10

(*i*, *j*)

(1, 6) *y* = 1 m (6, 6)

*j*

45

*i*

(11, 1)

*y* = 0 m

(16, 1)

m/s

(1, 11) *y* = 2 m (16, 11)

### 

|  |
| --- |
|  |
|  |
|  |
|  |
|  |

5

|  |
| --- |
|  |
|  |
|  |
|  |
|  |
|  |
|  |
|  |
|  |
|  |
|  |
|  |

m/s

**Fig. 8.30** Numerical model of po- tential flow through a two-dimen- sional 45° expansion. The nodal points shown are 20 cm apart.

There are 45 boundary nodes and 91 internal nodes.

1 m 1 m 1 m

**Fig. 8.31** Stream-function nodal val- ues for the potential flow of Fig.

All these boundary values must be input to the program and are shown printed in Fig. 8.31.

Initial guesses are stored for the internal points, say, zero or an average value of 5.0 m2/s. The program then starts at any convenient point, such as the upper left (2, 10), and evaluates Eq. (8.110) at every internal point, repeating this sweep iteratively until there are no further changes (within some selected maximum change) in the nodal values. The results are the finite-differ- ence simulation of this potential flow for this mesh size; they are shown printed in Fig. 8.31 to three-digit accuracy. The reader should test a few nodes in Fig. 8.31 to verify that Eq. (8.110) is satisfied everywhere. The numerical accuracy of these printed values is difficult to estimate, since there is no known exact solution to this problem. In practice, one would keep decreasing the mesh size to see whether there were any significant changes in nodal values.

This problem is well within the capability of a small personal computer. The values shown in Fig. 8.31 were obtained after 100 iterations, or 6 min of execution time, on a Macintosh SE personal computer, using BASIC.

|  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
|  = 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 | 10.00 |
| 8.00 | 8.02 | 8.04 | 8.07 | 8.12 | 8.20 | 8.30 | 8.41 | 8.52 | 8.62 | 8.71 | 8.79 | 8.85 | 8.91 | 8.95 | 9.00 |
| 6.00 | 6.03 | 6.06 | 6.12 | 6.22 | 6.37 | 6.58 | 6.82 | 7.05 | 7.26 | 7.44 | 7.59 | 7.71 | 7.82 | 7.91 | 8.00 |
| 4.00 | 4.03 | 4.07 | 4.13 | 4.26 | 4.48 | 4.84 | 5.24 | 5.61 | 5.93 | 6.19 | 6.41 | 6.59 | 6.74 | 6.88 | 7.00 |
| 2.00 | 2.02 | 2.05 | 2.09 | 2.20 | 2.44 | 3.08 | 3.69 | 4.22 | 4.65 | 5.00 | 5.28 | 5.50 | 5.69 | 5.85 | 6.00 |
|  = 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 1.33 | 2.22 | 2.92 | 3.45 | 3.87 | 4.19 | 4.45 | 4.66 | 4.84 | 5.00 |
|  |  |  |  |  |  | 0.00 | 1.00 | 1.77 | 2.37 | 2.83 | 3.18 | 3.45 | 3.66 | 3.84 | 4.00 |

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
|  | | 0.00 | 0.80 | 1.42 | 1.90 | 2.24 | 2.50 | 2.70 | 2.86 | 3.00 |
|  |  |  | 0.00 | 0.63 | 1.09 | 1.40 | 1.61 | 1.77 | 1.89 | 2.00 |

|  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- |
| 8.30. Boundary values are known in- | 0.00 | 0.44 | 0.66 | 0.79 | 0.87 | 0.94 | 1.00 |
| puts. Internal nodes are solutions to  Eq. (8.110). |  | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 | 0.00 |

Although Fig. 8.31 is the computer solution to the problem, these numbers must be manip- ulated to yield practical engineering results. For example, one can interpolate these numbers to sketch various streamlines of the flow. This is done in Fig. 8.32*a*. We see that the streamlines are curved both upstream and downstream of the corner regions, especially near the lower wall. This indicates that the flow is not one-dimensional.

The velocities at any point in the flow can be computed from finite-difference formulas such as Eqs. (8.106) and (8.107). For example, at the point (I, J) = (3, 6), from Eq. (8.107), the hor- izontal velocity is approximately

†(3, 7) — †(3, 6) 2.09 — 0.00

*u*(3, 6) ≈ —— = —— = 10.45 m/s

Δ*y* 0.2

and the vertical velocity is zero from Eq. (8.106). Directly above this on the upper wall, we es- timate

†(3, 11) — †(3,10) 10.00 — 8.07

*u*(3, 11) ≈ —— = —— = 9.65 m/s

Δ*y* 0.2

*p*1 *V*1

 = 10

8

6

4

2

0

*C* =

*p* – *p*1

*p*

*V* /2

2

1

0.75

Upper surface

One-dimensional approximation Eq. (1)

Lower surface

(*a*)

**Fig. 8.32** Useful results computed from Fig. 8.31: (*a*) streamlines of the flow; (*b*) pressure-coefficient distribution along each wall.

1.0

0.8

0.6

0.4

0.2

0.0

– 0.2

– 0.4

– 0.6

– 0.8

(*b*)

The flow is not truly one-dimensional in the entrance duct. The lower wall, which contains the diverging section, accelerates the fluid, while the flat upper wall is actually decelerating the fluid. Another output function, useful in making boundary-layer analyses of the wall regions, is the pressure distribution along the walls. If *p*1 and *V*1 are the pressure and velocity at the entrance (I =

1. , conditions at any other point are computed from Bernoulli’s equation (8.3), neglecting gravity

*p* + 1 p*V*2 = *p*1 + 1 p*V*2

1

—— ——

2 2

which can be rewritten as a dimensionless pressure coefficient

*p* — *p*1 *V* 2

—— p*V*1 (*V* )

*Cp* = —1

2

2

= 1 —

—

1

This determines *p* after *V* is computed from the stream-function differences in Fig. 8.31.

Figure 8.32*b* shows the computed wall-pressure distributions as compared with the one- dimensional continuity approximation *V*1*A*1 ≈ *V*(*x*)*A*(*x*), or

*Cp*(one-dim) ≈ 1 — (—)

*A*1

*A*

2

(1)

The one-dimensional approximation, which is rather crude for this large (45°) expansion, lies between the upper and lower wall pressures. One-dimensional theory would be much more ac- curate for a 10° expansion.

Analyzing Fig. 8.32*b*, we predict that boundary-layer separation will probably occur on the lower wall between the corners, where pressure is strongly rising (highly adverse gradient). There- fore potential theory is probably not too realistic for this flow, where viscous effects are strong. (Recall Figs. 6.27 and 7.8.)

Potential theory is *reversible*; i.e., when we reverse the flow arrows in Fig. 8.32*a*, then Fig. 8.32*b* is still valid and would represent a 45° *contraction* flow. The pressure would fall on both walls (no separation) from *x* = 3 m to *x* = 1 m. Between *x* = 1 m and *x* = 0, the pressure rises on the lower surface, indicating possible separation, probably just downstream of the corner.

This example should give the reader an idea of the usefulness and generality of numerical analysis of fluid flows.

# The Boundary-Element Method

### A relatively new technique for numerical solution of partial differential equations is the *boundary-element method* (BEM). Reference 7 is an introductory textbook outlin- ing the concepts of BEM, including FORTRAN programs for potential theory and elas- tostatics. There are no interior elements. Rather, all nodes are placed on the boundary of the domain, as in Fig. 8.33. The “element’’ is a small piece of the boundary sur- rounding the node. The “strength’’ of the element can be either constant or variable.

For plane potential flow, the method takes advantage of the particular solution

### 1

†\* = —

### 2π

ln 1

*r*

—

### (8.112)

which satisfies Laplace’s equation, ∇2† = 0. Each element *i* is assumed to have a dif- ferent strength †*i*. Then *r* represents the distance from that element to any other point in the flow field. Summing all these elemental effects, with proper boundary condi-

### tions, will give the total solution to the potential-flow problem.

At each element of the boundary, we typically know either the value of † or the value of †/*n*, where *n* is normal to the boundary. (Mixed combinations of † and

*n* Element *j*

Eleme



Node *j*

nt *i*

Node *i*

*rj*

*ds*

Domain:

2  = 0

**Fig. 8.33** Boundary elements of constant strength in plane potential flow.

### †/*n* are also possible but are not discussed here.) The correct strengths †*i* are such that these boundary conditions are satisfied at every element. Summing these effects over *N* elements requires integration by parts plus a careful evaluation of the (singu-

lar) effect of element *i* upon itself. The mathematical details are given in Ref. 7. The result is a set of *N* algebraic equations for the unknown boundary values. In the case of elements of constant strength, the final expression is

— †*i* + Σ †*j* (∫ — *ds*) = Σ (—) (∫ †\* *ds*) *i* = 1 to *N* (8.113)

1

2

*N*

*j*=1

*j*

†\*

*n*

*N*

*j*=1

†

*n*

*j*

*j*

### The integrals, which involve the logarithmic particular solution †\* from Eq. (8.112), are evaluated numerically for each element. Reference 7 recommends — and gives a program for — gaussian quadrature formulas.

Equations (8.113) contain 2*N* element values, †*i* and (†/*n*)*i*, of which *N* are known

### from the given boundary conditions. The remaining *N* are solved simultaneously from Eqs. (8.113). Generally this completes the analysis — only the boundary solution is computed, and interior points are not studied. In most cases, the boundary velocity and pressure are all that is needed.

We illustrated the method with stream function †. Naturally the entire technique also applies to velocity potential ф, if we are given proper conditions on ф or ф/*n* at each boundary element. The method is readily extended to three dimensions [7].

Reference 7 gives a complete FORTRAN listing for solving Eqs. (8.113) numeri- cally for constant, linear, and quadratic element strength variations. We now use their constant-element-strength program, POCONBE [7], to take an alternate look at Ex- ample 8.5, which used the finite-difference method.

**EXAMPLE 8.6**

Solve the duct expansion problem, Example 8.5, using boundary elements. Use the same grid spacing Δ*x* = Δ*y* = 0.2 m for the element sizes.

## Solution

The boundary nodes are equally spaced, as shown in Fig. 8.34. There are only 45 nodes, whereas there were 91 interior points for the FDM solution of Example 8.5. We expect the same accu- racy for 50 percent fewer nodes. (Had we reduced the grid size to 0.1 m, there would be 90

nodes as opposed to 406 interior points — a savings of 78 percent.) The program POCONBE [7] asks you to input the location of these 45 nodes. The stream-function values are known all around the boundary: † equals 0 on the bottom and 10.0 on the top and is linearly increasing from 0 to

10.0 at entrance and exit. These values of †, shown on the outside in Fig. 8.34, are inputted into

the program.

Once the input of nodes and element values is complete, the program immediately computes and displays or stores the 45 unknowns, which in this case are the values of †/*n* all around the boundary. These values are shown on the inside of the top and bottom surfaces in Fig. 8.34 and represent the local surface velocity near each element, in m/s. The values of †/*n* at en- trance and exit, which are small fractions representing vertical velocity components, are not

shown here.

10

10.8

8

6

Stream function values listed outside

10 10 10 10 10 10 10 10 10 10 10 10 10 10 10

9.73 9.68 9.46 9.12 8.68 8.16 7.62 7.10 6.63 6.23 5.88 5.59 5.28 5.54 9

Boundary velocities listed inside 8

**Fig. 8.34** Boundary elements corre-

*U* = 10.0 m/s

4

2

10.9 10.1 10.4 10.8 14.1

0 0 0 0 0 0

[In BEM there are no interior nodes.]

11.2

7.14

7

6

*U* = 5.0 5

4

3

sponding to the same grid size as Fig. 8.31. Nodal values of stream function and computed surface ve-

0 5.69

0

0

4.49

0

2.73

2

1

2.23 3.61 4.15 4.48 5.31

locity are shown.

0 0 0 0 0 0

The reader may verify that use of the surface velocities in Fig. 8.34 to compute surface pres- sure coefficients, as in Example 8.5, leads to curves very similar to those shown in Fig. 8.32. The BEM approach, using the same boundary nodes, has accuracy comparable to that for an FDM computation. For further details see Ref. 7.

# Viscous-Flow Computer Models

**One-Dimensional Unsteady Flow**

### Our previous finite-difference model of Laplace’s equation, e.g., Eq. (8.109), was very well behaved and converged nicely with or without overrelaxation. Much more care is needed to model the full Navier-Stokes equations. The challenges are quite different, and they have been met to a large extent, so there are now many textbooks [20, 23 to 27] on (fully viscous) *computational fluid dynamics* (CFD). This is not a textbook on CFD, but we will address some of the issues in this section.

We begin with a simplified problem, showing that even a single viscous term intro- duces new effects and possible instabilities. Recall (or review) Prob. 4.85, where a wall moves and drives a viscous fluid parallel to itself. Gravity is neglected. Let the wall be the plane *y* = 0, moving at a speed *U*0(*t*), as in Fig. 8.35. A uniform vertical grid, of

### spacing Δ*y*, has nodes *n* at which the local velocity *u j* is to be calculated, where su-

*n*

perscript *j* denotes the time-step *j*Δ*t*. The wall is *n* = 1. If *u* = *u*(*y, t*) only and v = *w* = 0, continuity, a· **V** = 0, is satisfied and we need only solve the *x*-momentum Navier-Stokes equation:

*u*

—*t*

2*u*

= *v* — 2

*y*

### (8.114)

where v = µ/p. Utilizing the same finite-difference approximations as in Eq. (8.106), we may model Eq. (8.114) algebraically as a forward time difference and a central spa- tial difference:

*u j*+1 — *u j*

*u j* — 2*u j* + *u j*

—*n* —*n* ≈ v —*n*+1

Δ*t*

—*n* Δ*y*2

—*n*—1

### Rearrange and find that we can solve explicitly for *un* at the next time-step *j* + 1:

*u j*+1 ≈ (1 — 2σ) *u j* + σ (*u j* + *u j* ) σ

vΔ*t*

### (8.115)

*n n n*—1

*n*+1 = —

Δ*y*

2

### Thus *u* at node *n* at the next time-step *j* + 1 is a weighted average of three previous values, similar to the “four-nearest-neighbors” average in the laplacian model of Eq. (8.109). Since the new velocity is calculated immediately, Eq. (8.115) is called an *ex-*

*plicit* model. It differs from the well-behaved laplacian model, however, because it may be *unstable*. The weighting coefficients in Eq. (8.115) must all be positive to avoid di- vergence. Now σ is positive, but (1 — 2σ) may not be. Therefore, our explicit viscous flow model has a stability requirement:

vΔ*t* 1

### σ = —2 ≤ — (8.116)

Δ*y* 2

### Normally one would first set up the mesh size Δ*y* in Fig. 8.35, after which Eq. (8.116) would limit the time-step Δ*t*. The solutions for nodal values would then be stable, but not necessarily that accurate. The mesh sizes Δ*y* and Δ*t* could be reduced to increase accuracy, similar to the case of the potential-flow laplacian model (8.109).

For example, to solve Prob. 4.85 numerically, one sets up a mesh with plenty of nodes (30 or more Δ*y* within the expected viscous layer); selects Δ*t* according to Eq.

**Fig. 8.35** An equally spaced finite- difference mesh for one-dimen- sional viscous flow [Eq. (8.114)].

*u* = *U*0



*n* + 1

Δ*y*

*n*

Δ*y*

*n* — 1

Δ*y*

Δ*y*

Wall

*n* = 1

### (8.116); and sets two boundary conditions for all *j*: *u*1 = *U*0 sin ω*t*† and *uN* = 0, where *N* is the outermost node. For initial conditions, perhaps assume the fluid initially at rest: *u*1 = 0 for 2 ≤ *n* ≤ *N* — 1. Sweeping the nodes 2 ≤ *n* ≤ *N* —1 using Eq. (8.115) (an Excel spreadsheet is excellent for this), one generates numerical values of *u j* for

*n*

*n*

as long as one desires. After an initial transient, the final “steady” fluid oscillation will approach the classical solution in viscous-flow textbooks [15]. Try Prob. 8.115 to demonstrate this.

# An Alternate Implicit Approach

### In many finite-difference problems, a stability limitation such as Eq. (8.116) requires an extremely small time-step. To allow larger steps, one can recast the model in an im- plicit fashion by evaluating the second-derivative model in Eq. (8.114) at the *next* time- step:

*u j*+1 — *u j u j*+1 — 2*u j*+1 + *u j*+1

—*n* —*n* ≈ v —*n*+1

Δ*t*

—*n*

Δ*y*2

—*n*—1

### This rearrangement is unconditionally stable for any σ, but now we have *three* un- knowns:

—σ*u j*+1 + (1 + 2σ)*uj*+1 — σ *u j*+1 ≈ *u j* (8.117)

*n*—1 *n n*+1 *n*

### This is an *implicit* model, meaning that one must solve a large system of algebraic equations for the new nodal values at time *j* + 1. Fortunately, the system is narrowly banded, with the unknowns confined to the principal diagonal and its two nearest di- agonals. In other words, the coefficient matrix of Eq. (8.117) is *tridiagonal*, a happy event. A direct method, called the *tridiagonal matrix algorithm* (TDMA), is available

and explained in most CFD texts [20, 23 to 27]. Appendix A of Ref. 20 includes a complete program for solving the TDMA. If you have not learned the TDMA yet, Eq. (8.117) converges satisfactorily by rearrangement and iteration:

*u j* + σ (*u j*+1 + *u j*+1)

*u j*+1 ≈ —*n*

*n*

—*n*—1

### 1 + 2σ

—*n*+1

### (8.118)

At each time-step *j* + 1, sweep the nodes 2 ≤ *n* ≤ *N* — 1 over and over, using Eq. (8.118), until the nodal values have converged. This implicit method is stable for any σ, however large. To ensure accuracy, though, one should keep Δ*t* and Δ*y* small com- pared to the basic time and length scales of the problem. This author’s habit is to keep Δ*t* and Δ*y* small enough that nodal values change no more than 10 percent from one (*n, j*) to the next.

**EXAMPLE 8.7**

SAE 30 oil at 20°C is at rest near a wall when the wall suddenly begins moving at a constant 1 m/s. Using the explicit model of Eq. (8.114), estimate the oil velocity at *y* = 3 cm after 1 sec- ond of wall motion.

†Finite differences are not analytical; one must set *U*0 and ω equal to numerical values.

## Solution

For SAE 30 oil, from Table A-3, v = 0.29/891 = 3.25 E-4 m2/s. For convenience in putting a node exactly at *y* = 3 cm, choose Δ*y* = 0.01 m. The stability limit (8.116) is vΔ*t*/Δ*y*2 < 0.5, or Δ*t* < 0.154 s. Again for convenience, to hit *t* = 1 s on the nose, choose Δ*t* = 0.1 s, or σ = 0.3255 and (1 — 2σ) = 0.3491. Then our explicit algebraic model (8.115) for this problem is

*u j*+1 ≈ 0.3491 *u j* + 0.3255(*u j* + *u j* ) (1)

*n n n*—1 *n*—1

We apply this relation from *n* = 2 out to at least *n* = *N* = 15, to make sure that the desired value of *u* at *n* = 3 is accurate. The wall no-slip boundary requires *u j* = 1.0 m/s = constant for all *j*. The outer boundary condition is *uN* = 0. The initial conditions are *u*1 = 0 for *n* ≥ 2. We then apply Eq. (1) repeatedly for *n* ≥ 2 until we reach *j* = 11, which corresponds to *t* = 1 s. This is easily programmed on a spreadsheet such as Excel. Here we print out only *j* = 1, 6, and 11 as follows:

1

*n*

|  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ***j*** | ***t*** | ***u*1** | ***u*2** | ***u*3** | ***u*4** | ***u*5** | ***u*6** | ***u*7** | ***u*8** | ***u*9** | ***u*10** | ***u*11** |
| 1 | 0.000 | 1.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 6 | 0.500 | 1.000 | 0.601 | 0.290 | 0.107 | 0.027 | 0.004 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 |
| 11 | 1.000 | 1.000 | 0.704 | 0.446 | **0.250** | 0.123 | 0.052 | 0.018 | 0.005 | 0.001 | 0.000 | 0.000 |

*Note:* Units for *t* and *u*’s are s and m/s, respectively.

Our numerical estimate is *u*11 = *u*(3 cm, 1 s) ≈ 0.250 m/s, which is about 4 percent high— this problem has a known exact solution, *u* = 0.241 m/s [15]. We could improve the accuracy indefinitely by decreasing Δ*y* and Δ*t*.

4

# Steady Two-Dimensional Laminar Flow

### The previous example, unsteady one-dimensional flow, had only one viscous term and no convective accelerations. Let us look briefly at incompressible two-dimensional steady flow, which has four of each type of term, plus a nontrivial continuity equation:

Continuity:

*u*

—*x*

v

### + — = 0 (8.119*a*)

*y*

*u* *u*

1 *p*

2*u* 2*u*

### *x* momentum: *u* — + v —

= —— —

+ v (— 2 + — 2 ) (8.119*b*)

*x y*

### v v

p *x*

1 *p*

*y*

*x*

*y*

*x y*

### 2v 2v

*y* momentum *u* — + v —

*x*

*y*

p

### = —— —

+ v (— 2 + — 2 ) (8.119*c*)

### These equations, to be solved for (*u,* v*, p*) as functions of (*x, y*), are familiar to us from analytical solutions in Chaps. 4 and 6. However, to a numerical analyst, they are odd,

because there is no *pressure equation*, that is, a differential equation for which the dom- inant derivatives involve *p*. This situation has led to several different “pressure-adjust- ment” schemes in the literature [20, 23 to 27], most of which manipulate the continu- ity equation to insert a pressure correction.

### A second difficulty in Eq. (8.119*b* and *c*) is the presence of nonlinear convective accelerations such as *u*(*u*/*x*), which create asymmetry in viscous flows. Early at-

tempts, which modeled such terms with a central difference, led to numerical instabil- ity. The remedy is to relate convection finite differences solely to the *upwind* flow en- tering the cell, ignoring the downwind cell. For example, the derivative *u*/*x* could be modeled, for a given cell, as (*u*upwind — *u*cell)/Δ*x*. Such improvements have made fully viscous CFD an effective tool, with various commercial user-friendly codes available.

### For details beyond our scope, see Refs. 20 and 23 to 27.

Mesh generation and gridding have also become quite refined in modern CFD. Fig- ure 8.36 illustrates a CFD solution of two-dimensional flow past an NACA 66(MOD) hydrofoil [28]. The gridding in Fig. 8.36*a* is of the C type, which wraps around the leading edge and trails off behind the foil, thus capturing the important near-wall and wake details without wasting nodes in front or to the sides. The grid size is 262 by 91. The CFD model for this hydrofoil flow is also quite sophisticated: a full Navier- Stokes solver with turbulence modeling [29] and allowance for cavitation bubble for- mation when surface pressures drop below the local vapor pressure. Figure 8.36*b* com- pares computed and experimental surface pressure coefficients for an angle of attack of

### 1°. The dimensionless pressure coefficient is defined as *Cp* = (*p*surface — *p*∞)/(p*V* 2 /2). The agreement is excellent, as indeed it is also for cases where the hydrofoil cavitates

∞

[28]. Clearly, when properly implemented for the proper flow cases, CFD can be an ex- tremely effective tool for engineers.

# Commercial CFD Codes

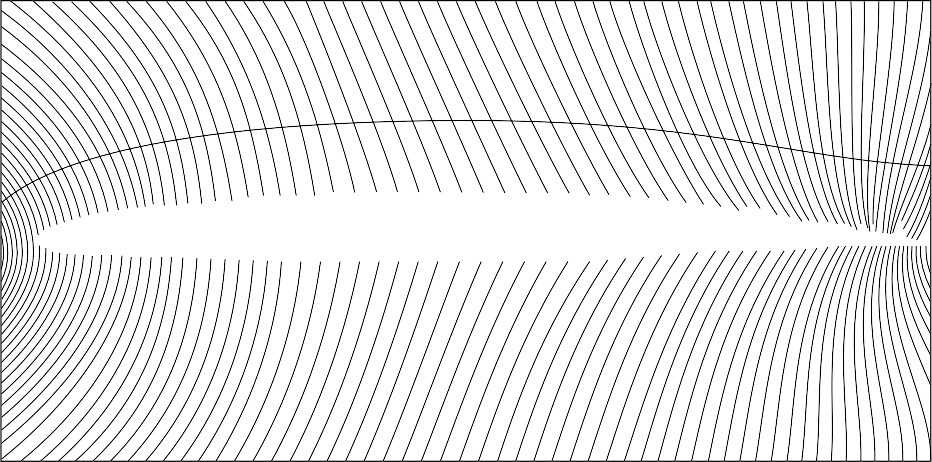
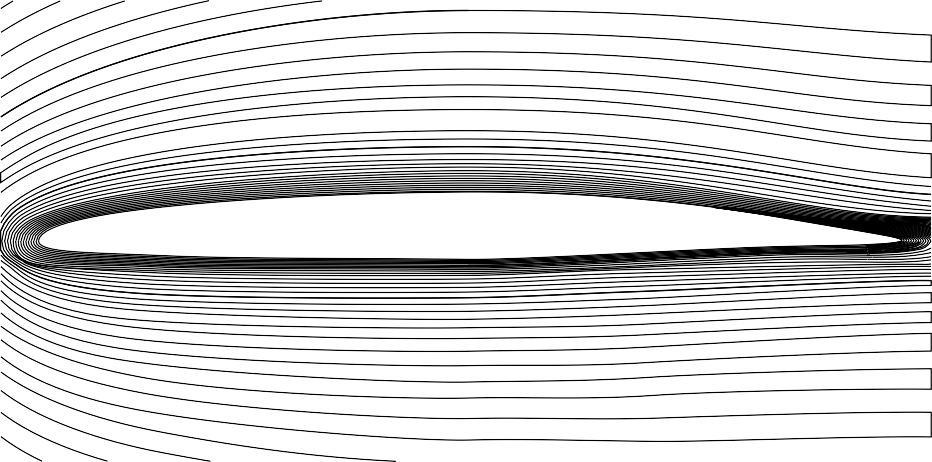
### The coming of the third millennium has seen an enormous emphasis on computer ap- plications in nearly every field, fluid mechanics being a prime example. It is now pos- sible, at least for moderately complex geometries and flow patterns, to model on a dig- ital computer, approximately, the equations of motion of fluid flow, with dedicated CFD textbooks available [20, 23 to 27]. The flow region is broken into a fine grid of elements and nodes, which algebraically simulate the basic partial differential equations of flow. While simple two-dimensional flow simulations have long been reported and can be programmed as student exercises, three-dimensional flows, involving thousands or even millions of grid points, are now solvable with the modern supercomputer.

Although elementary computer modeling was treated briefly here, the general topic of CFD is essentially for advanced study or professional practice. The big change over the past decade is that engineers, rather than laboriously programming CFD problems themselves, can now take advantage of any of several commercial CFD codes. These are extensive software packages which allow engineers to construct a geometry and boundary conditions to simulate a given viscous-flow problem. The software then grids the flow region and attempts to compute flow properties at each grid element. The con- venience is great; the danger is also great. That is, computations are not merely auto- matic, like when using a hand calculator, but rather require care and concern from the user. Convergence and accuracy are real problems for the modeler. Use of the codes

### requires some art and experience. In particular, when the flow Reynolds number, Re = p*VL*/µ, goes from moderate (laminar flow) to high (turbulent flow), the accuracy of the simulation is no longer assured in any real sense. The reason is that turbulent flows

are not completely resolved by the full equations of motion, and one resorts to using approximate turbulence models.

### Turbulence models [29] are developed for particular geometries and flow conditions and may be inaccurate or unrealistic for others. This is discussed by Freitas [30], who



(*a*)

0.6

●

●

●

●

●

●

●

●

●

●

●

● ●

* Expt.

Comp.

0.5

0.4

0.3

*Cp* 0.2

0.1

**Fig. 8.36** CFD results for water flow past an NASA 66(MOD) hy- drofoil [*from Ref 28, with permis- sion of the American Society of Mechanical Engineers*]: (*a*) C grid-

ding, 262 by 91 nodes; (*b*) surface pressures at α = 1°.

0.0

—0.1

—0.2

0.0 0.5 1.0

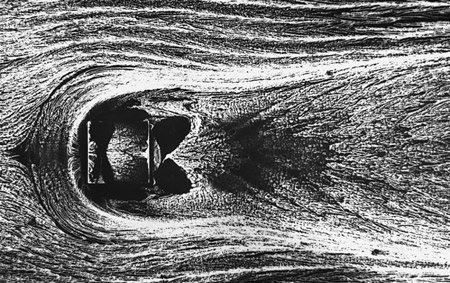
*x*/*C*

(*b*)

### compared eight different commercial-code calculations (FLOW-3D, FLOTRAN, STAR- CD, N3S, CFD-ACE, FLUENT, CFDS-FLOW3D, and NISA/3D-FLUID) with exper-

imental results for five benchmark flow experiments. Calculations were made by the vendors themselves. Freitas concludes that commercial codes, though promising in gen- eral, can be inaccurate for certain laminar- and turbulent-flow situations. Further re- search is recommended before engineers can truly rely upon such software to give gen- erally accurate fluid-flow predictions.

### In spite of the above warning to treat CFD codes with care, one should also realize that the results of a given CFD simulation can be spectacular. Figure 8.37 illustrates turbulent flow past a cube mounted on the floor of a channel whose clearance is twice



(*a*)

**Fig. 8.37** Flow over a surface- mounted cube creates a complex and perhaps unexpected pattern: (*a*) experimental oil-streak visualiza-

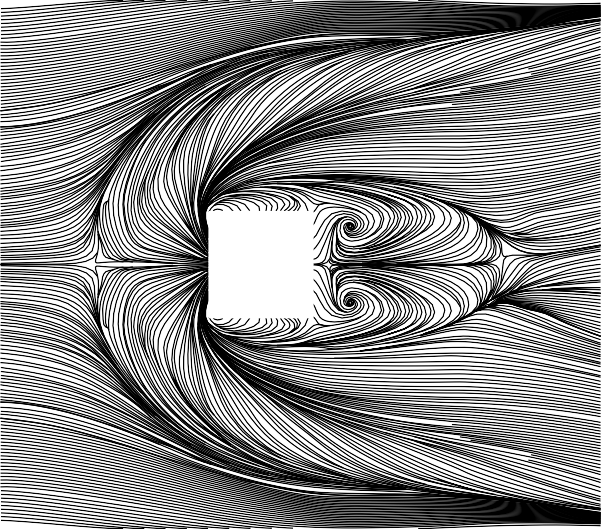
tion of surface flow at Re = 40,000 (based on cube height) ( *from Ref.*

*31, courtesy of Robert Martinuzzi, with the permission of the Ameri- can Society of Mechanical Engi- neers*); (*b*) computational large- eddy simulation of the surface flow in (*a*) ( *from Ref. 32, courtesy of Kishan Shah, Stanford University*); and (*c*) a side view of the flow in

(*a*) visualized by smoke generation and a laser light sheet ( *from Ref. 31, courtesy of Robert Martinuzzi, with the permission of the Ameri- can Society of Mechanical Engi- neers*).

(*c*)



(*b*)

the cube height. Compare Fig. 8.37*a*, a top view of the experimental surface flow [31] as visualized by oil streaks, with Fig. 8.37*b*, a CFD supercomputer result using the method of large-eddy simulation [32, 33]. The agreement is remarkable. The C-shaped flow pattern in front of the cube is caused by formation of a horseshoe vortex, as seen in a side view of the experiment [31] in Fig. 8.37*c*. Horseshoe vortices commonly re- sult when surface shear flows meet an obstacle. We conclude that CFD has a tremen- dous potential for flow prediction.

**Summary**

**Problems**

### This chapter has analyzed a highly idealized but very useful type of flow: inviscid, in- compressible, irrotational flow, for which Laplace’s equation holds for the velocity po- tential (8.1) and for the plane stream function (8.7). The mathematics is well devel- oped, and solutions of potential flows can be obtained for practically any body shape. Some solution techniques outlined here are (1) superposition of elementary line or point solutions in both plane and axisymmetric flow, (2) the analytic functions of a complex variable, (3) use of variable-strength vortex sheets, and (4) numerical analy- sis on a digital computer. Potential theory is especially useful and accurate for thin bodies such as airfoils. The only requirement is that the boundary layer be thin, i.e.,

that the Reynolds number be large.

### For blunt bodies or highly divergent flows, potential theory serves as a first approxi- mation, to be used as input to a boundary-layer analysis. The reader should consult the advanced texts [for example, 2 to 4, 10 to 13] for further applications of potential the- ory. Section 8.9 discusses computational methods for viscous (nonpotential) flows.

Most of the problems herein are fairly straightforward. More diffi- cult or open-ended assignments are labeled with an asterisk. Prob- lems labeled with an EES icon will benefit from the use of the En- gineering Equation Solver (EES), while problems labeled with a computer disk may require the use of a computer. The standard end- of-chapter problems 8.1 to 8.115 (categorized in the problem list below) are followed by word problems W8.1 to W8.7, comprehen- sive problems C8.1 to C8.3, and design projects D8.1 to D8.3.

**Problem Distribution**

|  |  |  |
| --- | --- | --- |
| **Section** | **Topic** | **Problems** |
| 8.1 | Introduction and review | 8.1–8.7 |
| 8.2 | Elementary plane-flow solutions | 8.8–8.17 |
| 8.3 | Superposition of plane flows | 8.18–8.34 |
| 8.4 | Plane flow past closed-body shapes | 8.35–8.59 |
| 8.5 | The complex potential | 8.60–8.71 |
| 8.6 | Images | 8.72–8.79 |
| 8.7 | Airfoil theory: Two-dimensional | 8.80–8.84 |
| 8.7 | Airfoil theory: Finite-span wings | 8.85–8.90 |
| 8.8 | Axisymmetric potential flow | 8.91–8.103 |
| 8.8 | Hydrodynamic mass | 8.104–8.105 |
| 8.9 | Numerical methods | 8.106–8.115 |

**P8.1** Prove that the streamlines †(*r*, θ) in polar coordinates from Eqs. (8.10) are orthogonal to the potential lines ф(*r*, θ).

**P8.2** The steady plane flow in Fig. P8.2 has the polar velocity components vθ = Ω*r* and v*r* = 0. Determine the circula- tion Г around the path shown.

R2



R1

#### P8.2

**P8.3** Using cartesian coordinates, show that each velocity com- ponent (*u*, v, *w*) of a potential flow satisfies Laplace’s equa- tion separately.

**P8.4** Is the function 1/*r* a legitimate velocity potential in plane polar coordinates? If so, what is the associated stream func- tion †(*r*, θ)?

**P8.5** Consider the two-dimensional velocity distribution *u* =

—*By*, v = *Bx*, where *B* is a constant. If this flow possesses a

stream function, find its form. If it has a velocity potential, find that also. Compute the local angular velocity of the flow, if any, and describe what the flow might represent.

**P8.6** An incompressible flow has the velocity potential ф = 2*Bxy*, where *B* is a constant. Find the stream function of this flow,

sketch a few streamlines, and interpret the pattern.

**P8.7** Consider a flow with constant density and viscosity. If the flow possesses a velocity potential as defined by Eq. (8.1),

**P8.12** Consider the flow due to a vortex of strength *K* at the ori- gin. Evaluate the circulation from Eq. (8.15) about the clockwise path from (*r*, θ) = (*a*, 0) to (2*a*, 0) to (2*a*, 3π/2)

to (*a*, 3π/2) and back to (*a*, 0). Interpret the result.

**P8.13** A well-known exact solution to the Navier-Stokes equa-

tions (4.38) is the unsteady circulating motion [15]

*K r*2

vθ = — [1 — exp(——)] v*r* = v*z* = 0

show that it exactly satisfies the full Navier-Stokes equa-

2π*r*

4v*t*

tions (4.38). If this is so, why for inviscid theory do we back away from the full Navier-Stokes equations?

**P8.8** For the velocity distribution of Prob. 8.5, evaluate the cir- culation Г around the rectangular closed curve defined by (*x*, *y*) = (1, 1), (3, 1), (3, 2), and (1, 2). Interpret your re- sult, especially vis-à-vis the velocity potential.

**P8.9** Consider the two-dimensional flow *u* = —*Ax*, v = *Ay*, where *A* is a constant. Evaluate the circulation Г around the rectangular closed curve defined by (*x*, *y*) = (1, 1),

where *K* is a constant and v is the kinematic viscosity. Does this flow have a polar-coordinate stream function and/or ve- locity potential? Explain. Evaluate the circulation Г for this motion, plot it versus *r* for a given finite time, and interpret

compared to ordinary line vortex motion.

**P8.14** A tornado may be modeled as the circulating flow shown in Fig. P8.14, with v*r* = v*z* = 0 and vθ(*r*) such that

 ω*r r* ≤ *R*

(4, 1), (4, 3), and (1, 3). Interpret your result, especially vis-à-vis the velocity potential.

vθ =  ω*R*2

*v* —

*r* > *R*

**P8.10** A mathematical relation sometimes used in fluid mechan-  *r*

ics is the theorem of Stokes [1]

ф **V** · *d***s** = ∫ ∫ (

∇×

*C A*

**V**) **n** *dA*

·

Determine whether this flow pattern is irrotational in ei- ther the inner or outer region. Using the *r*-momentum

equation (D.5) of App. D, determine the pressure distri-

bution *p*(*r*) in the tornado, assuming *p* = *p*∞ as *r*  ∞*.*

where *A* is any surface and *C* is the curve enclosing that sur- face. The vector *d***s** is the differential arc length along *C*, and **n** is the unit outward normal vector to *A*. How does this relation simplify for irrotational flow, and how does the re- sulting line integral relate to velocity potential?

**P8.11** A power plant discharges cooling water through the man- ifold in Fig. P8.11, which is 55 cm in diameter and 8 m high and is perforated with 25,000 holes 1 cm in diame- ter. Does this manifold simulate a line source? If so, what is the equivalent source strength *m*?

Find the location and magnitude of the lowest pressure.

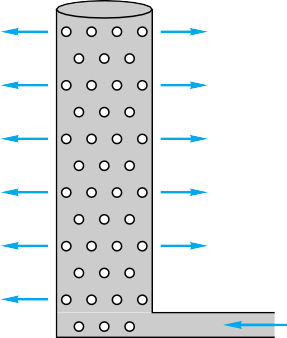
*r*



v (*r*)

*R*

#### P8.14



Inlet

**P8.11**

**P8.15** Evaluate Prob. 8.14 for the particular case of a small-scale tornado, *R* = 100 m, vθ*,*max = 65 m/s, with sea-level con- ditions at *r* = ∞. Plot *p*(*r*) out to *r* = 400 m.

**P8.16** Consider inviscid stagnation flow, † = *Kxy* (see Fig.

8.15*b*) superimposed with a source at the origin of strength

*m*. Plot the resulting streamlines in the upper half plane, using the length scale *a* = (*m/K*)1/2. Give a physical inter- pretation of the flow pattern.

**P8.17** Examine the flow of Fig. 8.30 as an *analytical* (not a nu- merical) problem. Give the appropriate differential equa- tion and the complete boundary conditions for both the stream function and the velocity potential. Is a Fourier- series solution possible?

**P8.18** Plot the streamlines and potential lines of the flow due to a line source of strength *m* at (*a*, 0) plus a source 3*m* at (—*a*, 0). What is the flow pattern viewed from afar?

**P8.19** Plot the streamlines and potential lines of the flow due to a line source of strength 3*m* at (*a*, 0) plus a sink —*m* at (—*a*, 0). What is the pattern viewed from afar?

**P8.20** Plot the streamlines of the flow due to a line vortex +*K* at

(0, + *a*) and a vortex —*K* at (0,—*a*). What is the pattern viewed from afar?

**P8.21** Plot the streamlines of the flow due to a line vortex +*K* at (+*a*, 0) and a vortex —2*K* at (—*a*, 0). What is the pattern viewed from afar?

**P8.22** Plot the streamlines of a uniform stream **V** = ***i****U* plus a clockwise line vortex —*K* located at the origin. Are there any stagnation points?

**P8.23** Find the resultant velocity vector induced at point *A* in Fig. P8.23 by the uniform stream, vortex, and line source.





40 m

#### P8.25

**P8.26** Find the resultant velocity vector induced at point *A* in Fig. P8.26 by the uniform stream, line source, line sink, and vortex.



EES

*m* = 12 m2 /s

1 m

2 m

2 m

20

*U* = 6 m /s

*A*

1 m

*m* = –10 m2 /s

*K* = 9 m2 /s

*U* = 8 m /s



*K* = 25 m2 /s

1.5 m

2 m

1 m

*m* = 15 m2 /s

*A*

#### P8.23

**P8.24** Line sources of equal strength *m* = *Ua*, where *U* is a ref- erence velocity, are placed at (*x*, *y*) = (0, *a*) and (0,—*a*). Sketch the stream and potential lines in the upper half plane. Is *y* = 0 a “wall’’? If so, sketch the pressure coef- ficient

*p* — *p*

#### P8.26

**P8.27** A counterclockwise line vortex of strength 3*K* at (*x*, *y*) = (0, *a*) is combined with a clockwise vortex *K* at (0, —*a*). Plot the streamline and potential-line pattern, and find the point of minimum velocity between the two

vortices.

**P8.28** Sources of equal strength *m* are placed at the four sym- metric positions (*x*, *y*) = (*a*, *a*), (—*a*, *a*), (—*a*, —*a*), and (*a*, —*a*). Sketch the streamline and potential-line patterns. Do any plane “walls’’ appear?

**P8.29** A uniform water stream, *U*∞ = 20 m/s and p = 998 kg/m3,

*C* = —0

—— p*U*

2

*p* 1 2

along the wall, where *p*0 is the pressure at (0, 0). Find the minimum pressure point and indicate where flow separa- tion might occur in the boundary layer.

**P8.25** Let the vortex/sink flow of Eq. (4.134) simulate a tor- nado as in Fig. P8.25. Suppose that the circulation about the tornado is Г= 8500 m2/s and that the pres-

sure at *r* = 40 m is 2200 Pa less than the far-field pres-

sure. Assuming inviscid flow at sea-level density, es- timate (*a*) the appropriate sink strength —*m*, (*b*) the pressure at *r* = 15 m, and (*c*) the angle β at which the streamlines cross the circle at *r* = 40 m (see Fig. P8.25).

combines with a source at the origin to form a half-body. At (*x*, *y*) = (0, 1.2 m), the pressure is 12.5 kPa less than *p*∞. (*a*) Is this point outside the body? Estimate (*b*) the ap-

propriate source strength *m* and (*c*) the pressure at the nose of the body.

**P8.30** Suppose that the total discharge from the manifold in Fig. P8.11 is 450 m3/s and that there is a uniform ocean cur- rent of 60 cm/s to the right. Sketch the flow pattern from above, showing the dimensions and the region where the cooling-water discharge is confined.

**P8.31** A Rankine half-body is formed as shown in Fig. P8.31. For the stream velocity and body dimension shown, com- pute (*a*) the source strength *m* in m2/s, (*b*) the distance *a*, (*c*) the distance *h*, and (*d*) the total velocity at point *A*.

#### P8.31

*A* **P8.40** Consider a uniform stream *U*∞ plus line sources +*m* at (*x*, *y*) = (+a, 0) and (—a, 0) and a single line sink —2*m* at the origin. Does a closed-body shape appear? If so, plot

its shape for *m*/(*U*∞*a*) equal to (*a*) 1.0 and (*b*) 5.0.

(0, 3 m)

*y*

*h*

*a*

7 m /s

*x*

+*m* Source

(4m, 0)

**P8.41** A Kelvin oval is formed by a line-vortex pair with *K* = 9 m2/s, *a* = 1 m, and *U* = 10 m/s. What are the height, width, and shoulder velocity of this oval?

**P8.42** For what value of *K*/(*U*∞*a*) does the velocity at the shoul- der of a Kelvin oval equal 4*U*∞? What is the height *h*/*a* of this oval?



EES

**P8.43** Consider water at 20°C flowing at 6 m/s past a 1-m-di-

**P8.32** Sketch the streamlines, especially the body shape, due to equal line sources + *m* at (—*a*, 0) and (+*a*, 0) plus a uni- form stream *U*∞ = *ma.*

**P8.33** Sketch the streamlines, especially the body shape, due to equal line sources + *m* at (0, + *a*) and (0, —*a*) plus a uni- form stream *U*∞ = *ma.*

**P8.34** Consider three equally spaced sources of strength *m* placed at (*x*, *y*) = (0, +*a*), (0, 0), and (0, —*a*). Sketch the result- ing streamlines, noting the position of any stagnation points. What would the pattern look like from afar?

**P8.35** Consider three equal sources *m* in a triangular configura- tion: one at (*a*/2, 0), one at (—*a*/2, 0), and one at (0, *a*). Plot the streamlines for this flow. Are there any stagnation points? *Hint*: Try the MATLAB contour command [34].

**P8.36** When a line source-sink pair with *m* = 2 m2/s is combined with a uniform stream, it forms a Rankine oval whose min- imum dimension is 40 cm. If *a* = 15 cm, what are the stream velocity and the velocity at the shoulder? What is

the maximum dimension?

**P8.37** A Rankine oval 2 m long and 1 m high is immersed in a stream *U*∞ = 10 m/s, as in Fig. P8.37. Estimate (*a*) the ve- locity at point A and (*b*) the location of point B where a

particle approaching the stagnation point achieves its max- imum deceleration.

*A*

*B* ?

1 m

10 m/s

2 m

ameter circular cylinder. What doublet strength h in m3/s is required to simulate this flow? If the stream pressure is 200 kPa, use inviscid theory to estimate the surface pres- sure at θ equal to (*a*) 180°, (*b*) 135°, and (*c*) 90°.

**P8.44** Suppose that circulation is added to the cylinder flow of

Prob. 8.43 sufficient to place the stagnation points at θ

equal to 50° and 130°. What is the required vortex strength

*K* in m2/s? Compute the resulting pressure and surface ve- locity at (*a*) the stagnation points and (*b*) the upper and lower shoulders. What will the lift per meter of cylinder width be?

**P8.45** What circulation *K* must be added to the cylinder flow in Prob. 8.43 to place the stagnation point exactly at the up- per shoulder? What will the velocity and pressure at the lower shoulder be then? What value of *K* causes the lower shoulder pressure to be 10 kPa?

**P8.46** A cylinder is formed by bolting two semicylindrical chan- nels together on the inside, as shown in Fig. P8.46. There are 10 bolts per meter of width on each side, and the in- side pressure is 50 kPa (gage). Using potential theory for the outside pressure, compute the tension force in each bolt if the fluid outside is sea-level air.

*D* = 2 m



*p* = 50 k Pa (gage)

*U* = 25 m /s

#### P8.37

**P8.38** A uniform stream *U* in the *x* direction combines with a source *m* at (*a*, 0) and a sink —*m* at (—*a*, 0). Plot the re- sulting streamlines and note any stagnation points.

**P8.39** Sketch the streamlines of a uniform stream *U*∞ past a line source-sink pair aligned vertically with the source at +*a* and the sink at —*a* on the *y*-axis. Does a closed-body shape form?

#### P8.46

**P8.47** A circular cylinder is fitted with two surface-mounted pres- sure sensors, to measure *pa* at θ = 180° and *pb* at θ = 105°. The intention is to use the cylinder as a stream velocime- ter. Using inviscid theory, derive a formula for estimating

*U*∞ in terms of *pa*, *pb*, p, and the cylinder radius a.

**\*P8.48** Wind at *U*∞ and *p*∞ flows past a Quonset hut which is a

half-cylinder of radius *a* and length *L* (Fig. P8.48). The in- ternal pressure is *pi.* Using inviscid theory, derive an ex-

pression for the upward force on the hut due to the dif- ference between *pi* and *p*s*.*

ft2. As sketched in Fig. P8.54, it had two rotors 50 ft high and 9 ft in diameter rotating at 750 r/min, which is far out- side the range of Fig. 8.11. The measured lift and drag co- efficients for each rotor were about 10 and 4, respectively. If the ship is moored and subjected to a crosswind of 25 ft/s, as in Fig. P8.54, what will the wind force parallel and normal to the ship centerline be? Estimate the power re- quired to drive the rotors.

#### P8.48



*ps* ( )

*U*, *p*

*A*

*pi*

*a*





**P8.49** In strong winds the force in Prob. 8.48 can be quite large. Suppose that a hole is introduced in the hut roof at point *A* to make *pi* equal to the surface pressure there. At what angle θ should hole *A* be placed to make the net wind force

zero?

**P8.50** It is desired to simulate flow past a two-dimensional ridge or bump by using a streamline which passes above the flow over a cylinder, as in Fig. P8.50. The bump is to be *a*/2 high, where *a* is the cylinder radius. What is the elevation *h* of this streamline? What is *U*max on the bump compared with stream velocity *U*?





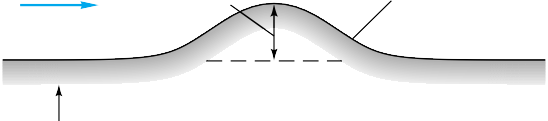


*U*

#### P8.54

|  |  |  |  |
| --- | --- | --- | --- |
|  |  |  |  |

*U a*/2 *U*max? Bump



*h?*

*a*

*U*

#### P8.50

**P8.51** Modify Prob. 8.50 as follows. Let the bump be such that *U*max = 1.5*U*. Find (*a*) the upstream elevation *h* and (*b*) the height of the bump.

**P8.52** The Flettner rotor sailboat in Fig. E8.2 has a water drag coefficient of 0.006 based on a wetted area of 45 ft2. If the rotor spins at 220 r/min, find the maximum boat velocity that can be achieved in a 15-mi/h wind. What is the optimum angle between the boat and the wind?

**P8.53** Modify Prob. 8.52 as follows. For the same sailboat data, find the wind velocity, in mi/h, which will drive the boat at an optimum speed of 10 kn parallel to its keel.

**P8.54** The original Flettner rotor ship was approximately 100 ft long, displaced 800 tons, and had a wetted area of 3500

**P8.55** Assume that the Flettner rotorship of Fig. P8.54 has a wa- ter resistance coefficient of 0.005. How fast will the ship sail in seawater at 20°C in a 20-ft/s wind if the keel aligns itself with the resultant force on the rotors? *Hint:* This is a problem in relative velocities.

**P8.56** The measured drag coefficient of a cylinder in crossflow, based on frontal area *DL*, is approximately 1.0 for the lam- inar-boundary-layer range (see Fig. 7.16*a*). Boundary- layer separation occurs near the shoulder (see Fig. 7.13*a*). This suggests an analytical model: the standard inviscid- flow solution on the front of the cylinder and constant pres- sure (equal to the shoulder value) on the rear. Use this model to predict the drag coefficient and comment on the results with reference to Fig. 7.13*c*.

**P8.57** In principle, it is possible to use rotating cylinders as air- craft wings. Consider a cylinder 30 cm in diameter, rotat- ing at 2400 r/min. It is to lift a 55-kN airplane cruising at 100 m/s. What should the cylinder length be? How much power is required to maintain this speed? Neglect end ef- fects on the rotating wing.

**P8.58** Plot the streamlines due to the combined flow of a line sink —*m* at the origin plus line sources +*m* at (*a*, 0) and (4*a*, 0). *Hint:* A cylinder of radius 2*a* will appear.

**P8.59** By analogy with Prob. 8.58 plot the streamlines due to counterclockwise line vortices + *K* at (0, 0) and (4*a*, 0) plus a clockwise vortex —*K* at (*a*, 0). Again a cylinder ap- pears.