

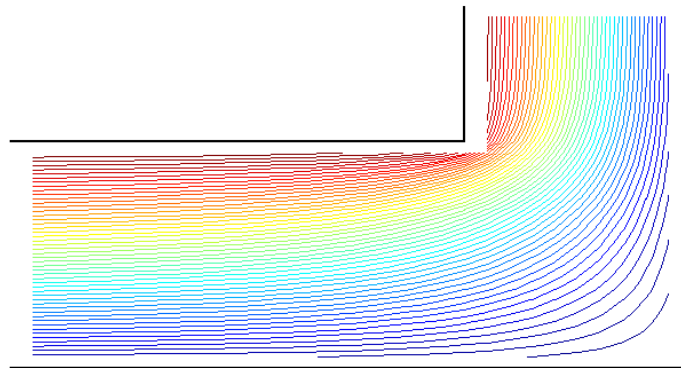
ENGR401

Introduction to Computational Fluid Mechanics

Dr. Mathieu Sellier

Assignment Two

FVM Simulation of a Potential Flow around a 90° Corner



Benjamin Munro

01st of May 2013

Introduction

The flow of an inviscid, irrotational fluid can be described by potential flow. Potential flow can be used to simplify more complicated flows in fluid dynamics and is used extensively in airfoil design. Two-dimensional, constant-density flows can also be described in terms of the stream function ψ , which is said to be constant along stream lines. We define the stream function as

$$u = \frac{\delta\psi}{\delta y}, v = -\frac{\delta\psi}{\delta x}$$

Governing Equation of Flow

Two dimensional, constant density flows may be described in terms of the *stream function* ψ , which is defined to be constant along stream lines. That means along a stream line we must have

$$d\psi = \frac{\delta\psi}{\delta x}dx + \frac{\delta\psi}{\delta y}dy = 0$$

Thus if we define the streamline to be

$$u = \frac{\delta\psi}{\delta y}, v = -\frac{\delta\psi}{\delta x}$$

the function ψ will be constant along stream lines. The stream function can be used to express rotational and irrotational flows; we can therefore use the condition of irrotationality as an extra constraint. Thus we have for an irrotational flow

$$\frac{\delta u}{\delta y} - \frac{\delta v}{\delta x} = \frac{\delta}{\delta y} \left(\frac{\delta\psi}{\delta y} \right) - \frac{\delta}{\delta x} \left(-\frac{\delta\psi}{\delta x} \right) = 0$$

Thus, the steady, incompressible, inviscid, irrotational flow equations may be cast simply as the Laplace equation for the stream function

$$\boxed{\frac{\delta^2\psi}{\delta x^2} + \frac{\delta^2\psi}{\delta y^2} = 0}$$

Boundary Conditions

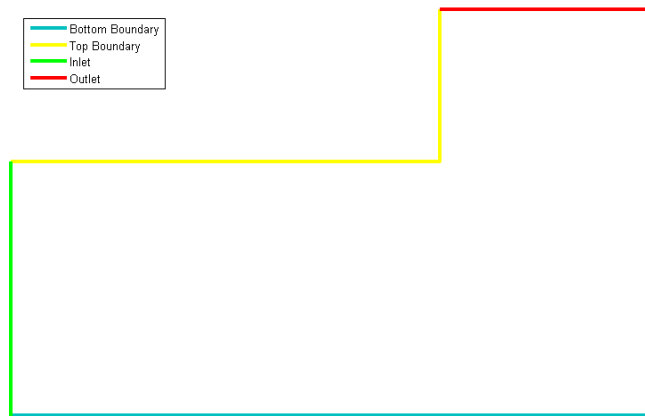


Figure 1 – Domain Boundaries

The boundary conditions were constructed by considering the properties of a potential flow and ensuring that continuity is satisfied. Figure 1 shows the individual boundaries.

- Bottom Boundary: At any point of a streamline, it is tangent to the line, i.e., there is no flow across a streamline, thus solid boundaries are considered lines of constant stream function. The stream function is set to $\psi(x, y) = 0$ at the bottom boundary.
- Top Boundary: As with the bottom boundary, the top boundary is considered a line of constant stream function. The stream function is set to $\psi(x, y) = 100$; the unit discharge at the bottom boundary.
- Inlet: The inlet boundary condition is a specified flow velocity in the x direction ($u = 10\text{ms}^{-1}$) with no flow in the y direction
- Outlet: To ensure that continuity is satisfied, the total mass flux going out of the domain is set equal to the mass flux entering the domain. Thus the outlet plane velocity is set to 20 ms^{-1} . The velocity is assumed to have a zero x component at the outlet boundary.

Discretization of the Governing Equation

The first step of the finite volume method is to divide the domain into discrete control volumes. A portion of the 2-dimensional grid used to discretize the governing equation is shown in figure 2; this shows an internal node surrounded by the control volume.

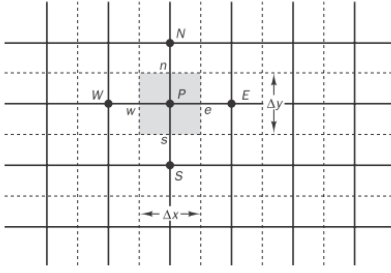


Figure 2- Part of the two dimensional grid

When the governing equation is integrated over the control volume we obtain

$$\int_{\Delta V} \frac{\delta}{\delta x} \left(\frac{\delta \psi}{\delta x} \right) dx \cdot dy + \int_{\Delta V} \frac{\delta}{\delta x} \left(\frac{\delta \psi}{\delta y} \right) dx \cdot dy + \int_{\Delta V} S_{\phi} dV = 0$$

where S_{ϕ} is the source. With A the cross-sectional area of the control face, we note that

$A_e = A_w = \Delta y$ and $A_n = A_s = \Delta x$ to obtain

$$\left[A_e \left(\frac{\delta \psi}{\delta x} \right)_e - A_w \left(\frac{\delta \psi}{\delta x} \right)_w \right] + \left[A_n \left(\frac{\delta \psi}{\delta y} \right)_n - A_s \left(\frac{\delta \psi}{\delta y} \right)_s \right] + \bar{S}_{\phi} \Delta V = 0$$

This equation represents the balance of the ψ in the control volume and the flux through the cell faces. Using the central difference approximation, we can write expressions for the flux through each of the control faces

$$\text{Flux across the west face} = A_w \left(\frac{\delta \psi}{\delta x} \right)_w = A_w \frac{(\psi_P - \psi_W)}{\delta x_{WP}}$$

$$\text{Flux across the east face} = A_e \left(\frac{\delta \psi}{\delta x} \right)_e = A_e \frac{(\psi_E - \psi_P)}{\delta x_{EP}}$$

$$\text{Flux across the south face} = A_s \left(\frac{\delta \psi}{\delta x} \right)_s = A_s \frac{(\psi_P - \psi_S)}{\delta x_{SP}}$$

$$\text{Flux across the north face} = A_n \left(\frac{\delta \psi}{\delta x} \right)_n = A_n \frac{(\psi_N - \psi_P)}{\delta x_{NP}}$$

Substituting these equations into equations into equation we obtain

$$\left[A_e \frac{(\psi_E - \psi_P)}{\delta x_{EP}} - A_w \frac{(\psi_P - \psi_W)}{\delta x_{WP}} \right] + \left[A_n \frac{(\psi_N - \psi_P)}{\delta x_{NP}} - A_s \frac{(\psi_P - \psi_S)}{\delta x_{SP}} \right] + \bar{S}_{\phi} \Delta V = 0$$

This equation can be rearrange with the linearization of the source term ($\overline{S_\phi} \Delta V = S_u + S_p \psi_P$) into the following form

$$\begin{aligned} & \left(\frac{A_e}{\delta x_{EP}} + \frac{A_w}{\delta x_{WP}} + \frac{A_n}{\delta x_{NP}} + \frac{A_s}{\delta x_{SP}} - S_p \right) \psi_P \\ &= \frac{A_e}{\delta x_{EP}} \psi_E + \frac{A_w}{\delta x_{WP}} \psi_W + \frac{A_n}{\delta x_{NP}} \psi_N + \frac{A_s}{\delta x_{SP}} \psi_S + S_u \end{aligned}$$

Arranging this equation into the standard discretized equation form for interior nodes:

$$a_p \psi_P = a_e \psi_E + a_w \psi_W + a_n \psi_N + a_s \psi_S + S_u$$

were

$$\begin{aligned} a_e &= \frac{A_e}{\delta x_{EP}}, & a_w &= \frac{A_w}{\delta x_{WP}}, & a_n &= \frac{A_n}{\delta x_{NP}}, & a_s &= \frac{A_s}{\delta x_{SP}}, \\ a_p &= (a_e + a_w + a_n + a_s - S_p) \end{aligned}$$

At the boundary where the values of the stream function are known, the discretized equations are modified to incorporate the boundary conditions. The flux entering across the boundary is introduced as a source term in the matrix equations. The inlet and exit boundary ψ values were calculated using difference in flow rate planar to the stream line.

Solution Algorithm

- Mesh Generation

The first step of a finite volume method solution is to set up the mesh. The mesh is formed by dividing the geometry shown in figure 4 into discrete control volumes. The nodes are situated within the middle of the control volume, and thus surrounded by the faces of the control volume. This can be seen in figure 2 for a general point P. Initially the distance between nodes was set to 1 m, this was then refined to 0.25m as can be seen in figure 3.

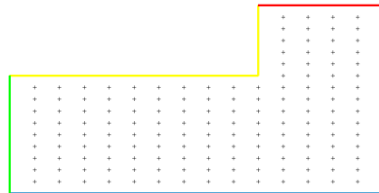


Figure 3.1 Coarse Mesh $dy, dx = 1m$

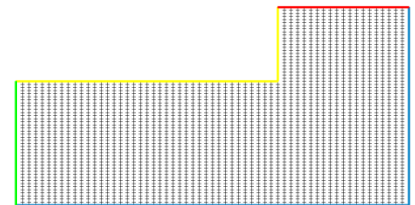


Figure 3.2 Refined Mesh $dy, dx = 0.25m$

- Equation Generation

Discretised equations of the form of equation 2.2 must be set up for each nodal point to solve the problem. For control volumes next to the boundaries, the general equation is modified to incorporate the boundary conditions as previously seen in section two.

The A matrix of the system is set up using conditional statements as a 'for' loop runs through the points of the discrete mesh. This places the appropriate values in the correct position for the A and b matrix at each node. The full MATLAB code for these operations can be found in the appendices.

- Solving the System of Equations

The system of equations is solved using a direct method, LU decomposition of the sparse matrix A to solve the system. For this system of equations it was decided that it would be more economical to solve using a direct solver rather than iterative methods. The cost of solving this matrix is $\frac{2}{3}n^3$ floating point operations, with n being a maximum of 2757 for a mesh size of 0.25m, was not considered to be computationally expensive. Residuals were calculated using the equation $Res = b - A \cdot \psi$, the max was found to be in the order of 10^{-12} .

Mesh Independence

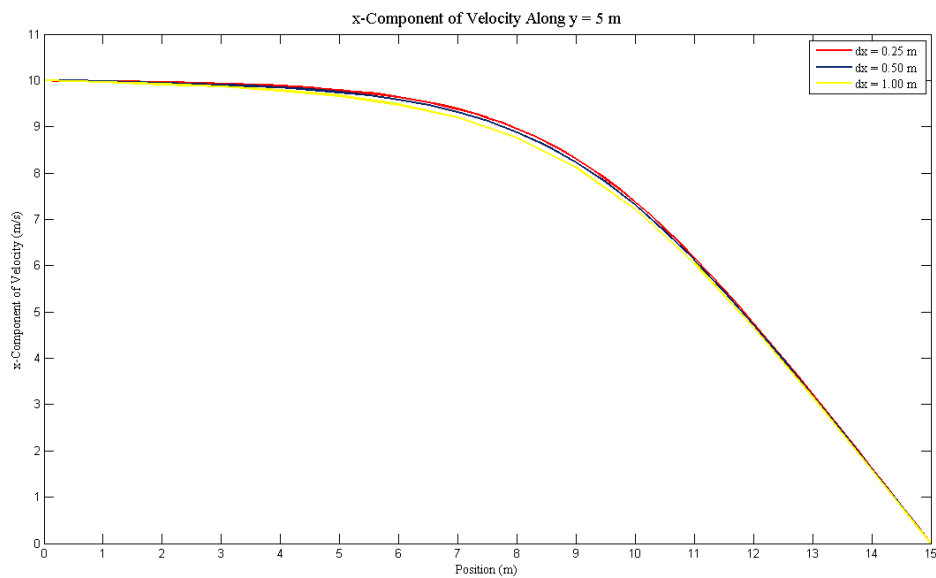


Figure 4 – Plot of x direction velocity along y = 5

<i>Distance Along x direction (m)</i>	<i>2</i>	<i>4</i>	<i>6</i>	<i>8</i>	<i>10</i>	<i>12</i>	<i>14</i>
<i>Velocity with $dx = 1m$</i>	<i>9.9632</i>	<i>9.8641</i>	<i>9.6642</i>	<i>9.1974</i>	<i>7.2192</i>	<i>5.0942</i>	<i>2.0123</i>
<i>Velocity with $dx = 0.25m$</i>	<i>9.9680</i>	<i>9.8934</i>	<i>9.6829</i>	<i>9.0816</i>	<i>7.6311</i>	<i>5.1254</i>	<i>2.0236</i>
<i>Percentage error</i>	<i>0.048</i>	<i>0.29</i>	<i>0.19</i>	<i>0.23</i>	<i>5.3</i>	<i>0.95</i>	<i>1.13</i>

Table 1 - Percentage error of velocity between

To investigate the dependency of the solution on the mesh size, the velocity along the x direction at $y = 5$ was calculated for 3 different mesh refinements. Table 1 and Figure four show that the solution is independent of the mesh, thus that our discretization error is small.

Streamlines

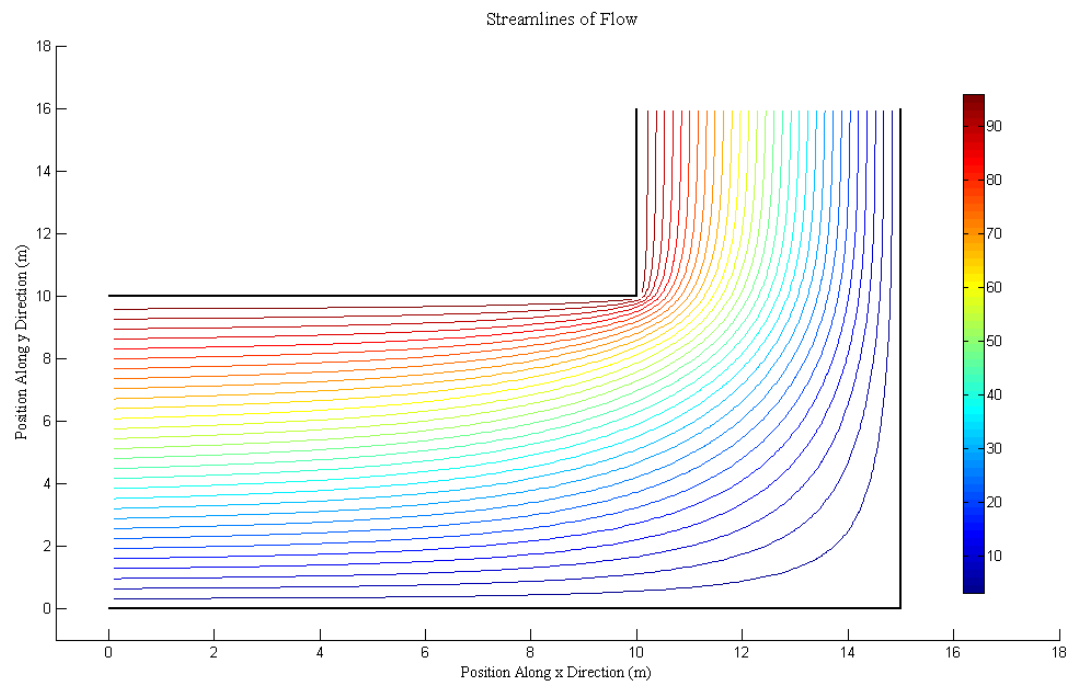


Figure 5 – Plot of Flow Streamlines at $dx = 0.25$

Figure 5 displays the constant streamlines of the fluid field at a mesh size of 0.25 m

Pressure Coefficient on the Wall of the Top and Bottom Boundaries

The pressure coefficient is an important metric in fluid flows. This coefficient is a dimensionless number that describes the relative pressure throughout a flow field. With the assumption of our flow field, namely incompressibility and steady state flow, we can describe the pressure coefficient by

$$C_p = 1 - \left(\frac{V}{V_\infty}\right)^2$$

The plots displayed in figures 6 and 7 shows the calculated values of the pressure coefficient along the top and bottom boundaries.

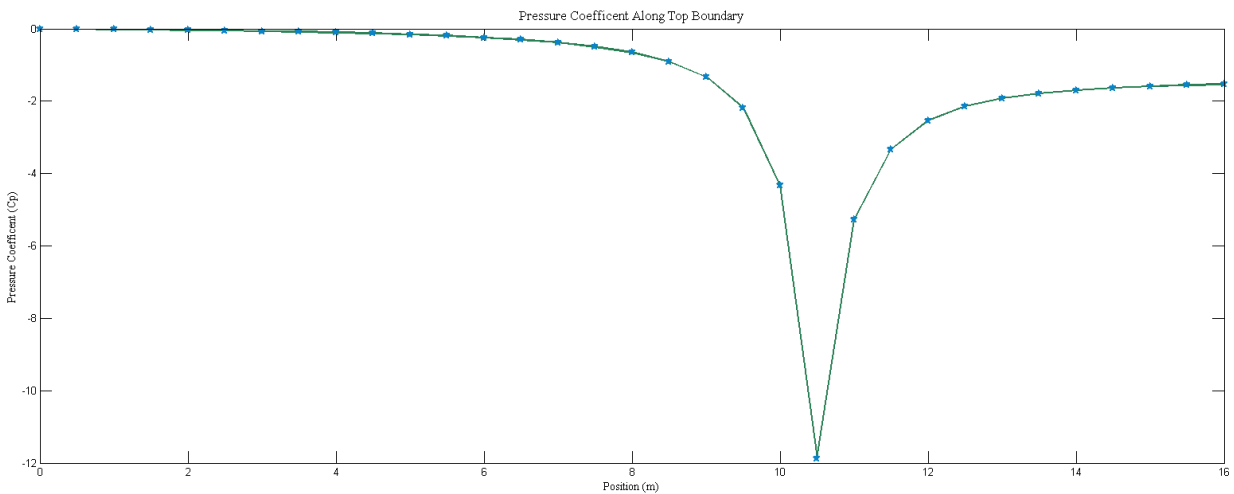


Figure 6 – Plot of Pressure Coefficient along the Top Boundary

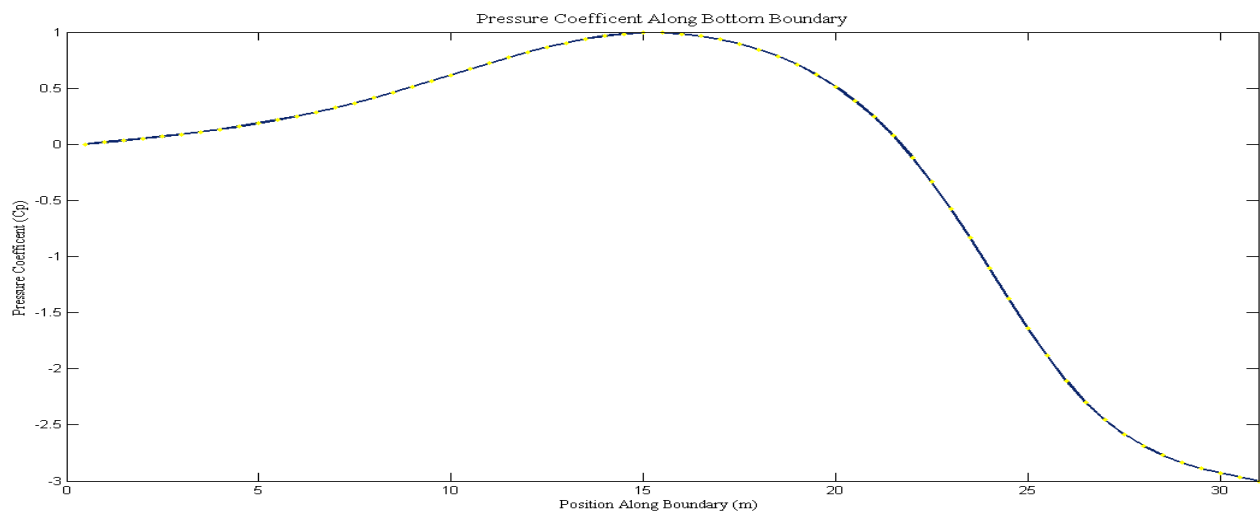


Figure 7 – Plot of Pressure Coefficient along Bottom Boundary

Limitations of the Model

- Errors

The possibilities for error source for this CFD model are numerical errors, coding errors and user errors. Coding errors are easy to make and can be hard to find within a large code, making it difficult to decipher if something is wrong with the code.

- Uncertainties

To simplify the model several assumptions were made in describing the fluid flow, considerable solution economy can be achieved from treating the flow as steady, incompressible, inviscid and two dimensional and irrotational. The deficiencies due to these simplifications of the flow will cause uncertainties in the results.

At the boundaries the calculation across the face of the control volume is reduced to first order accuracy due to taking the backwards difference approximation instead of the second order central difference method applied to the interior nodes. A first order approximation was also used to calculate the gradient between the stream lines.

- Lack of Verification and Validation

To quantify the errors and uncertainties in the CFD validation and verification should be carried out to confirm the results. As this was not completed, the solution cannot be full trusted.

