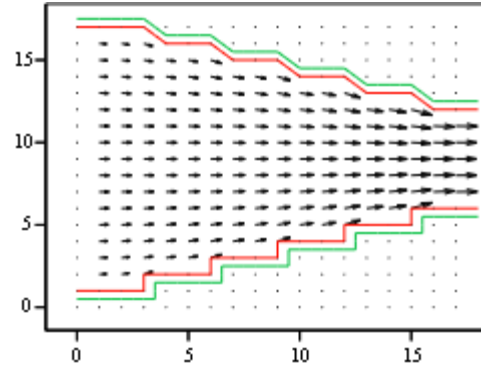


## Fluid flow I: The potential function

One of the equations describing the flow of a fluid is the continuity equation:

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot \rho \vec{u} = 0$$

where  $\rho$  is the fluid density and  $\mathbf{u}$  is the flow velocity. Many fluids, including water, are considered incompressible which results in  $\rho$  being constant and thus the divergence of the velocity  $\mathbf{u}$  is zero.



$(U_x, U_y), BC$

### Flow in a nozzle

The electric field  $\mathbf{E}$  in vacuum has no divergence and the corresponding electrostatic potential  $\phi$  satisfies Laplace's equation. In two dimensions, solutions to Laplace's equation for the potential can be found on a grid by a relaxation method in which the potential at each point is replaced by the average of the potential at the four neighboring points. In this exercise we will solve for the flow of an incompressible fluid by making an analogy with electrostatics. We will assume that the fluid velocity vector  $\mathbf{u}$  can be found from a potential  $\Phi$  that satisfies Laplace's equation:

$$\vec{u} = \vec{\nabla} \Phi \quad \vec{\nabla} \cdot \vec{u} = \nabla^2 \Phi = 0$$

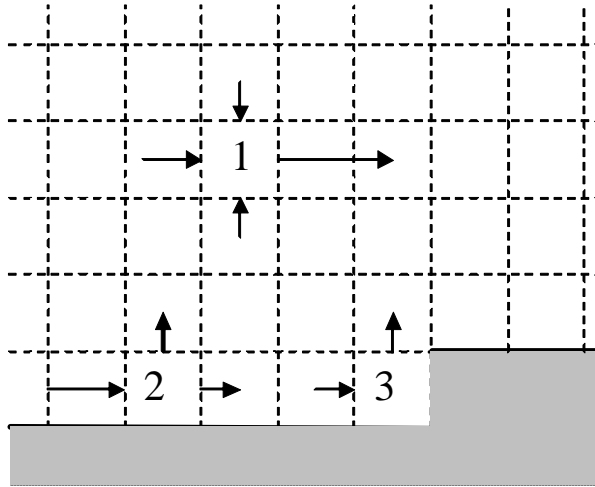
A solution found in this way is not the only solution that is possible. Fluid can circulate clockwise or counterclockwise around an object in a flow. The sum of two solutions is also a solution thus a solution with flow from left to right can be added to a solution with simple rotation to make a new solution. The uniqueness of solutions is discussed in reference 1.

We will create a rectangular grid and begin with a higher potential at the right boundary than at the left boundary. This potential profile will create a gradient so that  $\mathbf{u}$  points to the right. If, for example, the grid has 20 points and the potential increases by 20 from left to right, the average length of a velocity vector will be about 1 unit. In this exercise, the variables are dimensionless.

### Incompressible flow on a grid

The drawing below shows a set of rectangular cells that have grid points at their centers. First consider the cell labelled 1 which is in the fluid. The flow into and out of the cell is indicated by arrows representing the flow speed through the cell wall. The sum of the magnitudes of the vectors pointing in must equal the magnitude of the vectors pointing out. If the flow potential at a point is  $\Phi_{i,j}$ , then the flow into cell 1 from the left is  $u = \Phi_{i,j} - \Phi_{i,j-1}$  if the grid spacing  $\Delta x = 1$ . The vector points to the right if the potential on the right is higher, which is "backwards" from the relationship of  $\mathbf{E}$  to  $\phi$ . If  $\Phi_{i,j}$  is equal to the average of the  $\Phi$  values in the four neighboring cells, then it is easy to show that the divergence of the flow is zero. The relaxation method will average a cell with the four surrounding cells until this relationship is satisfied with a very small error.

Now consider the cell 2 that is on the boundary. The boundary is a solid wall and there is no flow across that boundary. Then  $\Phi_{i,j}$  for this cell is the average of the three surrounding cells that are in the fluid. Similarly the potential of cell 3 is the average of the two cells that are in the surrounding fluid if the flow is to be divergenceless. These considerations suggest an easy way to implement the averaging. A Boolean matrix  $BC_{i,j}$  (for boundary conditions) is defined in which the values are 1 (true) when the cell is in the fluid and 0 (false) when the cell is not in the fluid.  $BC_{i,j}$  is zero when a cell is in the boundary or within an object in the flow.



If  $BC_{i,j}$  for a cell is true, then the new value of  $\Phi$  for that cell is found from an average of the surrounding cells and if false the cell is ignored. The surrounding cells  $(i+1,j)$ ,  $(i-1,j)$ ,  $(i,j-1)$  and  $(i,j+1)$  are included in the averaging if they have their corresponding BC value as 1 (true), otherwise they are omitted from the sum. In the averaging, the number of cells with  $BC = 1$  is the denominator.

### **The grid**

There is no reason to assign values to the  $x_i$  and  $y_j$  that are on the grid. It is sufficient to know the number of  $x$  grid points  $i_{max}+1$ , the number of  $y$  grid points  $j_{max}+1$ , and the grid spacings are  $\Delta x$  and  $\Delta y$ .

$i_{max} := 18$       The number of  $x$  grid points. These are on the horizontal axis.

$j_{max} := 18$       The number of  $y$  grid points. These are on the vertical axis.

The subscripts will have the values:  $i := 0..i_{max}$        $j := 0..j_{max}$

$\Delta x := 1$        $\Delta y := 1$       The grid spacing is assumed to unity in each direction.

### **The boundary**

The domain will be a duct in which fluid flows to the right and which is tapered so that the duct has a smaller opening at the right end. The continuity equation then requires that the fluid leave with a higher velocity than it entered.

The Boolean boundary matrix BB will have the points outside the boundaries of the duct ( $j = 0$  and  $j = j_{\max}$ , for example) set equal to zero so that these cells are not part of the averaging process. If they were, the occurrence of  $i, j+1$  (for example) in the sum would result in an error because  $j+1$  is undefined at  $j = j_{\max}$ .

The top and bottom of the duct containing the flow are given a slope so that the cross section is reduced at the right end, thus creating a nozzle.

$$\text{BB} := \begin{cases} \text{BB}_{i_{\max}, j_{\max}} \leftarrow 0 \\ \text{for } i \in 0..i_{\max} \\ \quad \text{for } j \in 0..j_{\max} \\ \quad \quad \text{BB}_{i,j} \leftarrow 1 \\ \quad \quad \text{BB}_{i,j} \leftarrow 0 \text{ if } \frac{j}{j_{\max}} < \frac{i}{3 \cdot i_{\max}} \vee \frac{j_{\max} - j}{j_{\max}} < \frac{i}{3 \cdot i_{\max}} \vee j = 0 \vee j = j_{\max} \\ \text{BB} \end{cases}$$

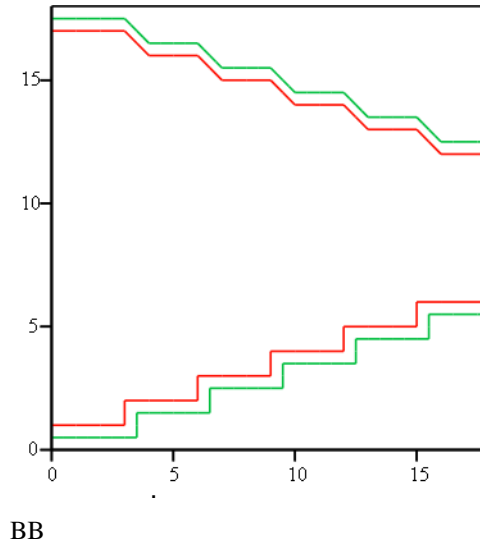
If the transpose of the matrix is displayed, the  $i$  direction is horizontal and the  $j$  direction is vertical. This display puts  $i$  and  $j$  on the same axes as on contour and vector field plots, but the display is upside down relative to the plots. This is not obvious because there is up-down symmetry.

$$\text{BB}^T = \begin{array}{c|cccccccccccccccccccc} & 0 & 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 & 9 & 10 & 11 & 12 & 13 & 14 & 15 & 16 & 17 & 18 \\ \hline 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & 1 & 1 & 1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 2 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 3 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 4 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 5 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 0 & 0 & 0 \\ 6 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 \\ 7 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 \\ 8 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 \\ 9 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 & \dots \end{array}$$

The values above and the plot below show that the nozzle is tapering so that it has a smaller opening at the right hand side.

## Plot of the nozzle

The contour plot of the matrix BB at the right shows that the fluid boundary is a nozzle that is decreasing in cross section. Keep in mind that we are working in two dimensions, which means that the duct extends infinitely into and out of the page.



## The program loop

The program loop is similar to the one used for solving Laplace's equation. The main difference is that surrounding points are not part of the average if their BB value is zero. The number of iterations should be several times the number of grid points ( $i_{\max} \cdot j_{\max}$ ).

iters := 3 ·  $i_{\max} \cdot j_{\max}$       iters = 972

Iters is the number of iterations.

```

Phi(BB) :=
  Phiimax,jmax ← 0
  for j ∈ 0 .. jmax
    for i ∈ 0 .. imax
      Phii,j ←  $\frac{-i_{\max}}{2} + i$ 
  Avg ← Phi
  for k ∈ 0 .. iters
    for i ∈ 1 .. imax - 1
      for j ∈ 1 .. jmax - 1
        Avgi,j ←  $\frac{BB_{i-1,j} \cdot \Phi_{i-1,j} + BB_{i+1,j} \cdot \Phi_{i+1,j} + BB_{i,j-1} \cdot \Phi_{i,j-1} + BB_{i,j+1} \cdot \Phi_{i,j+1}}{BB_{i-1,j} + BB_{i+1,j} + BB_{i,j-1} + BB_{i,j+1}}$  if BBi,j = 1
        Avgi,j ← 0.5 · (Phii-1,j + Phii+1,j) otherwise
  Phi ← Avg
Phi
```

Avg is a temporary matrix in which the average is stored. If the grid spacing were not unity, we would have to use a weighted average rather than a simple average of the Phi values to obtain Avg. The line labelled "otherwise" is used on points that are not in the fluid. This line creates a potential in the "dead space" that has contours that are nearly vertical and are continuations of the contours in the fluid. This averaging creates a more pleasant appearance for the contour plots. These points are not in the fluid thus their potential value has no physical significance.

The program loop is made a function of BB so that we can repeat the calculation with different boundaries and not have to write again the program loop.

The first line at right from the program loop assigns initial "guess" values to the potential Phi. The guess potential increases from -imax/2 at the left boundary to imax/2 at the right boundary. The gradient is one unit per cell which creates velocity vectors about one unit in length. The second line "calls" the function Phi(BB) and puts the value in the matrix Phi1 for plotting.

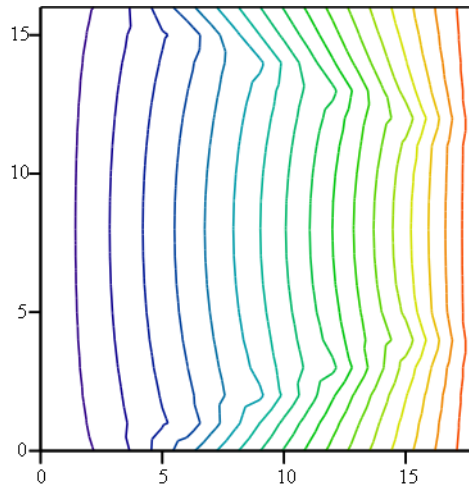
$$\Phi_{i,j} \leftarrow \frac{-i_{max}}{2} + i$$

$$\Phi1 := \Phi(BB)$$

The top and bottom rows of the answer matrix Phi are regions that are not fluid. The submatrix command at the right removes these two lines so that they do not affect the contour plot.

```
Phi2 := submatrix(Phi1,0,imax,1,jmax - 1)
```

Plot of potential contours



The potential plot shows that the potential indeed increases from left to right. The contour lines are closer together at the right side indicating a steeper potential gradient and a larger fluid velocity.

The contour lines outside the boundaries of the duct have no physical significance.

Below, part of the answer matrix Phi1 is displayed with the values set to zero that are not in the fluid.

Phi2

	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14
0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
1	-9	-8.5	-8.1	-7.7	0	0	0	0	0	0	0	0	0	0	0
2	-9	-8.5	-7.9	-7.4	-6.7	-6.1	-5.7	0	0	0	0	0	0	0	0
3	-9	-8.4	-7.8	-7.2	-6.6	-5.9	-5.3	-4.4	-3.7	-3.2	0	0	0	0	0
4	-9	-8.4	-7.8	-7.1	-6.4	-5.8	-5	-4.3	-3.5	-2.7	-1.7	-0.8	-0.2	0	0
5	-9	-8.4	-7.7	-7	-6.3	-5.6	-4.9	-4.1	-3.3	-2.4	-1.5	-0.6	0.4	1.6	2.7
6	-9	-8.3	-7.7	-7	-6.3	-5.5	-4.8	-4	-3.1	-2.3	-1.3	-0.4	0.7	1.8	3
7	-9	-8.3	-7.6	-6.9	-6.2	-5.5	-4.7	-3.9	-3	-2.1	-1.2	-0.2	0.9	2	3.2
8	-9	-8.3	-7.6	-6.9	-6.2	-5.4	-4.6	-3.8	-3	-2.1	-1.1	-0.1	1	2.1	3.3
9	-9	-8.3	-7.6	-6.9	-6.2	-5.4	-4.6	-3.8	-2.9	-2	-1.1	-0.1	1	2.2	...

$$\left[ (\Phi1 \cdot BB) \right]^T =$$

### Plot of the velocity vector field

The vector components of  $\mathbf{u}$  are obtained from the finite difference form of the definitions. The definitions at the right are centered at the point  $i,j$ .

$$u_x = (\Phi_{i+1,j} - \Phi_{i-1,j}) / 2\Delta x$$

$$u_y = (\Phi_{i,j+1} - \Phi_{i,j-1}) / 2\Delta y$$

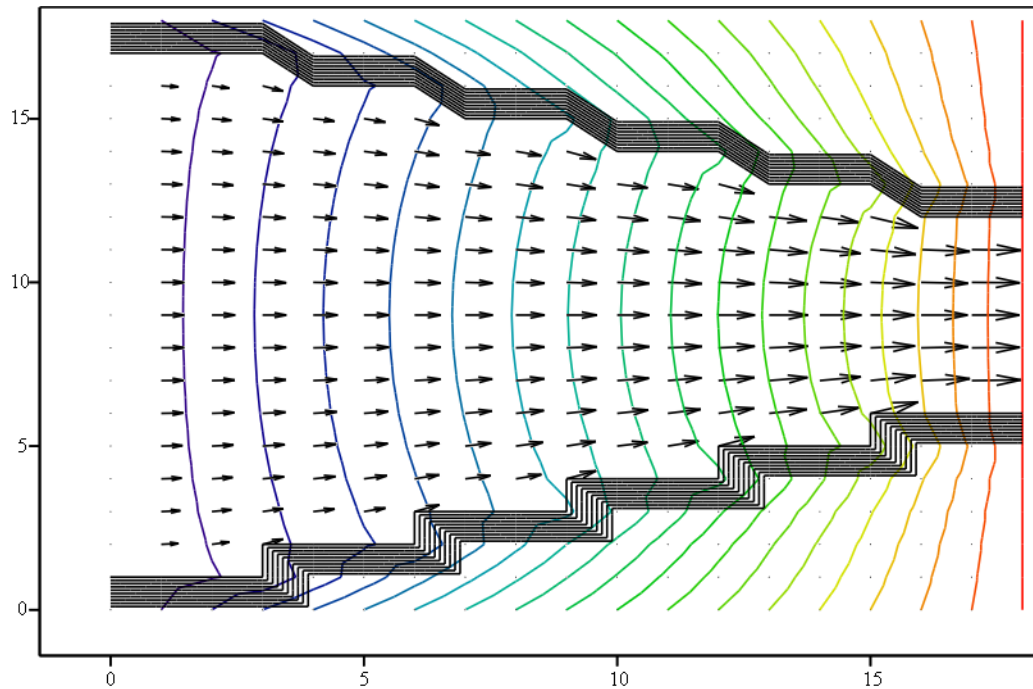
The range of the subscripts  $ii$  and  $jj$  must not include the end points because  $i+1$ ,  $i-1$  etc. are used in the formulas. The derivatives are centered at grid points not on the boundary. The derivative is changed to zero (through multiplication by  $BB_{i,j}$ ) if any of the four points used are not in the fluid.

$$ii := 1 .. imax - 1 \quad jj := 1 .. jmax - 1$$

$$U_{x,ii,jj} := \frac{1}{2 \cdot \Delta x} \cdot [(\text{Phi1}_{ii+1,jj}) - (\text{Phi1}_{ii-1,jj})] \cdot (BB_{ii,jj-1} \cdot BB_{ii,jj+1} \cdot BB_{ii-1,jj} \cdot BB_{ii+1,jj})$$

$$U_{y,ii,jj} := \frac{1}{2 \cdot \Delta y} \cdot [(\text{Phi1}_{ii,jj+1}) - (\text{Phi1}_{ii,jj-1})] \cdot (BB_{ii,jj-1} \cdot BB_{ii,jj+1} \cdot BB_{ii-1,jj} \cdot BB_{ii+1,jj})$$

Three contour plots: Vector field  $\mathbf{u}$ , flow potential  $\Phi$ , and boundary Boolean matrix



(Ux, Uy), Phi1, BB

Note that the vectors are longer at the small end of the duct indicating that the flow velocity has increased. Below we will check that the flow into the large opening is equal to the flow out of the small opening. The boundary is drawn as contour lines.

### Conservation of fluid

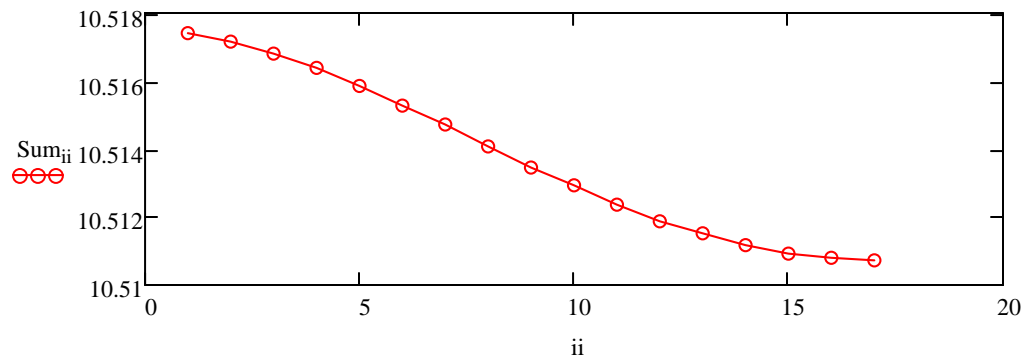
The net flow into the duct is found by integrating the velocity  $u_x$  across the opening at the left boundary. The integration is simply a sum over the index  $i$  of the  $u_x$  values at the left boundary ( $j = 0$ ). We will calculate the left-to-right flow across each vertical column in the matrix using forward finite differencing.

$$U_{x,ii,jj} := \left[ (\text{Phi1}_{ii+1,jj}) - (\text{Phi1}_{ii,jj}) \right] \cdot (\text{BB}_{ii,jj} \cdot \text{BB}_{ii+1,jj}) \quad \text{The flow to the right at each grid point}$$

$$\text{Sum}_{ii} := \sum_{jj} U_{x,ii,jj}$$

The sum over  $jj$  values at horizontal position  $ii$ . In the definition of  $U_x$ , the value is set to zero (through multiplication by  $\text{BB}_{ii,jj}$ ) if the point is not in the fluid.

### Integrated flow at each cross section



Note the vertical scale. The fluid crossing each cross section is very nearly the same.

$$\frac{\text{Sum}_1}{\text{Sum}_{j_{\max}-1}} = 1.0006357$$

There is strong conservation of fluid. The integrated flow into the duct is very nearly equal to the flow out of the duct.

### Flow around an object

An object can be placed in the flow by defining a set of points in the  $\text{BB}$  matrix that have value 0 which indicates that there is no fluid. No other changes are necessary.

The new definition of BB that includes an oval object on the midplane

$$\text{BB} := \begin{cases} \text{BB}_{i_{\max}, j_{\max}} \leftarrow 0 \\ \text{for } i \in 0 \dots i_{\max} \\ \quad \text{for } j \in 0 \dots j_{\max} \\ \quad \quad \text{BB}_{i,j} \leftarrow 1 \\ \quad \quad \text{BB}_{i,j} \leftarrow 0 \text{ if } j < \frac{i}{4} \vee (j_{\max} - j) < \frac{i}{4} \vee j = 0 \vee j = j_{\max} \\ \quad \quad \text{BB}_{i,j} \leftarrow 0 \text{ if } \left[ \left( \frac{i_{\max}}{3} - i \right)^2 + \left( \frac{j_{\max}}{2} - j \right)^2 \right] < \left( \frac{j_{\max}}{9} \right)^2 \end{cases}$$

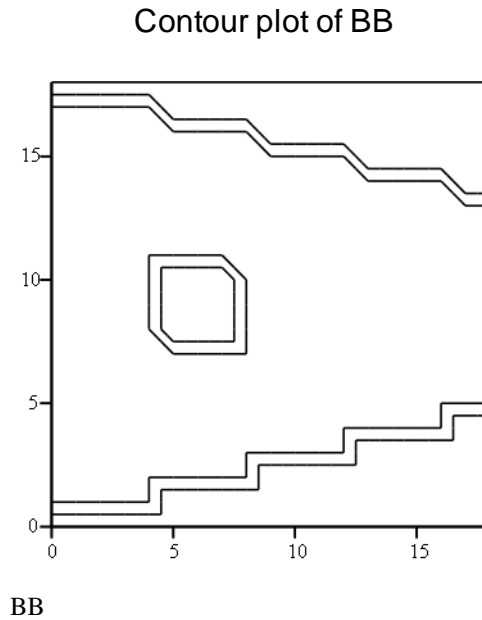
The object is centered at  $(i_{\max}/3, j_{\max}/2)$  and has radius  $j_{\max}/9$ .

At right is a contour map of the matrix BB indicating the boundaries of the region with fluid.

The oval object is turned into a polygon because the number of grid points is not sufficient to resolve the details. Also, the duct has smooth walls, not stair-stepped walls.

Create a new matrix of potential values using the current BB:

$$\text{Phi1} := \text{Phi}(\text{BB})$$



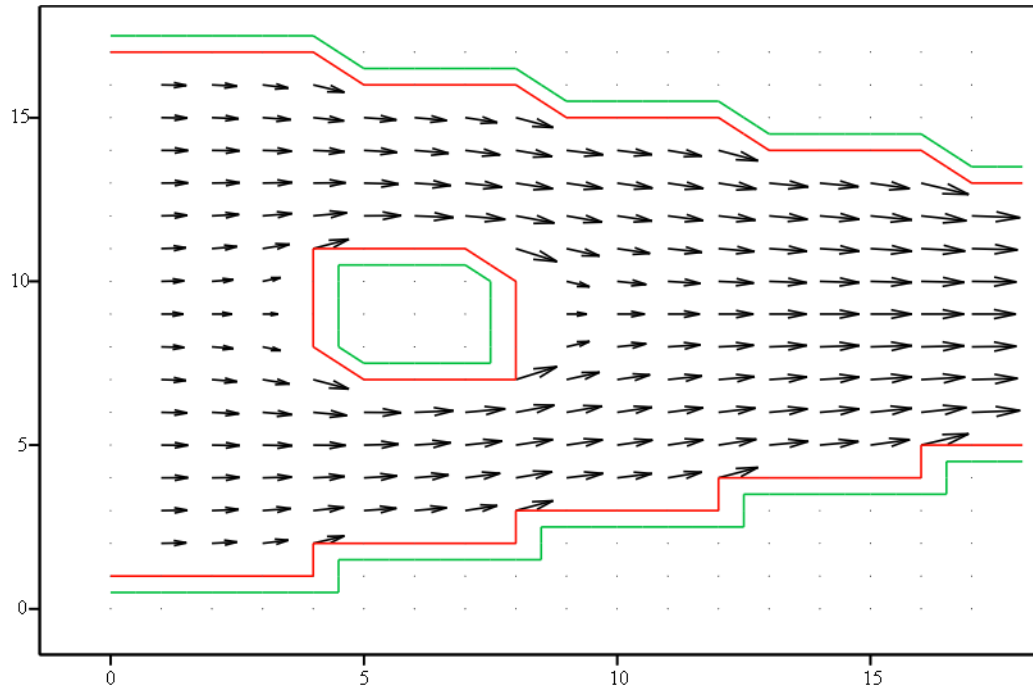
Create new matrices of fluid velocity components:

$$U_{x_{ii}, jj} := 0.5 \cdot \left[ \left( \text{Phi1}_{ii+1, jj} \right) - \left( \text{Phi1}_{ii-1, jj} \right) \right] \cdot \left( \text{BB}_{ii, jj-1} \cdot \text{BB}_{ii, jj+1} \cdot \text{BB}_{ii-1, jj} \cdot \text{BB}_{ii+1, jj} \right)$$

$$U_{y_{ii}, jj} := 0.5 \cdot \left[ \left( \text{Phi1}_{ii, jj+1} \right) - \left( \text{Phi1}_{ii, jj-1} \right) \right] \cdot \left( \text{BB}_{ii, jj-1} \cdot \text{BB}_{ii, jj+1} \cdot \text{BB}_{ii-1, jj} \cdot \text{BB}_{ii+1, jj} \right)$$



Plot of the fluid flow vectors in nozzle with an oval object



$(U_x, U_y), BB$

Is there conservation of fluid? To answer this question first define the flow vectors at each cross section:

$$U_{x,ii,jj} := \left[ (\text{Phi1}_{ii+1,jj}) - (\text{Phi1}_{ii,jj}) \right] \cdot (\text{BB}_{ii,jj} \cdot \text{BB}_{ii+1,jj}) \quad \text{The flow to the right at each grid point.}$$

$$\text{Sum}_{ii} := \sum_{jj} U_{x,ii,jj} \quad \text{The sum over values at each column in the plot.}$$

$$\frac{\text{Sum}_1}{\text{Sum}_{j_{\max}-1}} = 1.0002549 \quad \text{At left is the ratio of the flow at the left boundary to the flow at the right boundary. There is strong conservation of fluid.}$$

**Try it:** Change the number of iterations from  $3 \cdot i_{\max} \cdot j_{\max}$  to double that number and observe the improvement in conservation of fluid.

**Try it:** Remove the taper from the boundary and increase the size of the obstacle.

**Try it:** Rewrite the exercise so that  $\Delta x = \Delta y = 1$  is not required.

### Reference

A. M. Kuethe and J. D. Schetzer, *Foundation of Aerodynamics* (Wiley, New York, 1967), chapter 2.