A Computer-based Textbook for Introductory Fluid Mechanics

David A. Caughey and James A. Liggett
Cornell University

Abstract

An interactive textbook that uses the power of the personal computer to teach introductory fluid mechanics has been developed by the authors. This mode of presentation integrates hypertext navigational and search features, the presentation of videos and animations to illustrate phenomena and concepts, and computation to allow the presentation of results for a variety of parameter values and the solution of nonlinear problems without the tedium of table look-up or iteration on the part of the student. The authors' experience using an early version of the book to teach junior-level students in mechanical engineering and in civil and environmental engineering indicates that the students appreciate the increased understanding that comes with dynamic figures, the easy access to data, the ability to locate quickly definitions and specific material, and, most of all, the computational facilities.

1. Introduction

Fluid mechanics is an engineering science of fundamental importance to most branches of engineering, including aerospace, chemical, civil, environmental, and mechanical engineering, as well as to some aspects of electrical engineering and materials engineering. Fluid mechanics typically is taught to engineering students in curricula in the above fields, starting with one or more courses in the junior year. In spite of the fact that we spend our lives surrounded by, and immersed in, fluids, this course is considered difficult by most students -- largely because of the abstract nature of the formulations of many problems in fluid mechanics, for which the typical student has not developed an intuitive feel, and the frequency with which nonlinearity is a factor in the formulation of even the most common engineering problems.

The authors have developed a textbook[3] that uses the power of the personal computer to try to address issues of visualization of phenomena, the connection between fluids phenomena and their mathematical description, and the inherent difficulty of meaningful computation. The book is designed as a stand-alone text, not as a supplement to an existing text. Although a paper version will be available to accompany the electronic form, the book is designed to be read on a computer, where it integrates hypertext features, animations and video sequences that illustrate kinematic and dynamic phenomena, graphics to present data dynamically, and computational facilities for the solution of complex problems without the tedium associated with classical (graphical or tabular iterative) methods.

The navigational features include an extensive glossary of key words for which definitions and related key words can be found in context, a history list of previously visited pages, and several types of tables of contents that allow the student to jump quickly to any particular portion of the text. Animations illustrating key concepts or complex derivations are included as "QuickTime" movies, as are video clips to illustrate various fluid flow phenomena.

All numerical computation is done within MATLAB[*] but data are input and results are presented through Graphical User Interfaces, so no programming is required of the student. This underlying computational power allows the incorporation of dynamic figures in which the student can change the value of one or more parameters and see immediately the effect in plotted results, and an "Active Equations" utility that allows the student to plot any meaningful dependent variable as a function of any meaningful independent variable for many equations. More than a dozen computational utilities are integrated with the text, including a units conversion program, a general utility for solving systems of nonlinear or transcendental equations, a one-dimensional energy equation solver for pipe-flow problems with friction, both superposition and boundary-integral equation methods for two-dimensional potential flow problems, spreadsheet implementations of the compressible flow functions that normally are presented in tabular form, and a number of others. These computational utilities enable the student to do enough exercises that he/she can develop some intuition for the behavior of fluid systems.

The present paper provides an overview of the features of this new form of textbook and some of the experiences of the authors using a preliminary version of the text to teach junior-level courses in mechanical and civil engineering. The navigational features are described in the next section, followed by a description of the animation and video features. The computational features are described using several examples. Finally, observations on how the new textbook is likely to change the nature of the teacher/student interaction and experiences using preliminary versions of the text teaching junior-level courses to students in civil and environmental engineering and mechanical engineering are summarized.

2. Navigation and Integration

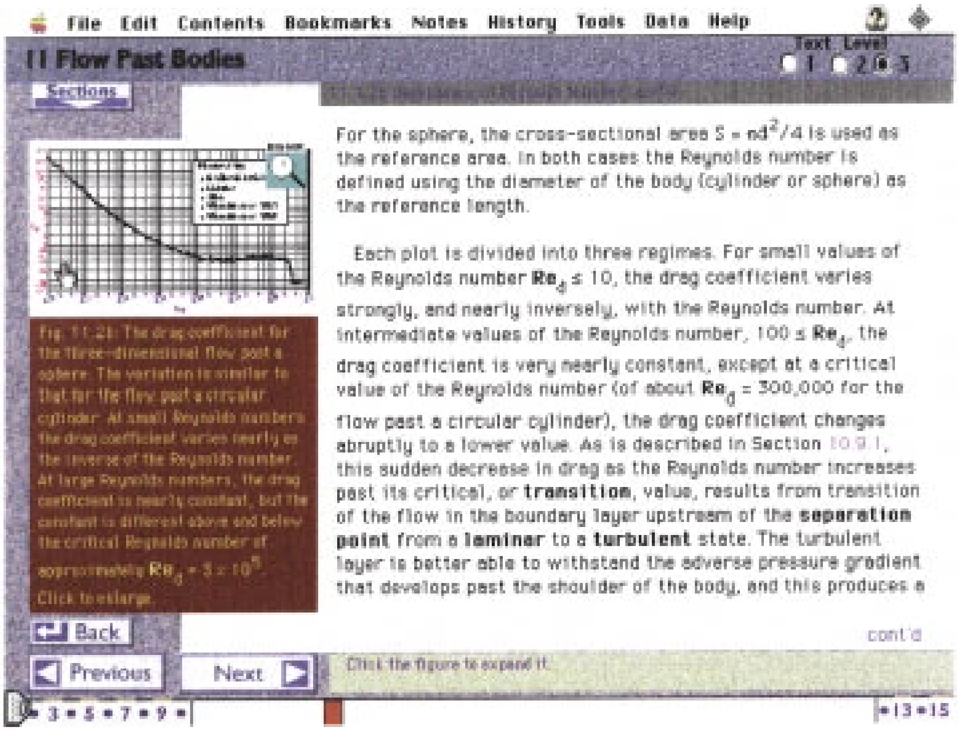
The visual presentation of the textbook on the computer screen is designed to mimic the appearance of a conventional (paper) textbook. The screens are page-oriented (rather than scrolled), so that visual memory of where objects appear on the page is preserved, and a number of visual cues are provided to remind the student of the current location in the book. The appearance of a typical page is illustrated in Fig. 1.

The basic navigational controls are located in the lower left corner of the page; the "Next" and "Previous" buttons take the student to the next, or previous, linear page in the text at the current

---

[*] MATLAB is a versatile numerical analysis and graphics software package developed and distributed by MathWorks, Inc. of Natick, Massachusetts. An executable version of MATLAB will be distributed with the book, but will be available only to run the utilities that the authors have developed.

level.[**] The "Back" button steps the student back through the pages most recently visited -- i.e., through a "History" list -- regardless of their location relative to the current page. The history list of pages visited most recently is accessible from the "History" menu, and the student can jump to any selected page on the history list. The location of the current page in the chapter is indicated by the location of the red rectangle in the slider bar near the bottom of the page; the page shown in Fig. 1 is approximately 20% of the way through Chapter 11. The numbers to the left and right of the slider bar indicate other chapters in the book (with only odd numbers displayed because of space limitations; even-numbered chapters are denoted by the bullet markers); clicking on a chapter number (or marker) takes the student to the first page of the corresponding chapter.



For the sphere, the cross-sectional area $S = \pi d^2/4$ is used as the reference area. In both cases the Reynolds number is defined using the diameter of the body (cylinder or sphere) as the reference length.

Each plot is divided into three regimes. For small values of the Reynolds number $\mathbf{Re}_d \le 10$, the drag coefficient varies strongly, and nearly inversely, with the Reynolds number. At intermediate values of the Reynolds number, $100 \le \mathbf{Re}_d$, the drag coefficient is very nearly constant, except at a critical value of the Reynolds number (of about $\mathbf{Re}_d = 300{,}000$ for the flow past a circular cylinder), the drag coefficient changes abruptly to a lower value. As is described in Section 10.9.1, this sudden decrease in drag as the Reynolds number increases past its critical, or **transition**, value, results from transition of the flow in the boundary layer upstream of the **separation point** from a **laminar** to a **turbulent** state. The turbulent layer is better able to withstand the adverse pressure gradient that develops past the shoulder of the body, and this produces a

cont'd

Fig. 11.21: The drag coefficient for the three-dimensional flow past a sphere. The variation is similar to that for the flow past a circular cylinder. At small Reynolds numbers the drag coefficient varies nearly as the inverse of the Reynolds number. At large Reynolds numbers, the drag coefficient is nearly constant, but the constant is different above and below the critical Reynolds number of approximately $\mathbf{Re}_d = 3 \times 10^5$. Click to enlarge.

Click the figure to expand it.

Fig. 1 Typical page (or screen) of the interactive text book.

A detailed table of contents for the current chapter can be viewed at any time by clicking on "Sections" in the upper left corner of the page, and the table of contents for any chapter can be viewed from the book table of contents which lists all the chapters. The student also can search the entire book (or any selected subset of chapters) for any figures (including animations, videos, etc.) using the "Figure Search" item under the Table of Contents menu which also has the ability to find only those figures containing a particular text string in their captions if desired.

The textbook provides most of the functionality associated with hypertext presentations. These include the ability to call up short definitions of most important technical terms directly from

---

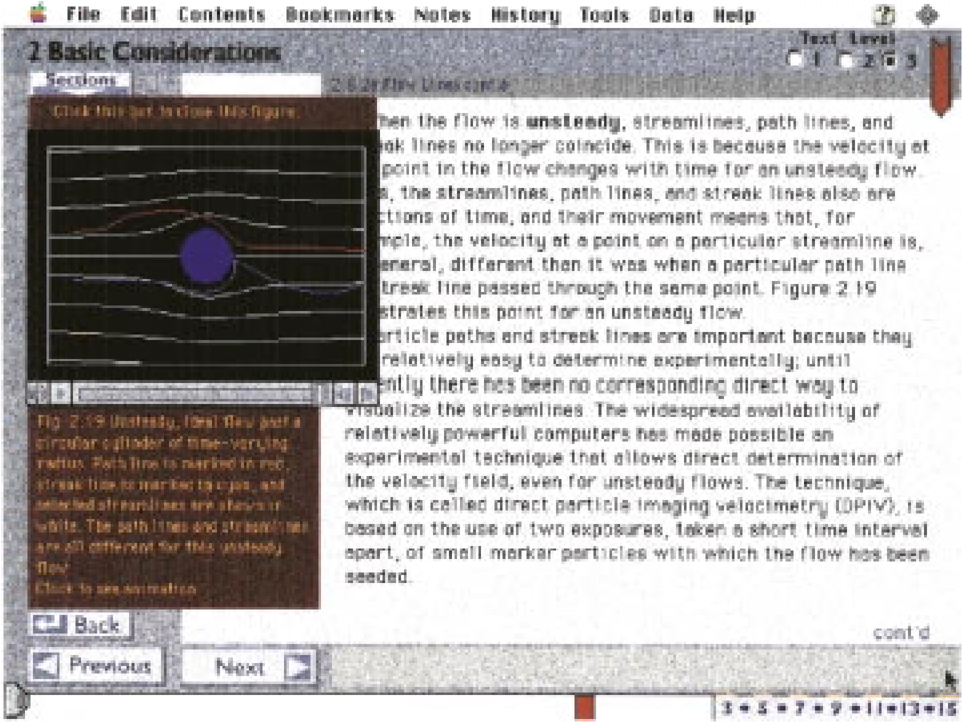
[**] The sections in the text are organized into three levels: Level 1 contains overview material, Level 2 contains all Level 1 material plus that required to understand how to do engineering applications, and Level 3 contains all Level 2 material plus detailed background on complex derivations or material of a more specialized nature. The current level is indicated by the radio buttons in the upper right corner of the page. Although Levels 1 and 2 "hide" some material from the reader, that material is easily available by changing levels or by using the table of contents.

their appearance in the text, to call up definitions of related terms from within the Glossary, and to jump to pages in the text where the term or related terms are first introduced. Glossary terms that are treated as "hot text" in this way are denoted using bold face type: the terms **transition**, **separation point**, **laminar**, and **turbulent** appearing on the page in Fig. 1 are examples. Equations and figures from earlier pages that are referred to on the current page also can be viewed in a similar way.

The student also can write "Notes" on any page, or place a bookmark on any page so that it can be found (and returned to) easily using the "Bookmark" menu.

3. Illustrations and Data

Three types of illustrations are provided in the text: (1) static figures, (2) dynamic animations/videos, and (3) active graphs. Static figures are similar to figures in a conventional paper text. Simple figures are displayed full size in the left margin of the page. More complex figures, such as the one appearing on the page in Fig. 1, are represented by a thumbnail sketch in the margin carrying a "magnifying glass" icon. Clicking on the thumbnail sketch of such a figure causes it to be displayed full size on the screen for detailed study.



hen the flow is **unsteady**, streamlines, path lines, and
eak lines no longer coincide. This is because the velocity at
point in the flow changes with time for an unsteady flow.
s, the streamlines, path lines, and streak lines also are
ctions of time, and their movement means that, for
mple, the velocity at a point on a particular streamline is,
eneral, different than it was when a particular path line
treak line passed through the same point. Figure 2.19
strates this point for an unsteady flow.
rticle paths and streak lines are important because they
relatively easy to determine experimentally; until
ntly there has been no corresponding direct way to
visualize the streamlines. The widespread availability of
relatively powerful computers has made possible an
experimental technique that allows direct determination of
the velocity field, even for unsteady flows. The technique,
which is called direct particle imaging velocimetry (DPIV), is
based on the use of two exposures, taken a short time interval
apart, of small marker particles with which the flow has been
seeded.

cont'd

Fig. 2.19 Unsteady, ideal flow past a circular cylinder of time-varying radius. Path line is marked in red, streak line is marked in cyan, and selected streamlines are shown in white. The path lines and streamlines are all different for this unsteady flow.

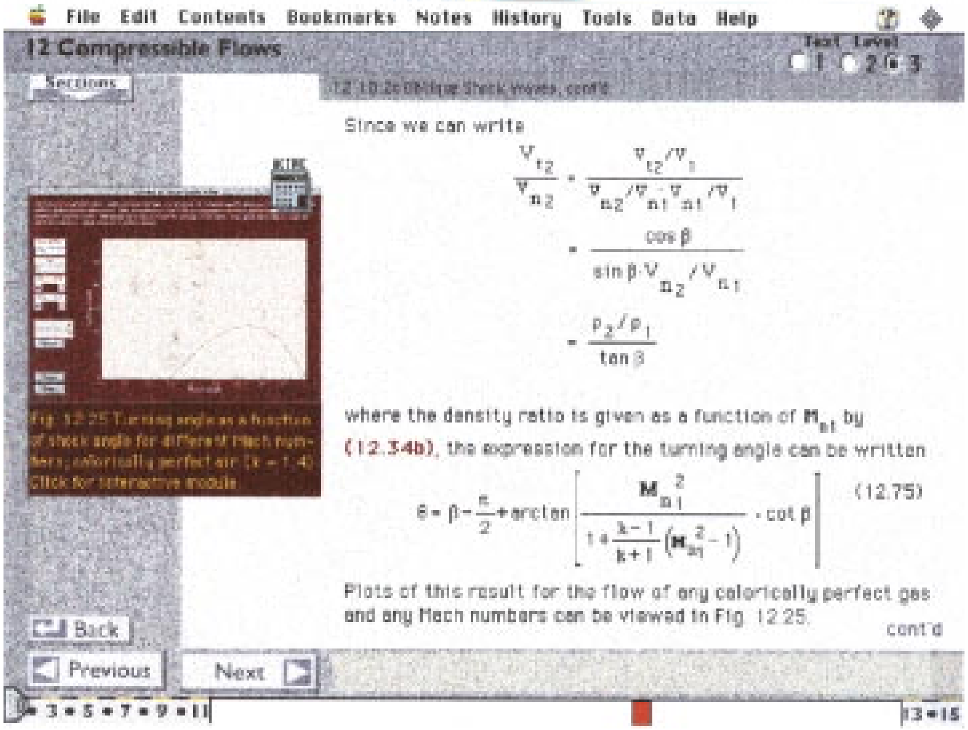
Click to see animation.

Fig. 2   Single frame of a video animation illustrating the distinction between path lines, streak lines, and streamlines for the ideal, incompressible flow past a circular cylinder whose radius varies with time.

Dynamic animations or videos are used to illustrate dynamic features, to provide video displays of actual flow phenomena, and to provide several types of introductory information. This introductory information includes overviews of some of the chapters (appearing on the first page of the chapter) and tutorial sessions illustrating many of the computational utilities included as part of the text. These animations and videos are represented by thumbnail sketches in the left margin of the page and are identified by a "filmstrip" icon. An animation or video is launched as a "QuickTime" movie by clicking on its identifying thumbnail sketch. The "QuickTime" movie can be played forward or backward, can be halted at any frame, or can be stepped one frame at a time, in either the forward or backward directions. An example of an animated sequence illustrating the distinction between path lines, streak lines, and streamlines for the unsteady flow past a circular cylinder of time-varying radius is shown in Fig. 2. Unfortunately, in the format of this paper, only a single frame of the video can be shown.

Active graphs are used to display quantitative results that depend on one or more parameters that might be changed by the student. They are identified by the active "calculator" icon, as shown on the page illustrated in Fig. 3. Clicking anywhere within the thumbnail sketch for the graph launches the active version of the graph. In most plots of quantitative data presented in the textbook, the region near a particular point on any curve can be zoomed for more accurate reading of values. A click of the mouse button on any point in the plot doubles the scale in the vicinity of that point, or a "zoom box" can be defined by clicking and dragging between opposite corners to define an area to be enlarged to the full plot area. A double click of the mouse button anywhere in the figure area zooms the plot out to the original scale.

The active graph on the page shown in Fig. 3 presents the pressure ratio and shock angle for the turning of supersonic flow through a given angle. The result depends upon both the upstream Mach number and the ratio of specific heats ($k = c_p / c_v$) of the gas. When the student changes either the Mach number or the ratio of specific heats, the plot is re-drawn automatically to



12 Compressible Flows

12.10.2c Oblique Shock Waves, cont'd

Fig. 12.25 Turning angle as a function of shock angle for different Mach numbers; calorically perfect air (k = 1.4). Click for interactive module

Since we can write

$$\frac{V_{t2}}{V_{n2}} = \frac{V_{t2}/V_1}{V_{n2}/V_{n1} \cdot V_{n1}/V_1} = \frac{\cos\beta}{\sin\beta \cdot V_{n2}/V_{n1}} = \frac{\rho_2/\rho_1}{\tan\beta}$$

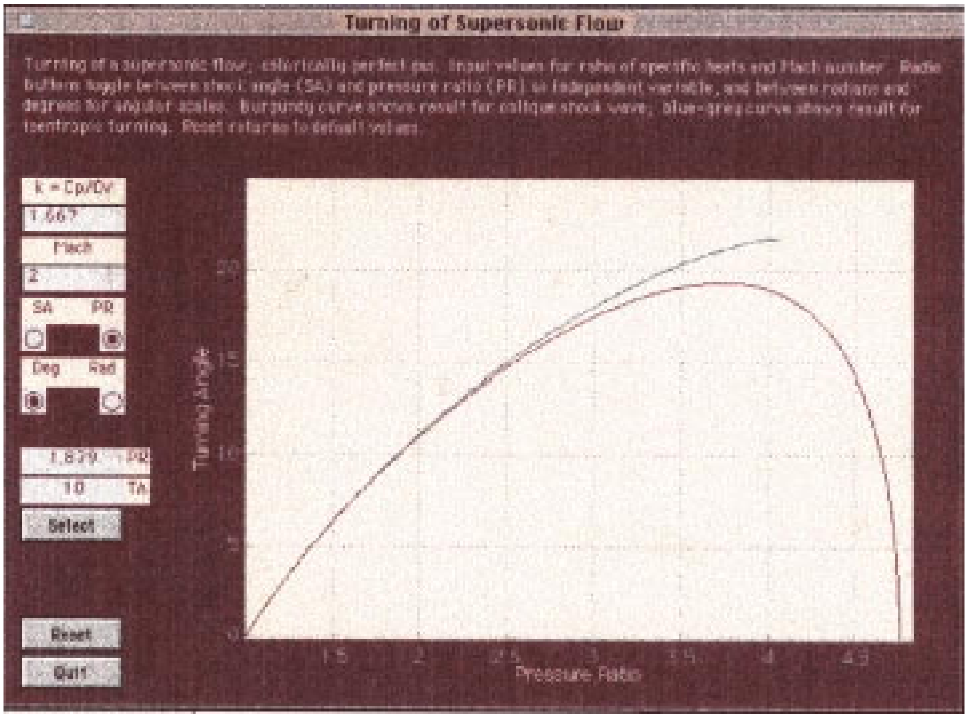
where the density ratio is given as a function of $\mathbf{M}_{n1}$ by (12.34b), the expression for the turning angle can be written

$$\theta = \beta - \frac{\pi}{2} + \arctan\left[\frac{\mathbf{M}_{n1}^2}{1 + \frac{k-1}{k+1}\left(\mathbf{M}_{n1}^2 - 1\right)} \cdot \cot\beta\right] \quad (12.75)$$

Plots of this result for the flow of any calorically perfect gas and any Mach numbers can be viewed in Fig. 12.25.

cont'd

Fig. 3 Page having an active graph. The "calculator icon" indicates that the thumbnail sketch can be clicked on to launch the active graph.



Turning of a supersonic flow; calorically perfect gas. Input values for ratio of specific heats and Mach number. Radio buttons toggle between shock angle (SA) and pressure ratio (PR) as independent variable, and between radians and degrees for angular scales. Burgundy curve shows result for oblique shock wave; blue-grey curve shows result for isentropic turning. Reset returns to default values.

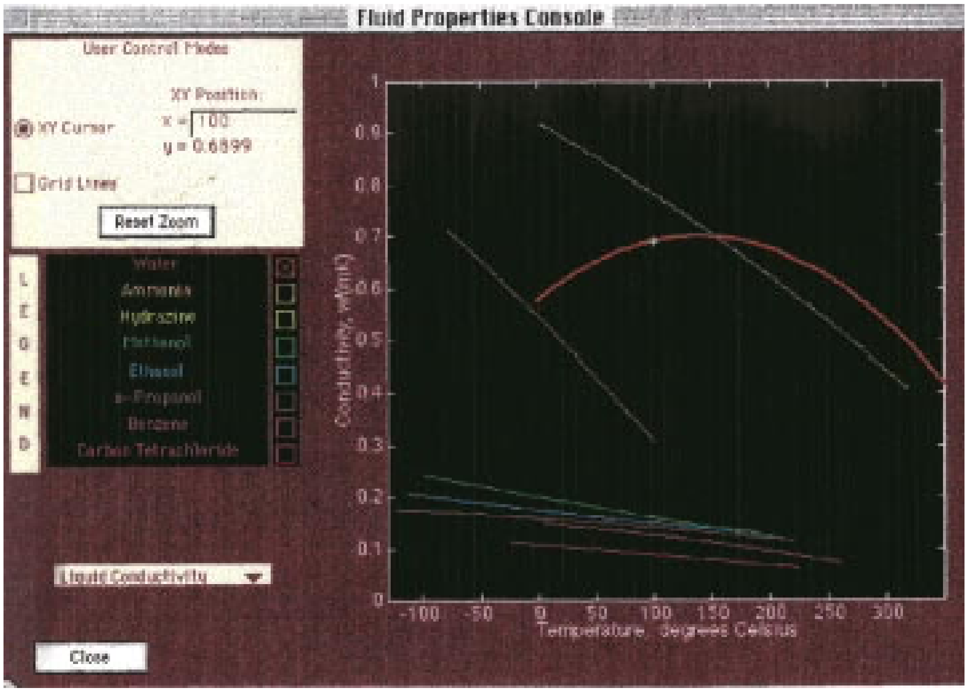
k = Cp/Cv: 1.667

Mach: 2

1.879 PR

10 TA

Fig. 4 An active graph presenting the pressure ratio as a function of turning angle for supersonic flow. The figure illustrates the curves for a single upstream Mach number and ratio of specific heats, but the student can enter any values for these parameters to see the corresponding plot.



x = 100
y = 0.6899

Fig. 5 An active graph presenting the dependence of coefficient of thermal conductivity on temperature for several common liquids. The student can read values by clicking on the desired curve or by entering the temperature value in the corresponding data box and reading the resulting thermal conductivity once a particular curve has been selected. The curves are computed using semi-empirical formulas and experimental values from Reid, Prausnitz & Poling[5].

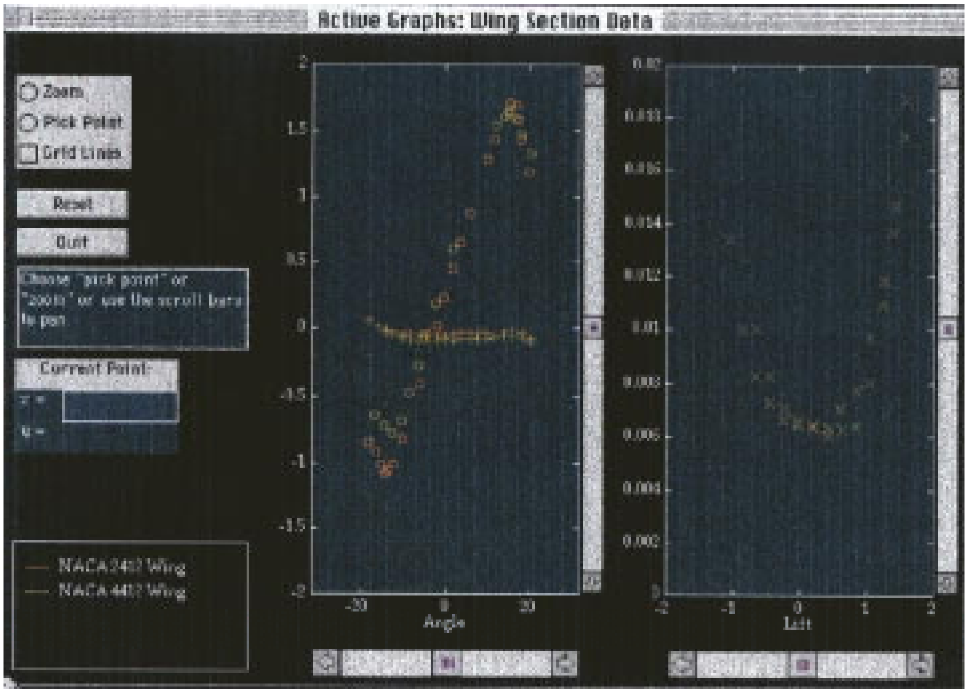
correspond to the new values. The active version of this plot is shown (for upstream Mach number $\mathbf{M} = 2.0$ and ratio of specific heats $k = 1.667$) in Figure 4. When the pressure ratio is selected as the dependent variable (rather than the shock angle) the corresponding curve for isentropic flow also is plotted. The student can choose to have angles displayed either in radians or degrees by clicking on the appropriate radio button.

Active graphs also are used to display fluid properties, such as coefficient of viscosity or thermal conductivity, and can be launched by selecting the relevant fluid property from the "Data" menu. Figure 5 shows the dependence of coefficient of thermal conductivity on temperature for several common liquids.

As a final example, Fig. 6 shows the lift, drag, and pitching moment characteristics of several airfoils tested by the National Advisory Committee on Aeronautics (NACA), the forerunner of NASA. Once the active graph has been launched, the student can display the lift, drag, and pitching moment characteristics of several different airfoil shapes.

4. Computation and Utilities

Computational utilities to solve a variety of engineering problems that may require the solution of nonlinear (or systems of nonlinear) equations are integrated into the textbook. Historically, these problems were solved by graphical methods, by iteration on tabulated functions requiring repeated interpolation, or by requiring the student to implement numerical methods. The interactive textbook provides Graphical User Interfaces (GUIs) to a number of routines written in MATLAB that allow the student to solve a greater variety and number of such problems in the same amount



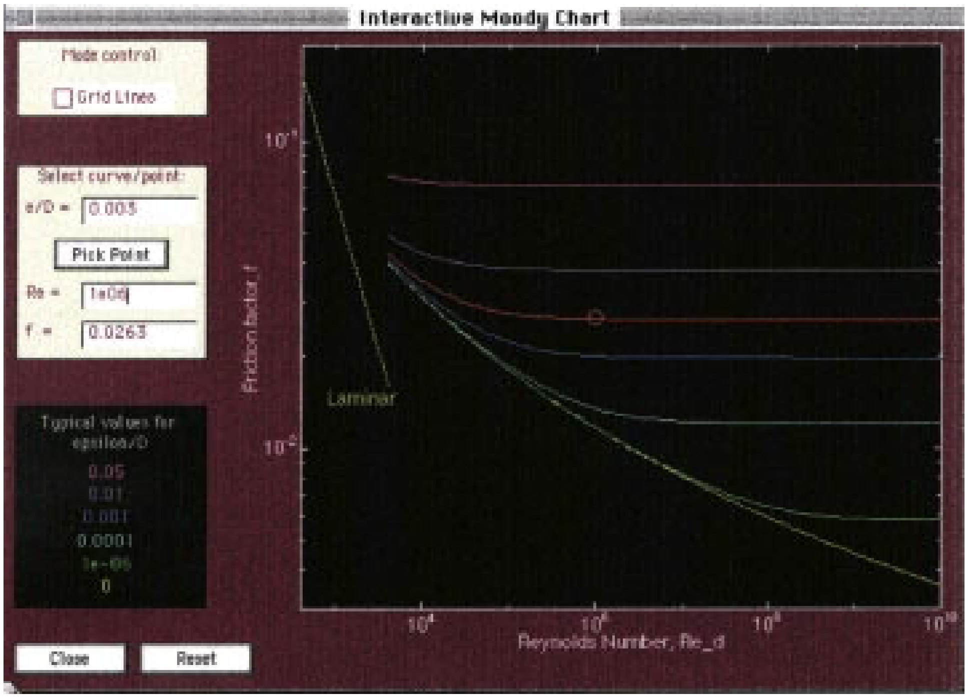
Choose "pick point" or "zoom" or use the scroll bars to pan.

Fig. 6 Lift, drag, and moment coefficients for a variety of NACA airfoil shapes. (a) Lift and moment coefficients are plotted as functions of the angle of attack; (b) the drag coefficient is plotted as a function of the lift coefficient. After Abbott & von Doenhoff[1].

of time (or with similar effort required) to solve a single problem in the past. Such GUI-based utilities are provided for solving:

- Compressible flows with area change for isentropic flows and flows with normal shock waves;
- Potential flow problems within arbitrary two-dimensional geometries for any mix of Neumann and Dirichlet boundary conditions;
- Dimensional analysis problems to find dimensionless groups of variables for any set of dimensional input variables;
- Solutions of steady-state pipe network flows;
- Compressible flows in constant area ducts with friction;
- Numerical integration of functions;
- Incompressible pipe flow problems to determine pipe sizes, flow rates, head loss, etc., for a variety of formulations of the frictional losses for turbulent flow;
- One- and two-dimensional plotting of functions and data with integration to determine volumes under surface plots;
- Planar and axisymmetric incompressible potential flow problems by superposition of elementary solutions (sources, sinks, vortices, doublets, and uniform streams);
- Solutions of open channel profile problems;
- Compressible flows in constant area ducts with heat addition;
- Any system of nonlinear or transcendental equations;
- Properties in the standard atmosphere;

- Units conversion problems from any system to virtually any other;
- Waterhammer problems in elastic pipes.



e/D = 0.003
Re = 1e06
f = 0.0263

Fig. 7 Moody chart for the friction factor in fully-developed, turbulent pipe flow. The friction factor is a function of the nondimensional pipe roughness and the Reynolds number.

Here, we will provide brief descriptions of three of the utilities to give an indication of the features.

4.1 Pipe Flow

The head loss h_f due to friction for fully-developed turbulent flow in a length L of circular pipe having diameter d is given by the relation

$$h_f = \mathbf{f}\frac{L}{d}\frac{V^2}{2g}\,, \tag{4.1}$$

where V is the mean flow velocity, g is the acceleration of gravity, and $\mathbf{f}$ is the (Darcy-Weisbach) friction factor. The friction factor is a function only of the non-dimensional roughness ε / d of the pipe surface and the Reynolds number $\mathbf{Re}_d = Vd / \nu$, where ν is the kinematic viscosity of the fluid. The formula due to Colebrook[2] provides an interpolation between the friction laws for smooth and rough pipes

$$\frac{1}{\mathbf{f}^{1/2}} = -2.0 \log\left(\frac{\varepsilon / d}{3.7} + \frac{2.51}{\mathrm{Re}_d\ \mathbf{f}^{1/2}}\right), \tag{4.2}$$

A plot of the friction factor as a function of Reynolds number, usually called a Moody Chart[4], is shown in Fig. 7.

Use of the Moody Chart is relatively straightforward for some problems. For example, the head loss for a given flow rate through a pipe of specified length and diameter can be found in closed form. For this case, the Reynolds number can be computed directly from the given quantities, and the friction factor determined either (by iteration) from Eq. (4.2) or directly from the Moody Chart. The friction factor can then be used to determine the viscous head loss using Eq. (4.1).

On the other hand, some problems are much more difficult. For example, the diameter for a specified head loss h_f and volumetric flow rate Q through a pipe of given length L requires solution of simultaneous, nonlinear equations. In this case, the Reynolds number must be determined as part of the solution, since neither the average flow velocity V nor the pipe diameter d is known *a priori*. Equations (4.1) and (4.2), recast in terms of the volumetric flow rate Q, must be solved simultaneously to determine the velocity V and diameter d corresponding to the given h_f. Since no general techniques are available to solve such *systems* of nonlinear equations, one usually resorts to simple iterative schemes in which one makes an initial estimate for the pipe diameter, then computes the Reynolds number according to

$$\mathbf{Re}_d = \frac{4}{\pi} \frac{Q}{d\nu}, \tag{4.3}$$

The Moody Chart (or, equivalently, Eq. (4.2)) can then be used to compute the corresponding friction factor $\mathbf{f}$, and a new approximation for the pipe diameter can be computed from the head loss by solving the equation

$$h_f = \frac{8}{\pi^2} \mathbf{f} \frac{L}{d^5} \frac{Q^2}{g},$$

for the diameter

$$d = \left(\frac{8\mathbf{f}}{\pi^2} \frac{L}{h_f} \frac{Q^2}{g} \right)^{1/5}, \tag{4.4}$$

If the flow contains entrance or exit losses, minor losses, or a pump or turbine, the equations become even more complex. This process is quite tedious if the Moody Chart or Eq. (4.2) must be used to determine the friction factor for the Reynolds number at each iteration, but it can be automated quite easily on the computer.

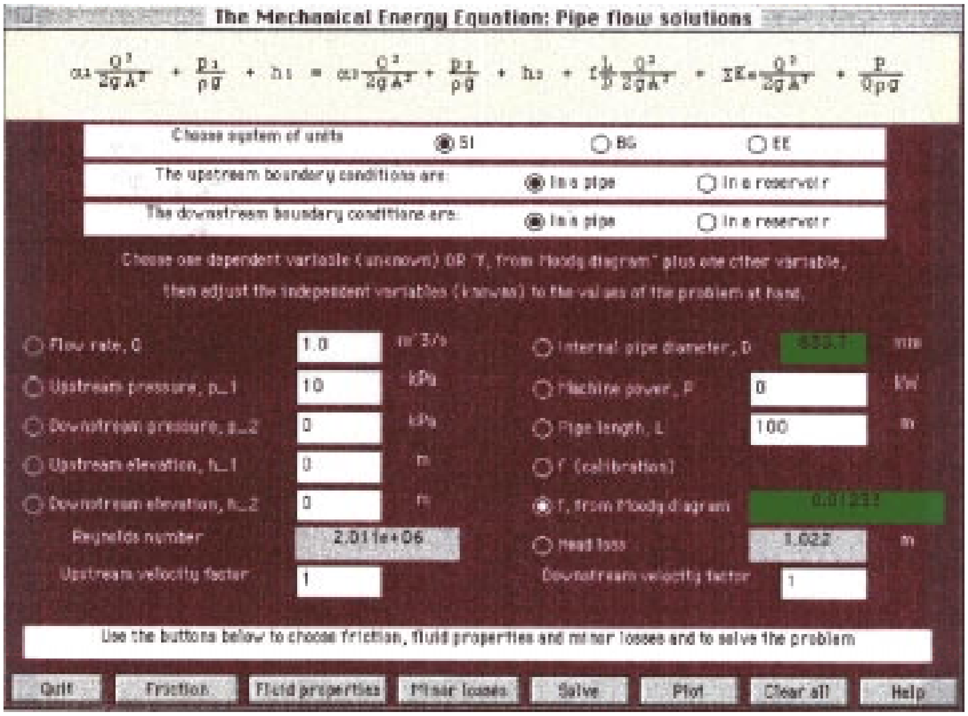
The solution to this problem, and many others, can be found using the GUI-based utility PipeFlow. PipeFlow solves the one-dimensional mechanical energy equation, the Darcy-Weisbach equation (4.1), and the Colebrook equation (4.2), either singly or in combination, for the friction factor $\mathbf{f}$ plus another variable, allowing for various boundary conditions and minor losses. It can choose roughness from a pipe type (e.g., concrete), look up water properties (or allow specification of the properties of other fluids), and find real (instead of nominal) pipe dimensions from a table of commercial pipe sizes. The main screen of PipeFlow used to specify these problems is shown in Fig. 8. The screen shown in Fig. 8 illustrates the solution of one example of the type of problem just described. The solution is displayed for the pipe diameter required for a fully-developed turbulent flow of $Q = 1.0$ m^3/s in a pipe having length $L = 100$ m, driven by a pressure difference of $\Delta p = 10,000$ Pa.

4.2 Compressible Flow with Area Change

Another example of problems for which an inexpensive personal computer can be used to advantage to allow solution of problems that are otherwise rather tedious involves compressible flows in ducts of varying cross-sectional area. The Mach number $\mathbf{M}$ as a function of cross-sectional area A for isentropic flow in a nozzle can be expressed as

$$\frac{A}{A^*} = \frac{1}{\mathbf{M}} \left(\frac{2 \ + \ (k-1)\mathbf{M}^2}{k+1} \right)^{\frac{k+1}{2(k-1)}}, \tag{4.5}$$

where k is the ratio of specific heats and A^* is the effective sonic area for the nozzle (which may, or may not, correspond to the minimum area of the nozzle under consideration, depending upon



The Mechanical Energy Equation: Pipe flow solutions

$$\alpha_1 \frac{Q^2}{2gA^2} + \frac{p_1}{\rho g} + h_1 = \alpha_2 \frac{Q^2}{2gA^2} + \frac{p_2}{\rho g} + h_2 + f \frac{L}{D} \frac{Q^2}{2gA^2} + \Sigma K_m \frac{Q^2}{2gA^2} + \frac{P}{Q\rho g}$$

Choose system of units ● SI ○ BG ○ EE

The upstream boundary conditions are: ● In a pipe ○ In a reservoir

The downstream boundary conditions are: ● In a pipe ○ In a reservoir

Choose one dependent variable (unknown) OR "f, from Moody diagram" plus one other variable, then adjust the independent variables (knowns) to the values of the problem at hand.

○ Flow rate, Q: 1.0 m^3/s

○ Upstream pressure, p_1: 10 kPa

○ Downstream pressure, p_2: 0 kPa

○ Upstream elevation, h_1: 0 m

○ Downstream elevation, h_2: 0 m

Reynolds number: 2.011e+06

Upstream velocity factor: 1

○ Internal pipe diameter, D: 633.7 mm

○ Machine power, P: 0 kW

○ Pipe length, L: 100 m

○ f (calibration)

● f, from Moody diagram: 0.01232

○ Head loss: 1.022 m

Downstream velocity factor: 1

Use the buttons below to choose friction, fluid properties and minor losses and to solve the problem

Fig. 8 Main panel for PipeFlow, a GUI-based utility for solving a variety of fluids problems involving turbulent flow losses in pipes.

the operating conditions); see, e.g., Shapiro[6]. Once the Mach number is known, other flow variables can be determined directly from the isentropic relations; e.g., the pressure p is given by

$$\frac{p_0}{p} = \left(1 + \frac{k-1}{2}\,\mathbf{M}^2\right)^{\frac{k}{k-1}} \tag{4.6}$$

where p_0 is the isentropic stagnation pressure.

For completely isentropic flow, the problem of determining the flow properties as a function of nozzle cross-sectional area is simply one of solving Eq. (4.5) or, equivalently, interpolating tabulated values of this function, to determine the Mach number $\mathbf{M}$, and then using formulas such as Eq. (4.6) to determine the flow properties from the Mach number.

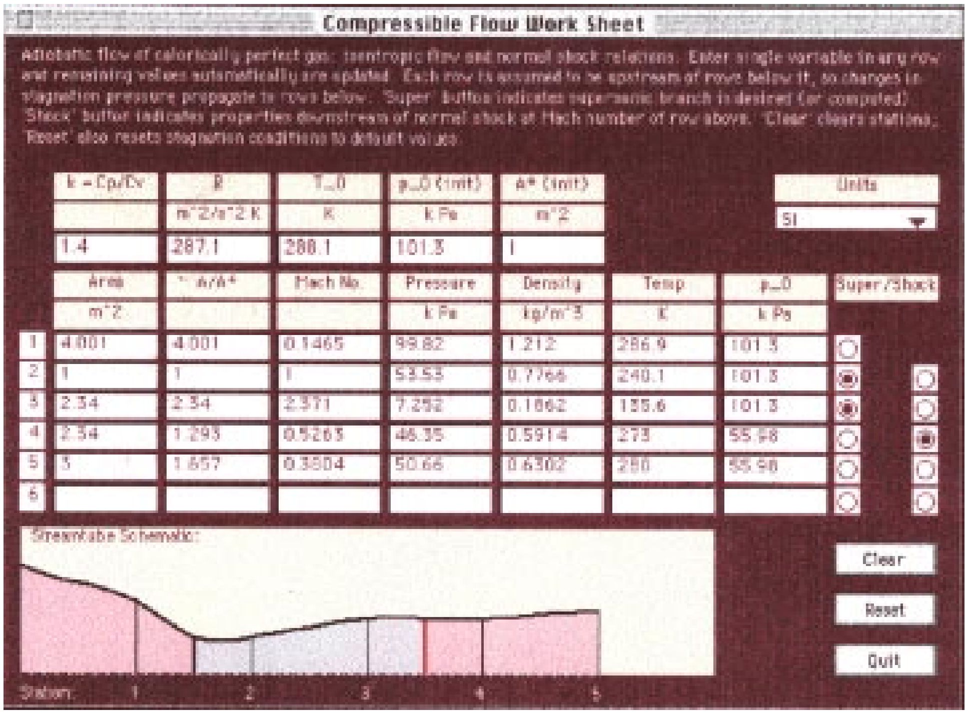
If shock waves are present in the nozzle, the problem becomes more complex. A typical problem, usually too tedious even to assign as a homework exercise, is to determine the shock location in a nozzle of given geometry (i.e., for a given $A(x)$) for a given exit pressure.[*] This problem, like many of the turbulent pipe-flow problems described in the previous section, requires iteration. For a given shock location, the jump in pressure can be determined from the normal shock relations, e.g.,

$$\frac{p_2}{p_1} = 1 + \frac{2k}{k+1}\left(M_1^2 - 1\right) \tag{4.7}$$

---

[*] An analogous problem exists in open channel flow where the location of a hydraulic jump is to be determined for specified boundary conditions and channel geometry. The channel profile solver in combination with the transcendental equation solver provides an easy method of solution for that problem.

where the subscripts $(\ )_1$ and $(\ )_2$ refer to the states immediately upstream and downstream of the shock, respectively. The Mach number immediately downstream of the shock is given by


Compressible Flow Work Sheet

Adiabatic flow of calorically perfect gas: isentropic flow and normal shock relations. Enter single variable in any row and remaining values automatically are updated. Each row is assumed to be upstream of rows below it, so changes in stagnation pressure propagate to rows below. 'Super' button indicates supersonic branch is desired (or computed). 'Shock' button indicates properties downstream of normal shock at Mach number of row above. 'Clear' clears stations; 'Reset' also resets stagnation conditions to default values.

k = Cp/Cv	R	T_0	p_0 (init)	A* (init)
	m^2/s^2 K	K	k Pa	m^2
1.4	287.1	288.1	101.3	1

Units: SI

	Area	A/A*	Mach No.	Pressure	Density	Temp	p_0	Super	Shock
	m^2			k Pa	kg/m^3	K	k Pa		
1	4.001	4.001	0.1465	99.82	1.212	286.9	101.3	○	
2	1	1	1	53.53	0.7766	240.1	101.3	●	○
3	2.34	2.34	2.371	7.252	0.1862	135.6	101.3	●	○
4	2.34	1.293	0.5263	46.35	0.5914	273	55.98	○	●
5	3	1.657	0.3804	50.66	0.6302	280	55.98	○	○
6								○	○

Fig. 9 Data panel for AreaFlow, a GUI-based utility providing access to the equations of compressible flow.

$$\mathbf{M}_2{}^2 = \frac{k+1+(k-1)\left(\mathbf{M}_1{}^2 - 1\right)}{k+1+\ 2k\left(\mathbf{M}_1{}^2 - 1\right)} , \tag{4.8}$$

and this can be used with Eq. (4.5) to determine the new effective A^*, while

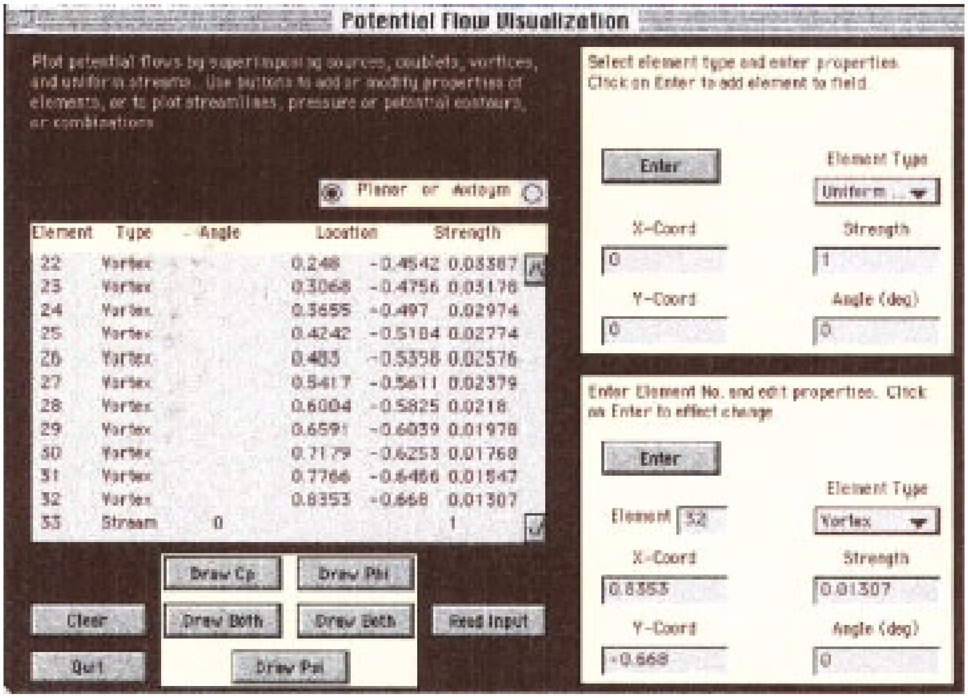
$$\frac{p_{0_2}}{p_{0_1}} = \left(\frac{(k+1)\mathbf{M}_1{}^2}{2\ +\ (k-1)\mathbf{M}_1{}^2}\right)^{\frac{k}{k-1}} \left(\frac{k+1}{2k\mathbf{M}_1{}^2\ -\ (k-1)}\right)^{\frac{1}{k-1}} \tag{4.9}$$

can be used to determine the new stagnation pressure. Equation (4.9), along with the area ratio from the shock to the exit, can be used with the formulas above to compute the pressure at the nozzle exit. Comparing this computed pressure with the desired exit pressure suggests whether the shock needs to be moved upstream or downstream to more nearly match the required exit pressure.

The GUI-based utility AreaFlow, which provides easy access to the complete set of formulas for isentropic flow in ducts of varying cross-sectional area and the normal shock relations, makes sequences of calculations such as that described above less tedious. The data are entered and presented in the form of a spreadsheet that is updated whenever a value is entered or changed so that it always is internally consistent. The AreaFlow display panel is shown in Fig. 9, showing the values computed to determine the shock strength required to produce an exit pressure to upstream stagnation pressure ratio of 0.5 when the exit area is twice the nozzle throat area. This solution was computed in about half a dozen iterations and required less than 3 minutes of user interaction time from start to finish.

4.3 Incompressible Potential Flow

Our final example of the computational utilities integrated into the textbook is PotFlow, which plots selected streamlines, lines of constant velocity potential, and lines of constant pressure for two-dimensional potential flows (either planar or axisymmetric), generated by the superposition of any



Plot potential flows by superimposing sources, doublets, vortices, and uniform streams. Use buttons to add or modify properties of elements, or to plot streamlines, pressure or potential contours, or combinations

Element	Type	Angle	Location		Strength
22	Vortex		0.248	-0.4542	0.03387
23	Vortex		0.3068	-0.4756	0.03176
24	Vortex		0.3655	-0.497	0.02974
25	Vortex		0.4242	-0.5184	0.02774
26	Vortex		0.483	-0.5398	0.02576
27	Vortex		0.5417	-0.5611	0.02379
28	Vortex		0.6004	-0.5825	0.0218
29	Vortex		0.6591	-0.6039	0.01978
30	Vortex		0.7179	-0.6253	0.01768
31	Vortex		0.7766	-0.6466	0.01547
32	Vortex		0.8353	-0.668	0.01307
33	Stream	0			1

Select element type and enter properties. Click on Enter to add element to field.

Element Type: Uniform ...
X-Coord: 0
Strength: 1
Y-Coord: 0
Angle (deg): 0

Enter Element No. and edit properties. Click on Enter to effect change

Element: 32
Element Type: Vortex
X-Coord: 0.8353
Strength: 0.01307
Y-Coord: -0.668
Angle (deg): 0

Fig. 10 Main control panel for PotFlow, a GUI-based utility for studying two-dimensional potential flows generated by superposition of sources, vortices, doublets, and uniform streams.

number of sources, sinks, vortices, doublets, and uniform streams. The main control panel for PotFlow is shown in Fig. 10. Elements are defined and added to the flow using the controls in the upper right corner of the panel, the properties of existing elements can be altered using the controls in the lower right corner of the panel, and the properties of the current elements are displayed in the upper left portion of the panel.

The controls in the lower left portion of the panel are used to clear the data space, to read element data from a file, and to select the form of results to be plotted. Figure 11 shows a plot, generated by PotFlow, of the streamlines and selected contours of constant pressure coefficient for the flow past a discrete vortex representation of a flat plate airfoil at 20 degrees angle of attack. The strengths of 32 vortices, spaced uniformly along the flat plate, were computed using thin-airfoil theory for the case of a flat plate airfoil (a "Level 3" topic in the textbook).

5. Teaching Experiences

Both authors have used preliminary versions of the interactive text book to teach parts, or all, of introductory courses in fluid mechanics to junior-level students in civil and environmental engineering and mechanical engineering. The students generally are enthusiastic about the graphics and computational utilities provided by the new medium, but are less enthusiastic about reading large amounts of text and equations from the screen. This is not really surprising, since computation and video are things that the computer is good at. These dynamic interactions are engaging and encourage active learning. We all are reluctant to spend very much time reading text from a computer screen, given an alternative. This experience has led us to provide a paper version of the text to complement the electronic form, but we still feel the integration of the graphical presentations and computation with the textural material is important.

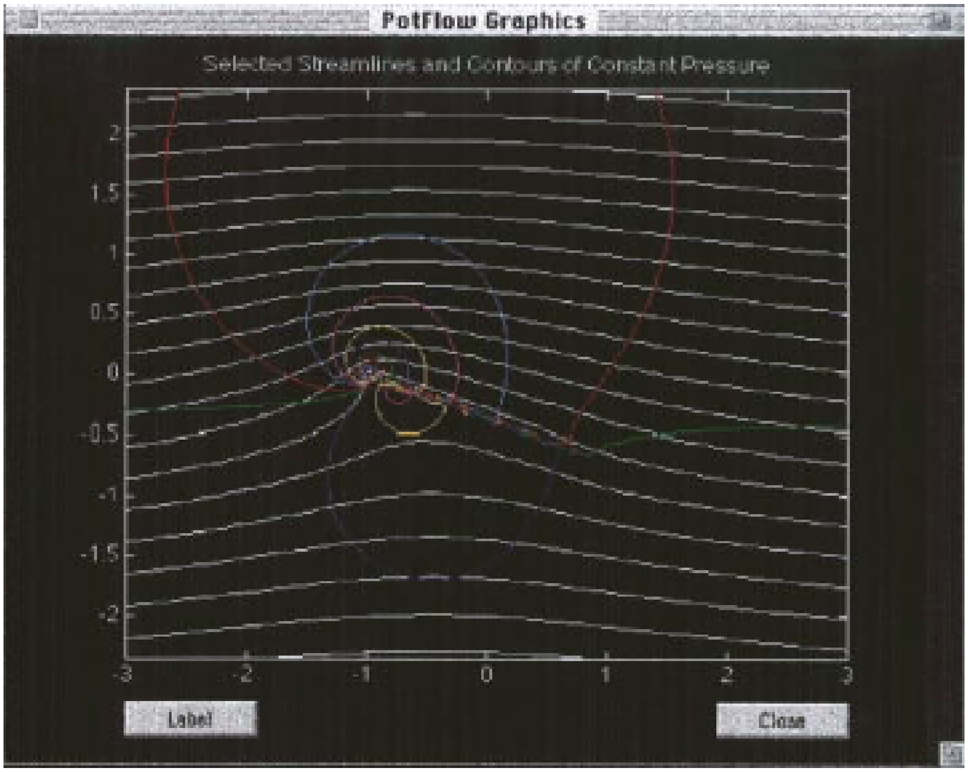


Fig. 11 PotFlow graphics showing streamlines and contours of constant pressure for a discrete vortex approximation to the flow past a flat plate airfoil at 20 degrees angle of attack.

We have experimented with several ways to use the textbook in lecture and recitation sections. In lectures, we have found that it is most effective to have computer projection equipment available, and to use the features of the textbook in a limited way to illustrate topics, rather than to base the entire lecture on projected pages, illustrations, and utilities. Increasingly, computer projection equipment is available in lecture halls, and the ease with which it is possible, in most cases, to switch back and forth between the conventional blackboard mode, video and animation sequences, and computation bodes well for the increased incorporation of tools such as this into lecture presentations.

An instructor can use bookmarks to locate animations, movies, graphs, and active equations easily. The instructor can use these same features on his/her own machine at home for further study and the solution of exercises.

We have found that it is effective to hold at least some recitation sections, especially early in the course, in a facility in which each student has access to a computer. In this way the students can get early experience with the new medium in the presence of a teacher or teaching assistant who is experienced in its use. (Instruction in the use of the text may become less important as the textbook becomes more robust and students become more comfortable with the new medium.) The "studio" environment in which students are encouraged to explore exercises using the various computational utilities under the watchful eye of a roving instructor also has proven effective.

6. Acknowledgements

The textbook was written using MacroMedia AuthorWare and will be distributed on a CD-ROM that will run under the Macintosh, Windows 3.1, Windows 95, and Windows NT operating systems. Video segments and animations are run as QuickTime movies, and all numerical computation is done within MATLAB.

The authors would like to take this opportunity to thank David Dresia, Managing Director of Publications for the American Society of Civil Engineers for his continued support of this project. We also express our profound gratitude to the personnel of the MultiMedia CourseWare Studio at Cornell University for their contributions. In particular, the project could not have been completed without the dedication and expertise of Rob Levine, Kate Mink, Melanie Swain, Mike Tolomeo, Kenny Unice, Dave Wickstrom, John Wolf, and the numerous undergraduate students at Cornell University who have made significant contributions to the success of this project.

7. References

1. I. H. Abbott & A. E. von Doenhoff, **Theory of Wing Sections**, Dover, New York, 1959.
2. C. F. Colebrook, *Turbulent Flow in Pipes, with Particular Reference to the Transition between the Smooth and Rough Pipe Laws*, **J. Inst. Civ. Eng. London**, Vol. 11, pp. 133-156, 1938-39.
3. J. A. Liggett & D. A. Caughey, **Fluid Mechanics: An Interactive Text**, American Society of Civil Engineers, 1998.
4. L. F. Moody, *Friction Factors for Pipe Flow*, **ASME Trans.**, Vol. 66, pp. 671-684, 1944.
5. R. C. Reid, J. M. Prausnitz, & B. E. Poling, **The Properties of Gases and Liquids**, McGraw-Hill, New York, 1987.
6. A. H. Shapiro, **Dynamics and Thermodynamics of Compressible Fluid Flow**, Vol. 1, Ronald, New York, 1953.

8. Biographical Information

David A. Caughey is Professor and Director of the Sibley School of Mechanical and Aerospace Engineering at Cornell University. He received his baccalaureate degree in Aeronautical and Astronautical Engineering from the University of Michigan and the Ph.D. in Aerospace and Mechanical Sciences from Princeton University. Before joining Cornell, Dr. Caughey worked as a research scientist at the McDonnell Douglas Research Laboratories in St. Louis, Missouri.

James A. Liggett is Professor Emeritus in the School of Civil and Environmental Engineering at Cornell University. He received his baccalaureate degree from Texas Technological College and the M.S. and Ph.D. degrees from Stanford University. Dr. Liggett has also held positions at the University of Wisconsin and Chance-Vought aircraft.