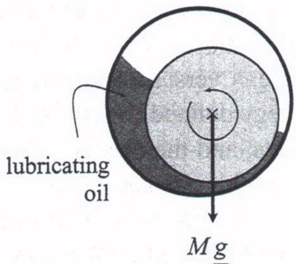


Figure 7.52 Schematic of a lubrication flow. The central axis may be the axle of an automobile, for example, and the outer ring represents part of the housing.

The concept of the boundary layer is important in fluid mechanics and it is discussed again in Section 8.2. Boundary-layer formation is a common trait of flows for which both inertial and viscous contributions are important. For engineering applications, consideration of the boundary layer is essential because heat and mass transfer often occur through walls; thus, heat and mass must traverse the boundary layer.

When the inertial contribution to the flow momentum is slight, analytical solutions are sometimes found using a quasi-unidirectional technique known as the lubrication approximation. Lubrication flow is named for flow in narrow gaps between moving parts in which the role of the fluid is to lubricate the parts (Figure 7.52). In such gaps, the flow is only slightly different from unidirectional; the lubrication approximation takes advantage of this similarity by considering the flow to be locally unidirectional and parallel [43]. With this assumption, analytical solutions may be found. The lubrication approximation is useful in polymer-processing flow calculation. Chapter 13 in Denn [43] discusses the lubrication approximation.

This chapter demonstrates that the continuum modeling method is versatile and capable of providing insight to a wide variety of flow problems. The overall strategy is outlined in Section 7.1.2.3: When tackling a difficult flow problem, begin by identifying an idealized version of the flow that can be solved. Then, use the solution to the idealized problem to nondimensionalize the equations of change so that information in the governing equations can be accessed. Finally, solve for $\underline{v}$ and $p$ or conduct experiments and develop data correlations so that the engineering problem may be solved.

Problems that are unidirectional and steady are not difficult to solve—the left-hand side of the Navier-Stokes equation goes to zero, eliminating the nonlinear terms. When we stray from these flows, inertia becomes increasingly important and the flow behavior becomes more complex and fascinating. Chapter 8 confronts these issues as we move on to external flows, which almost always exhibit both viscous and inertial contributions.

## 7.4 Problems

1. The governing equations for fluid flow are four coupled equations in four unknowns. What are these equations? What is a strategy for solving them?
2. What is the role of dimensional analysis in fluid mechanics?
3. Using the methods in this chapter, write the continuity equation (i.e., microscopic-mass balance) in dimensionless form. What can we learn from the result?
4. Figure 7.6 plots results for the velocity and pressure profiles for steady, Poiseuille flow in a tube. We choose to plot these functions using

dimensionless combinations of the variables and characteristic quantities. Why do we use dimensionless combinations? What difficulties would we encounter if we choose to plot the bare $v_z$ versus $r$ and $p$ versus $z$?

5. In terms of the problem-solving strategy defined in Section 7.1.2.3, identify the idealized problem, the experiments, and the data correlations that were used to solve the burst-pipe problem of this chapter.
6. Complete the calculation of the velocity profile and the total-stress tensor for steady, pressure-driven flow in a tube (i.e., Poiseuille flow in a tube). In other words, show that Equations 7.22, 7.23, and 7.34 result from the integration and application of Equations 7.18 and 7.19.
7. Show that the Hagen-Poiseuille equation (Equation 7.28) for pressure drop as a function of flow rate in laminar flow follows from the integration of the velocity field across the pipe cross section (Equation 7.26).
8. In the calculation of total drag in a pipe, show using matrix calculations that the simplified expression in Equation 7.122 is equivalent to the definition of axial drag in Equation 7.120.
9. In laminar flow in a tube, calculate the axial drag by beginning with the surface integral in Equation 7.125 and incorporating the solution for the velocity profile. Neglect the effect of gravity.
10. The solution for pressure-driven laminar flow in a tube includes the effect of gravity. How does the solution change if the flow is upward instead of downward? How does the solution change if the pipe is mounted at a 30-degree angle to horizontal? Show that the effect of gravity in all cases can be accounted for by defining the dynamic pressure as given here [43] (see the Glossary):

$$\mathcal{P} \equiv p - \rho g_z Z$$

11. For the burst-pipe problem discussed in this chapter, we first attempt to solve by assuming laminar flow. For the laminar-flow result, what was the Reynolds number calculated in the small pipe? If the flow could have remained laminar up to that Reynolds number (it cannot; the flow becomes unstable), what would have been the Fanning friction factor? Compare this number and the pressure drop it implies to the actual $f$ and $\Delta p$ that we calculated. Discuss your answer.
12. We neglect the presence of fittings and the velocity change in the burst-pipe example in this chapter. What would be the effect on the burst-pipe calculation if we include the frictional loss due to velocity head, bends, fittings, and valves? Assume that there are eight 90-degree bends, two gate valves, and one globe valve half open in the smaller piping section.
13. We assume a smooth pipe in the burst-pipe example in this chapter. Repeat Example 7.5 assuming that the pipes are galvanized iron with a pipe roughness of 0.0005 foot. Was smooth pipe a good assumption?
14. An 80-foot section of 1/2-inch ID Schedule 40 piping branches into two pipes of the same diameter, one of which is 160 feet long and the other 200.0 feet long (all horizontal). The main pipe is connected to the municipal water supply, which supplies a constant 50.0 psig at the pipe entrance. What are the flow rates through the two pipe exits? What is the pressure at the

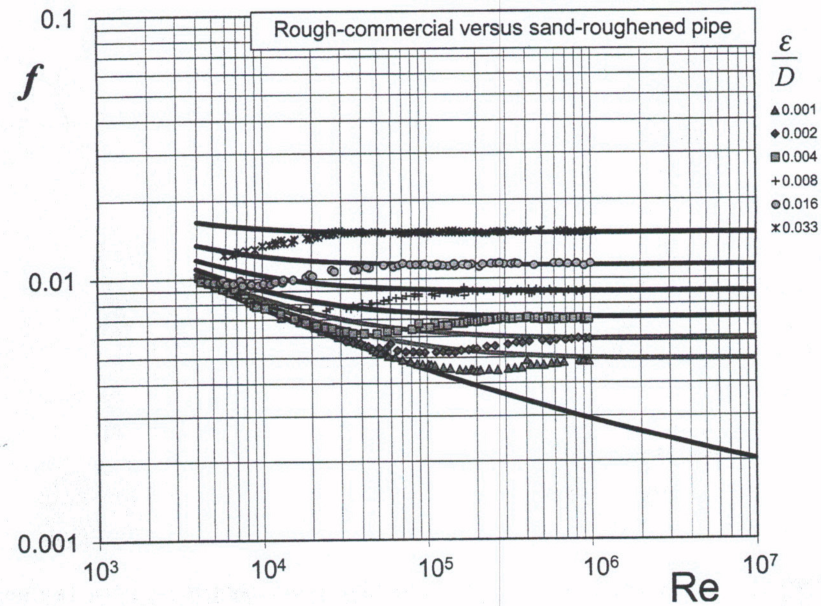


Figure 7.53 Data on rough commercial pipes, represented by the Colebrook correlation (solid lines), are compared with the data of Nikuradse [126] for sand-roughened pipes (discrete points). The two measurements agree at large Re but not at lower values (Problem 17).

splitting point? Assume smooth pipes; do not consider friction losses due to fittings.

15. For turbulent pipe flow, show that Equation 7.156—the Prandtl correlation for fluid friction—is equivalent to the case $\varepsilon = 0$ in the Colebrook correlation (Equation 7.161).
16. For steady pipe flow, repeat branched-piping, Example 7.9 for pipes with roughness $\varepsilon = 0.05$ mm.
17. The Colebrook correlation (i.e., Equation 7.161) gives friction factor as a function of Reynolds number and roughness ratio for commercial pipes. The values of roughness $\varepsilon$ for commercial pipes were deduced by comparing the measured asymptotic values of $f$ for real pipes, with the values for $f$ at large Re obtained by Nikuradse [126] on pipes roughened with well-characterized sand of uniform size. The Colebrook equation and Nikuradse's data are compared in Figure 7.53. The two datasets have different shapes at Reynolds numbers below the asymptotic values. What differences can you think of between the wall surfaces on commercial pipes and those on the artificially roughened walls of Nikuradse that might account for these differences? Discuss your answer.
18. In Section 7.1.1, we initially neglect the pressure difference $p_0 - p_L$ when analyzing the Cannon-Fenske viscometer (see Figure 7.11) before ultimately resorting to experimental calibration to account for the small pressure effect (see Equation 7.56). We can account for the pressure difference $p_0 - p_L$ more formally by performing a quasi-steady-state analysis on the system.

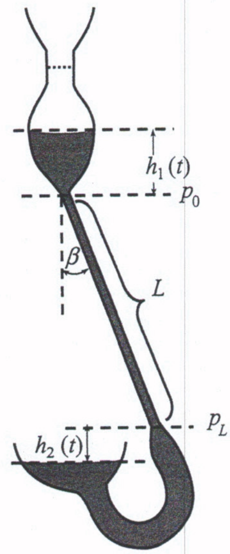


Figure 7.54 The Cannon-Fenske viscometer measures fluid viscosity by allowing the user to time the passage of a set volume of fluid through a long narrow capillary. The flow is driven primarily by gravity; the imposed pressure drop due to the changing driving fluid head $h_1(t)$ and the back pressure due to the head $h_2(t)$ may be accounted for by applying a quasi-steady-state analysis, as described in Problem 18.

Consider the expanded view of the Cannon-Fenske viscometer shown in Figure 7.54. Let $h_1(t)$ represent the time-dependent height of the upper meniscus above the second timing mark and $h_2(t)$ represent the time-dependent height difference between the fluid level in the lower reservoir and the exit of the capillary tube. In the quasi-steady-state approach, we write relationships between variables as if time were moving slowly and the system were nearly in steady state.

(a) Using the principles of fluid statics on our quasistationary system, what is the relationship among $p_0$, $h_1$, and atmospheric pressure?
(b) Using the same approach, what is the relationship among $p_L$, $h_2$, and atmospheric pressure?
(c) Writing the volumetric flow rate $Q$ as the rate of change of the fluid volume V in the upper reservoir $-dV/dt$, integrate the appropriate equation for volume with respect to time from 0 to $t_{\text{efflux}}$ to obtain a pressure-corrected equation for the measurement of fluid kinematic viscosity $\nu$ with the Cannon-Fenske viscometer. Assume that $h_1(t)$ and $h_2(t)$ vary linearly with time throughout the experiment:

Answer:

$$\frac{\mu}{\rho} = \left[\frac{\pi R^4 g}{8VL}\left(\frac{h_1(0)}{2} + \frac{h_2(0)}{2} + h_2(t_{\text{efflux}})/2 + L\cos\beta\right)\right] t_{\text{efflux}}$$

(d) Do $h_1(t)$ and $h_2(t)$ vary linearly with time? How important is this effect?

19. When using a calibrated Cannon-Fenske viscometer, it is necessary to employ the same fluid volume as during calibration. To achieve this, the viscometer

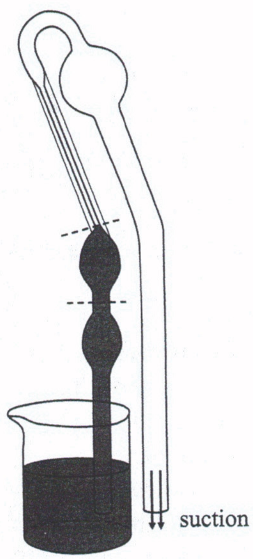


Figure 7.55 Schematic of the inverted loading technique that is required when using a Cannon-Fenske viscometer (Problem 19).

is loaded with fluid as shown in Figure 7.55. The viscometer is inverted into a beaker of fluid and suction is applied to the cleaning arm. In the inverted position, when the fluid reaches the timing mark nearest the capillary, the correct volume has been loaded.

When several concentrations of solution are being measured as part of a sequence, it is convenient to dilute a concentrated solution within the viscometer to make the subsequent measurements on less concentrated solutions. This technique is used in the study of polymers [60]. The Cannon-Fenske viscometer is inappropriate for this type of measurement due to the excess, unknown back pressure that would result from adding additional solvent.

The Ubbelhode viscometer is similar to the Cannon-Fenske, but the exit of the capillary in the former is vented, preventing the back-pressure problem (Figure 7.56). Following the quasi-steady-state technique outlined in Problem 18, calculate the equation that relates kinematic viscosity and efflux time in the Ubbelhode viscometer.

20. Liquid with the physical properties of water flows in a tube in laminar flow. A researcher studying biological flows in tubes wants to conduct experiments on the apparatus and must replace part of the wall with a different solid material that is transparent to a particular kind of electromagnetic radiation. What is the force on the patch of the wall being replaced? The patch is one-eighth the circumference of the tube and is of length $l$.
21. What is the purpose of the concept of the hydraulic diameter?
22. The correlation between the Fanning friction factor and the Reynolds number for turbulent flow through pipes (circular cross section) is shown in the Moody plot (Figure 7.22). Which plot do we use for the correlation of $f(\text{Re})$ for noncircular conduits? Explain.

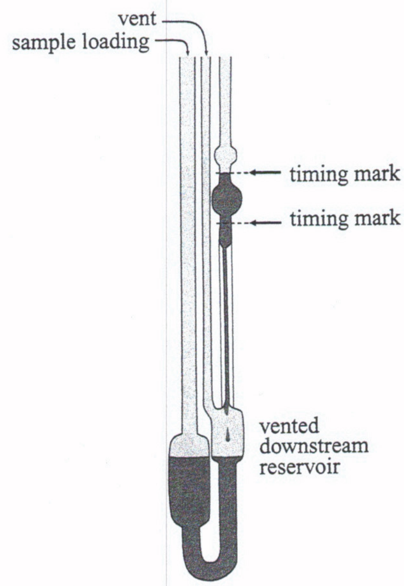


Figure 7.56 Schematic of a Ubbelhode viscometer. The Ubbelhode viscometer is vented at the exit of the capillary. Venting the exit ensures that the pressure at the exit is known (i.e., atmospheric) and allows the sample volume to vary (Problem 19).

23. Hydraulic radius [174] in a noncircular conduit is defined as:

$$\text{Hydraulic radius} \qquad r_H \equiv \frac{A}{\not{p}}$$

where $A$ is the cross-sectional area of the conduit and $\not{p}$ is the wetted perimeter of the conduit. With this definition, how are hydraulic radius and hydraulic diameter related? Discuss your answer.

24. For steady flow in a duct of rectangular cross section, carry out the integrations in Equation 7.229 to obtain the analytical expression for the wall drag in pressure-driven flow.

25. Calculate the Poiseuille number, $f_{D_H}\text{Re}_{D_H}$, for a conduit with elliptical cross section; compare your result with Figure 7.36. The major axis of the ellipse is of length $2a$ and the minor axis is $2b$. The velocity field for laminar flow through a conduit of elliptical cross section is given by White [174] as:

$$v_x = \frac{1}{2\mu}\frac{\Delta p}{L}\frac{a^2b^2}{a^2+b^2}\left[1-\frac{y^2}{a^2}-\frac{z^2}{b^2}\right]$$

The average velocity in this conduit is given by:

$$V = \langle v \rangle = \frac{\Delta p}{4\mu L}\frac{a^2b^2}{a^2+b^2}$$

What is the friction-factor/Reynolds-number relationship for this geometry?

26. In steady, pressure-driven, planar-slit flow of an incompressible Newtonian fluid, calculate the vector force on a plane given by the cross section at the exit (see Example 7.10).

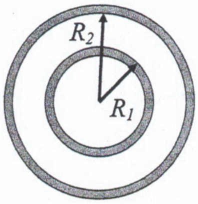


Figure 7.57 For the flow between the inner and outer surfaces of an annulus, the geometry is shown here (Problem 30).

27. Calculate the Poiseuille number, $f_{D_H}\text{Re}_{D_H}$, for a conduit the cross section of which is a rectangle of sides $a$ and $b$ $(b > a)$. What is the friction-factor/Reynolds-number relationship for this geometry?
28. Calculate the Poiseuille number, $f_{D_H}\text{Re}_{D_H}$, for a conduit the cross section of which is a square of side $a$. What is the friction-factor/Reynolds-number relationship for this geometry?
29. Calculate the Poiseuille number, $f_{D_H}\text{Re}_{D_H}$, for a conduit the cross section of which is a slit of infinite width. What is the friction-factor/Reynolds-number relationship for this geometry?
30. Calculate the Poiseuille number, $f_{D_H}\text{Re}_{D_H}$, for flow between the two circular surfaces of an annulus. Let $R_1$ be the outside radius of the inner pipe and $R_2$ be the inside radius of the outer pipe (Figure 7.57). What is the friction-factor/Reynolds-number relationship for this geometry?
31. For flow through a rectangular duct, show that in the limit of infinite width, the solution for velocity (Equation 7.212) becomes the solution for velocity in steady flow through a slit.
32. In Poiseuille flow in a slit, complete the integration in Example 7.10 to obtain the final velocity profile for Poiseuille flow in a slit (Equation 7.188). Calculate the flow rate per unit width by carrying out the missing calculus/algebra to arrive at Equation 7.194. equilateral
33. Water at 25° is forced through an ~~isosceles~~ triangular duct that is 1.0 mm on a side and 5.0 cm long. The driving pressure is 6.0 psig; the exit is open to the atmosphere. What is the flow rate through the slit? Assume the flow to be turbulent.
34. Under what conditions (i.e., limits) does the solution for tangential-annular flow (see figure for Problem 37) approach the parallel-plate solution (Example 6.3)? Using the solution given here, perform a coordinate transformation to show that this is so.

$$\underline{v} = \begin{pmatrix} 0 \\ \left(\frac{\kappa^2 \Omega R}{\kappa^2 - 1}\right)\left(\frac{r}{R} - \frac{R}{r}\right) \\ 0 \end{pmatrix}_{r\theta z}$$

35. For a tank draining through an exit in the bottom, calculate the flow rate by completing a quasi-steady-state calculation like that discussed in Example 7.19. You may neglect friction.

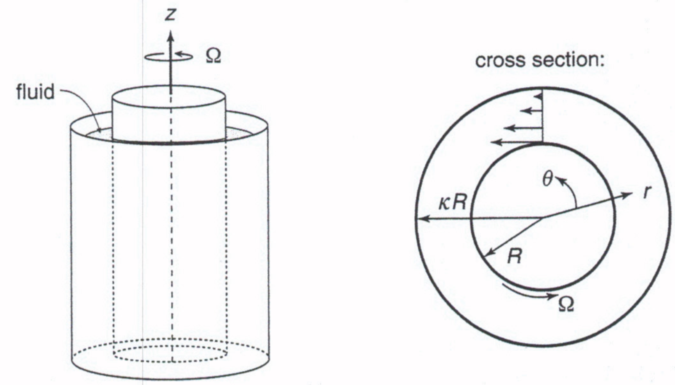


Figure 7.58 Tangential annular flow of a Newtonian fluid (Problem 37).

36. Using a numerical software package, calculate the total force on the wall for pressure-driven flow in a slit. How does your numerical result compare to the analytical result? Use the same boundary conditions in both solutions.
37. *Flow Problem: Tangential annular flow.* An incompressible Newtonian fluid fills the annular gap between a cylinder of radius $\kappa R$ and an outer cup of inner radius $R$ (Figure 7.58). The inner cylinder turns counter clockwise at an angular velocity $\Omega$ radians/s. The flow may be assumed to be symmetrical in the azimuthal direction (i.e., no $\theta$ variation). A pressure gradient develops in the radial direction; the pressure at $z = L$ at the inner cylinder is $p_1$. Calculate the steady state velocity profile, the radial pressure distribution, and the torque needed to turn the inner cylinder.
38. *Flow Problem: Pressure-driven flow of a Newtonian fluid in an annular gap.* Calculate the velocity profile and flow rate for pressure-driven flow of an incompressible Newtonian liquid in the annular gap between two vertical cylinders. The radius of the inner cylinder is $\kappa R$ and the radius of the outer cylinder is $R$. The pressure at an upstream point is $P_0$; at a point a distance $L$ downstream, the pressure is $P_L$. Assume that the flow is well developed and at steady state. You may neglect gravity.
39. *Flow Problem: Pressure-driven flow of a Newtonian fluid in an annular gap, numerical.* Solve Problem 38 using computer simulation software [27]. Calculate the forces on both the inner and outer surfaces.
40. *Flow Problem: Flow due to natural convection between two long plates.* The flow between the panes of glass in a double-pane window may be modeled as shown in Figure 7.59. Calculate the velocity profile at steady state. Assume the plates are infinitely long and wide (for answer, see Example 1.11). The density variation with position may be handled as follows. The density of the gas is a function of temperature as given by:

$$\rho = \bar{\rho} - \bar{\rho}\bar{\beta}(T - \bar{T})$$

where $\bar{\rho}$ is the mean density, $\bar{\beta}$ is the mean coefficient of thermal expansion, and $\bar{T}$ is the mean temperature (all constant). The temperature profile

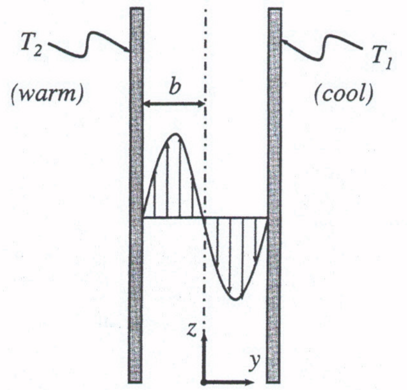


Figure 7.59 Temperature difference generates a flow between two long, wide plates (i.e., hot air rises). We obtain the velocity profile given in Equation 1.140 by using the methods in this chapter in conjunction with energy-balance equations (Problem 40).

obtained from the energy balance is:

$$T = \frac{T_1 - T_2}{2b}y + \frac{T_2 + T_1}{2}$$
$$= \frac{T_1 - T_2}{2b}y + \bar{T}$$

41. *Flow Problem: Radial flow between parallel disks.* An incompressible Newtonian fluid fills the gap between two parallel disks of radius $R$ (Figure 7.60). Fluid is injected through a hole in the center of the top disk, and a steady radial flow occurs. The flow may be assumed to be symmetrical in the azimuthal direction (i.e., no $\theta$ variation). A pressure gradient develops in the radial direction; the pressure near the center is $p_0$ and the pressure at the rim is $p_R$. Calculate the steady state velocity profile and the radial pressure distribution.
42. *Flow Problem: Unsteady one-dimensional flow, startup.* An incompressible Newtonian fluid is in contact with a long, tall wall that initially is stationary

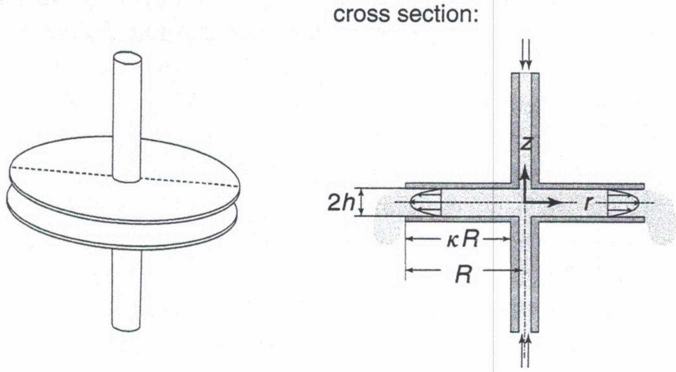


Figure 7.60 Radial flow of a Newtonian fluid from between parallel disks (Problem 41).

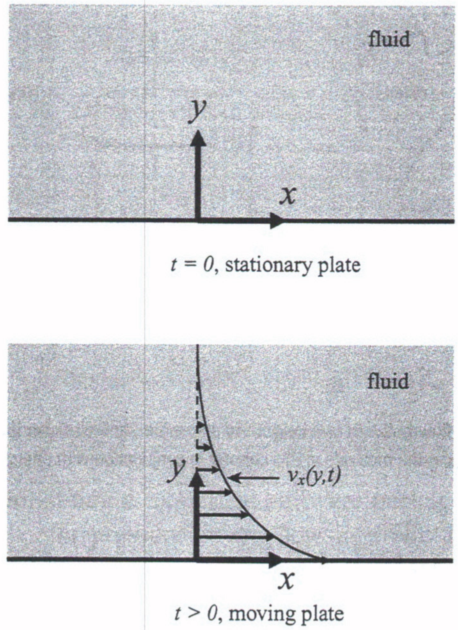


Figure 7.61 Startup flow of a plate in a semi-infinite Newtonian fluid (Problem 42).

(Figure 7.61). The wall suddenly accelerates and moves at steady velocity $V$. The pressure is uniform throughout the flow. Calculate the steady state velocity profile. Plot the velocity solution for various values of time.

43. *Flow Problem: Flow near an oscillating wall.* An incompressible Newtonian fluid is bounded on one side by a wall and is infinite in the $y$-direction (Figure 7.62). The wall is moved back and forth according to:

$$v_x(t)|_{\text{wall}} = V\cos\omega t = \mathcal{R}\{Ve^{i\omega t}\}$$

What is the time-dependent velocity profile in the fluid as a function of position and time? (see also page 102 of [104]).

44. *Flow Problem: Squeeze flow.* An incompressible Newtonian fluid fills the gap between two parallel disks of radius $R$ (Figure 7.63). The disks are subjected to axial forces that cause them to squeeze together. The fluid in the gap responds by producing a combined axial and radial flow that pushes fluid

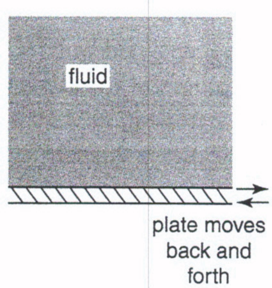


Figure 7.62 A plate forms a boundary for a semi-infinite fluid. The wall is moved according to a sinusoidal function (Problem 43).

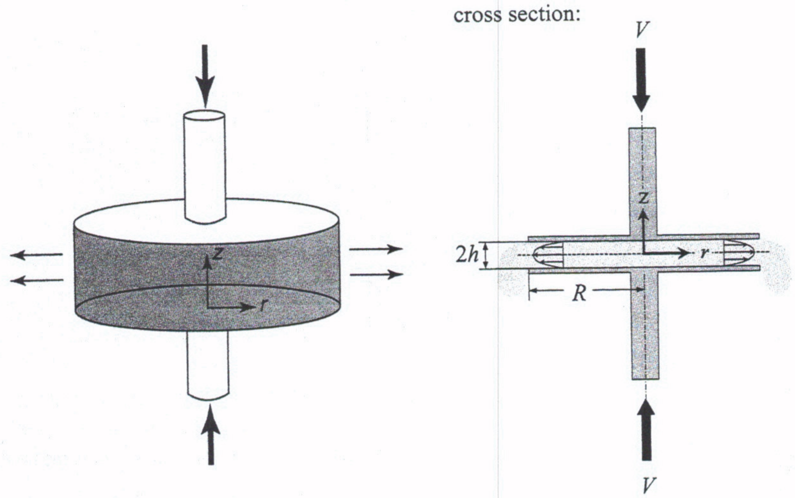


Figure 7.63 Squeeze flow of a Newtonian fluid between parallel disks (Problem 44).

out of the gap. The flow may be assumed to be symmetrical in the azimuthal direction (i.e., no $\theta$ variation). A pressure gradient develops in the radial direction; the pressure at the center is $p_0$ and the pressure at the rim is $p_R$. Calculate the steady state velocity profile and the radial pressure distribution. If the plates are moving with speed $V$, calculate the force needed to maintain the motion.

45. *Flow Problem: Rod turning in an infinite fluid.* A rod rotates counterclockwise in an infinite bath of fluid. What is the velocity field in the fluid? The radius of the rod is $R$, the length of the rod is $L$, and the rod turns at angular velocity $\Omega$ in a fluid of viscosity $\mu$. The flow is steady and the fluid is Newtonian.
46. *Flow Problem: Poiseuille flow in a rectangular duct.* An incompressible Newtonian fluid flows down the axis of a duct of rectangular cross section under the influence of a pressure gradient (Figure 7.31). The width of the duct is 2W and the height of the duct is 2H. The upstream pressure is $p_0$ and the pressure a distance $L$ downstream is $p_L$. Calculate the steady state velocity and pressure profiles. Note: the velocity is three-dimensional and the solution involves a series of hyperbolic trigonometric functions [174].
47. *Flow Problem: Poiseuille flow in a rectangular duct, numerical.* Calculate the velocity field and flow rate for steady, well-developed, pressure-driven flow in a duct of rectangular cross section (Poiseuille flow in a duct; see Figure 7.31). Compare your result to the analytical solution [174].
48. *Flow Problem: Two-dimensional planar flow in a right-angle tee-split, numerical solution.* Flow enters a two-dimensional right-angle tee-split as shown in Figure 7.64. The flow is steady, two-dimensional flow of an incompressible Newtonian fluid (water may be used). Calculate the flow field and the force on the wall as a function of the inlet Reynolds number. Produce appropriate plots to demonstrate the characteristics of the flow.

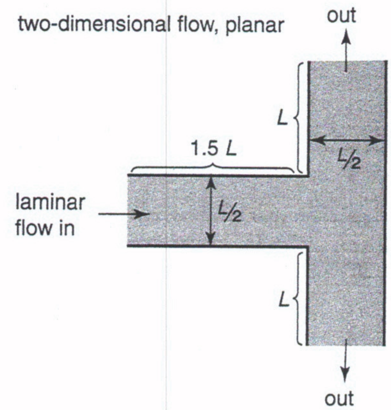


Figure 7.64 Numerical simulation software may be used to calculate the flow domain for two-dimensional planar flow in a right-angle split (Problem 48).

49. *Flow Problem: Two-dimensional axisymmetric flow into radial wall flow in a narrow gap, numerical solution.* Flow exits a pipe at the center of a disk and impinges on a wall producing a radial flow that spreads outward between parallel disks as shown in Figure 7.65. The flow is steady, two-dimensional, axisymmetric flow of an incompressible Newtonian fluid (water may be used). Calculate the flow field and the force on the wall as a function of the inlet Reynolds number. Produce appropriate plots to demonstrate characteristics of the flow.
50. *Flow Problem: Two-dimensional axisymmetric flow through an orifice, numerical solution.* Flow passes through an orifice positioned in the center of a tube as shown in Figure 7.66. The flow is steady, slow, two-dimensional flow of an incompressible Newtonian fluid (water may be used). Calculate the flow field and the pressure drop across the orifice as a function of the inlet Reynolds number. Produce appropriate plots to demonstrate characteristics of the flow.
51. *Flow Problem: Two-dimensional planar cavity flow, numerical solution.* Flow is produced in a cavity by the motion of the top wall as shown in Figure 7.67.

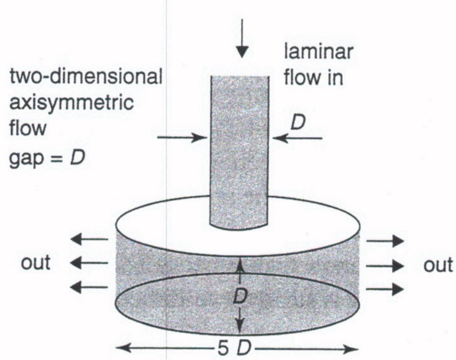


Figure 7.65 Numerical simulation software may be used to calculate two-dimensional axisymmetric flow into radial wall flow in a narrow gap (Problem 49).

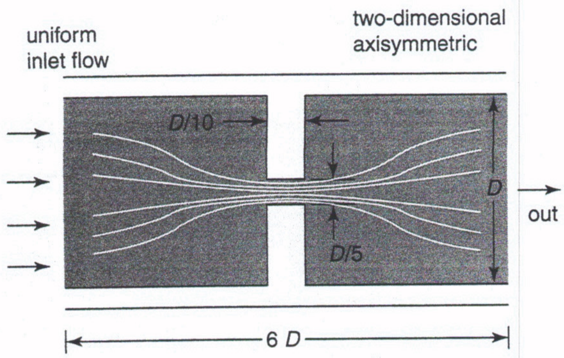


Figure 7.66 Numerical simulation software may be used to calculate the flow domain for two-dimensional axisymmetric flow through an orifice (Problem 50).

The flow is steady, two-dimensional planar flow of an incompressible Newtonian fluid (water may be used). Calculate the flow field and the force on the stationary walls as a function of a Reynolds number based on wall velocity and cavity depth. Produce appropriate plots to demonstrate the characteristics of the flow.

52. *Flow Problem: Two-dimensional planar gradual contraction near wall, numerical solution.* Flow enters a channel that gradually contracts as shown in Figure 7.68. The flow is steady, two-dimensional flow of an incompressible

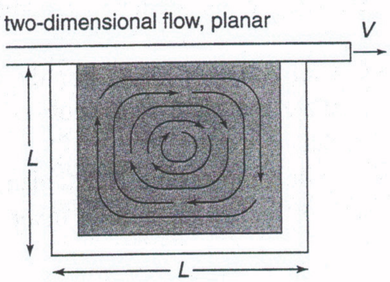


Figure 7.67 Numerical simulation software may be used to calculate the flow domain for two-dimensional planar cavity flow (Problem 51).

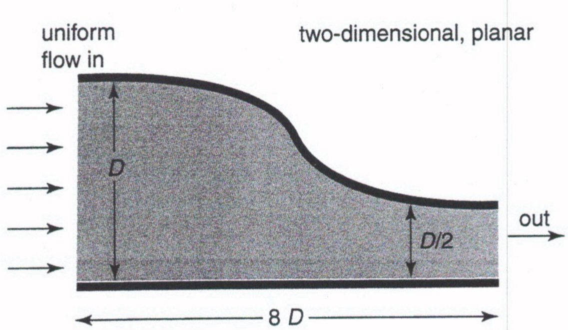


Figure 7.68 Numerical simulation software may be used to calculate the flow domain for two-dimensional planar gradual contraction near the wall (Problem 52).

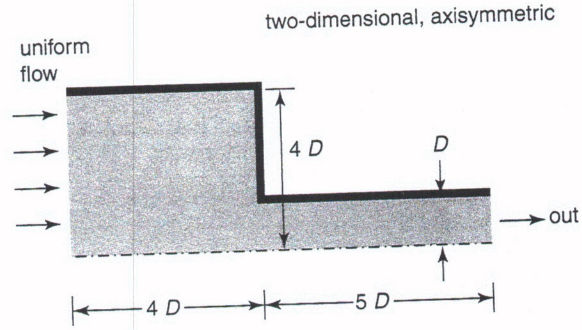


Figure 7.69 Numerical simulation software may be used to calculate the flow domain for two-dimensional axisymmetric 4:1 contraction (Problem 53).

Newtonian fluid (water may be used). Calculate the flow field and the force on the two walls as a function of the inlet Reynolds number. Produce appropriate plots to demonstrate characteristics of the flow.

53. *Flow Problem: Two-dimensional axisymmetric 4:1 contraction, numerical solution.* Flow enters 4:1 axial contraction as shown in Figure 7.69. The flow is steady, two-dimensional, axisymmetric flow of an incompressible Newtonian fluid (water may be used). Calculate the flow field and the force on the wall as a function of the outlet Reynolds number. Produce appropriate plots to demonstrate characteristics of the flow.
54. *Flow Problem: Flow in an obstructed channel, numerical.* For the obstructed flow shown in Figure 7.70, calculate the flow field with a numerical problem solver. What is the velocity field?
55. *Flow Problem: Squeeze flow with constant force.* For the same flow as described in Problem 44, calculate the plate separation as a function of time if the applied force is constant.
56. *Flow Problem: Helical flow.* An incompressible Newtonian fluid fills the annular gap between a cylinder of radius $\kappa R$ and an outer shell of inner

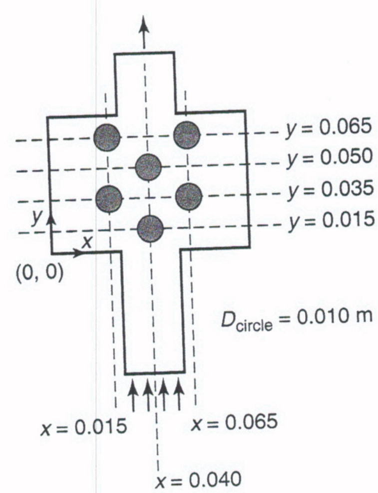


Figure 7.70 Numerical problem-solving software may be used for complex flow geometries shown here (Problem 54).

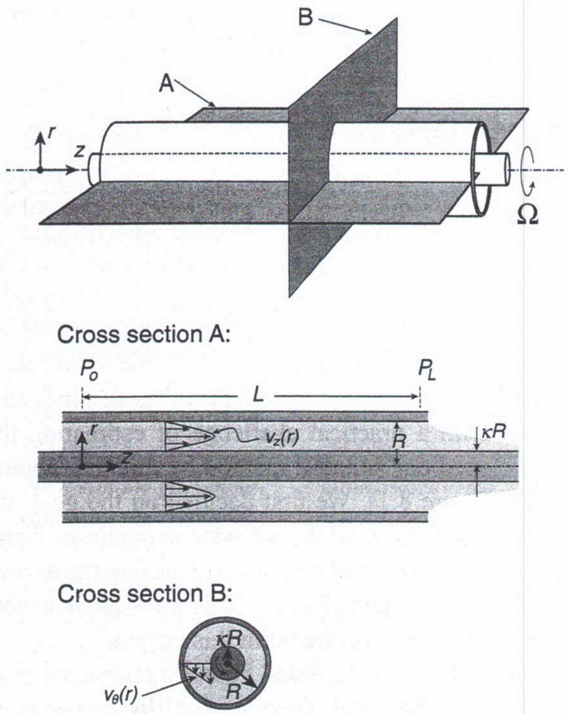


Figure 7.71 Helical flow of a Newtonian fluid (Problem 56).

radius $R$ (Figure 7.71). The inner cylinder turns counter clockwise at an angular velocity $\Omega$ radians/s. In addition, the inner cylinder is pulled to the right at a velocity V. The combined effect of these two motions produces a helical flow. The flow may be assumed to be symmetrical in the azimuthal direction (i.e., no $\theta$ variation). The axial pressure gradient is constant and denoted $\lambda$, and the pressure at the inner cylinder is $P_{\kappa R}$. Calculate the steady state velocity profile, the radial pressure distribution, and the torque needed to turn the inner cylinder.