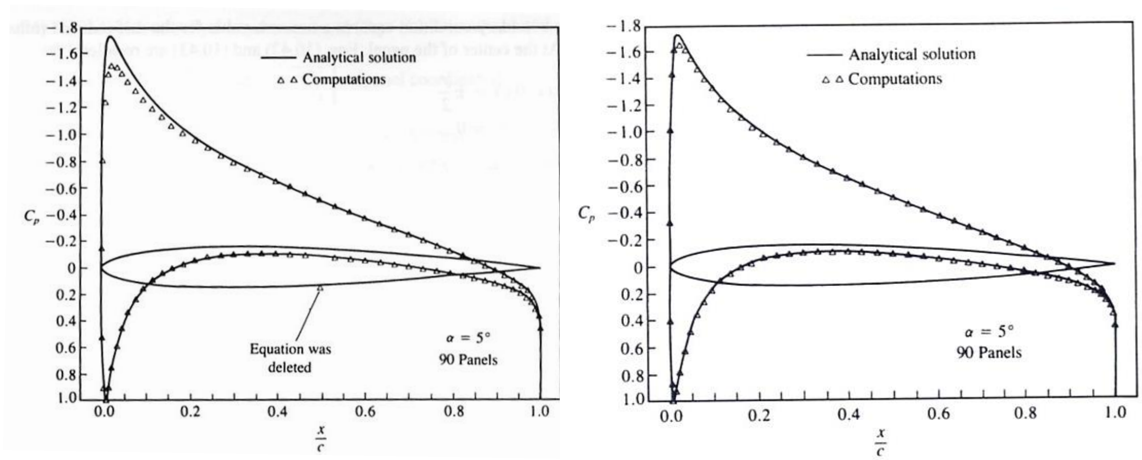
One way this can be achieved for the constant vortex panel method is by setting

$$\gamma_1 + \gamma_N = 0 \qquad (7)$$

Thus there are *N+1* equations and *N* unknowns. This can be solved using a least squares approach, or by modifying the imposition of the boundary conditions s and deleting one of the control point equations, away from the leading or trailing. You are not expected to know the details of this beyond the information given here. Note that if a linear vortex panel method was used (similar to that described above for source panels) then this complication would not arise if the Kutta condition (equation(7)) is applied, since there would by *N+1* unknowns; and from the no normal flow condition at control points and the Kutta condition; *N+1* equations.

3.4.2 Example Solution

The advantage of a 2D vortex panel method is that lifting flows over aerofoils can be calculated. An example solution is shown in Figure 20. Both have the same number of panels. It can be seen that the linear panels give a slightly better result near the leading edge, but note that exact agreement would not be expected due to the assumptions made in generating the panel method, namely ignoring viscous effects and compressibility.



Constant vortex panels

Linear vortex panels

Figure 20 Pressure distribution constant vortex and linear vortex

3.5 Final Thoughts

For both discrete singularity methods and panel methods using any type of distributed singularities the construction of a model has four stages

- 1. Geometry discretisation
- 2. Calculation of *influence coefficients* and *influence matrix* equation
- 3. Solution of the linear set of equations
- 4. Secondary calculations: pressures, forces, off-body velocities etc.

Whilst there are restrictions on the basic panel method due to the assumptions made for potential flows, modern panel methods add compressibility corrections and couple to boundary layer solvers to extend the range of applicability.

REVISION OBJECTIVES

You should be able to:

- Explain the basic concept behind generalised point singularity methods for obtaining potential models of the flow about arbitrary bodies.
- State which type of flow each singularity can be used for.
- Explain in detail the stages in the development of a generalised point source method.
- Explain what source sheet and vortex sheets are and develop expressions for the potential and stream function induced by the sheet. For a vortex sheet give expressions for the induced velocity, its total strength Γ is and how the velocity jump relates to the sheet strength.
- Explain that a vortex sheet element could be replaced by an equivalent doublet sheet element provided it has an appropriate distribution.
- Explain all the steps in the generation of a constant source panel method.
- Specify a linear source distribution in terms of the panel end values.
- Explain in overview the vortex sheet panel method.