and (*d*) the rate of heat transfer from the entire finned surface of the plate.

5–36 A hot surface at 100°C is to be cooled by attaching 3-cm-long, 0.25-cm-diameter aluminum pin fins (k = 237 W/m · °C) with a center-to-center distance of 0.6 cm. The temperature of the surrounding medium is 30°C, and the combined heat transfer coefficient on the surfaces is 35 W/m² · °C. Assuming steady one-dimensional heat transfer along the fin and taking the nodal spacing to be 0.5 cm, determine (*a*) the finite difference formulation of this problem, (*b*) the nodal temperatures along the fin by solving these equations, (*c*) the rate of heat transfer from a single fin, and (*d*) the rate of heat transfer from a 1-m × 1-m section of the plate.

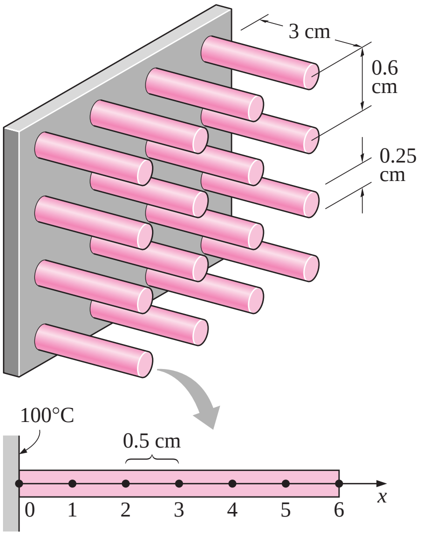


FIGURE P5–36

5–37 Repeat Problem 5–36 using copper fins (k = 386 W/m · °C) instead of aluminum ones.

Answers: (*b*) 98.6°C, 97.5°C, 96.7°C, 96.0°C, 95.7°C, 95.5°C

5–38 Two 3-m-long and 0.4-cm-thick cast iron (k = 52 W/m · °C, ε = 0.8) steam pipes of outer diameter 10 cm are connected to each other through two 1-cm-thick flanges of outer diameter 20 cm, as shown in the figure. The steam flows inside the pipe at an average temperature of 200°C with a heat transfer coefficient of 180 W/m² · °C. The outer surface of the pipe is exposed to convection with ambient air at 8°C with a heat transfer coefficient of 25 W/m² · °C as well as radiation with the surrounding surfaces at an average temperature of T_{surr} = 290 K. Assuming steady one-dimensional heat conduction along the flanges and taking the nodal spacing to be 1 cm along the flange (*a*) obtain the finite difference formulation for all nodes, (*b*) determine the temperature at the tip of the flange by solving those equations, and (*c*) determine the rate of heat transfer from the exposed surfaces of the flange.

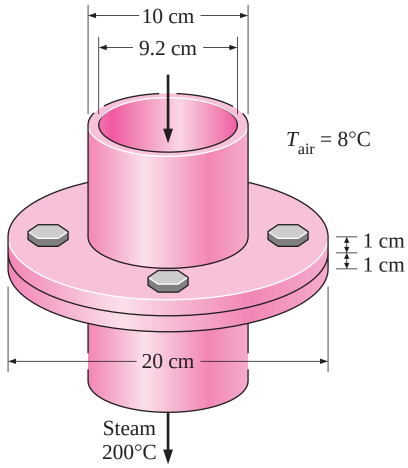


FIGURE P5–38

5–39 Reconsider Problem 5–38. Using EES (or other) software, investigate the effects of the steam temperature and the outer heat transfer coefficient on the flange tip temperature and the rate of heat transfer from the exposed surfaces of the flange. Let the steam temperature vary from 150°C to 300°C and the heat transfer coefficient from 15 W/m² · °C to 60 W/m² · °C. Plot the flange tip temperature and the heat transfer rate as functions of steam temperature and heat transfer coefficient, and discuss the results.

5–40 Using EES (or other) software, solve these systems of algebraic equations.

(*a*)
$$
\begin{aligned}
3x_1 - x_2 + 3x_3 &= 0 \\
-x_1 + 2x_2 + x_3 &= 3 \\
2x_1 - x_2 - x_3 &= 2
\end{aligned}
$$

(*b*)
$$
\begin{aligned}
4x_1 - 2x_2^2 + 0.5x_3 &= -2 \\
x_1^3 - x_2 + x_3 &= 11.964 \\
x_1 + x_2 + x_3 &= 3
\end{aligned}
$$

Answers: (*a*) $x_1 = 2$, $x_2 = 3$, $x_3 = -1$, (*b*) $x_1 = 2.33$, $x_2 = 2.29$, $x_3 = -1.62$

5–41 Using EES (or other) software, solve these systems of algebraic equations.

(*a*)
$$
\begin{aligned}
3x_1 + 2x_2 - x_3 + x_4 &= 6 \\
x_1 + 2x_2 - x_4 &= -3 \\
-2x_1 + x_2 + 3x_3 + x_4 &= 2 \\
3x_2 + x_3 - 4x_4 &= -6
\end{aligned}
$$

(*b*)
$$
\begin{aligned}
3x_1 + x_2^2 + 2x_3 &= 8 \\
-x_1^2 + 3x_2 + 2x_3 &= -6.293 \\
2x_1 - x_2^4 + 4x_3 &= -12
\end{aligned}
$$

5–42 Using EES (or other) software, solve these systems of algebraic equations.

(*a*)
$$4x_1 - x_2 + 2x_3 + x_4 = -6$$
$$x_1 + 3x_2 - x_3 + 4x_4 = -1$$
$$-x_1 + 2x_2 + 5x_4 = 5$$
$$2x_2 - 4x_3 - 3x_4 = -5$$

(*b*)
$$2x_1 + x_2^4 - 2x_3 + x_4 = 1$$
$$x_1^2 + 4x_2 + 2x_3^2 - 2x_4 = -3$$
$$-x_1 + x_2^4 + 5x_3 = 10$$
$$3x_1 - x_3^2 + 8x_4 = 15$$

Two-Dimensional Steady Heat Conduction

5–43C Consider a medium in which the finite difference formulation of a general interior node is given in its simplest form as

$$T_{\text{left}} + T_{\text{top}} + T_{\text{right}} + T_{\text{bottom}} - 4T_{\text{node}} + \frac{\dot{g}_{\text{node}} l^2}{k} = 0$$

(*a*) Is heat transfer in this medium steady or transient?
(*b*) Is heat transfer one-, two-, or three-dimensional?
(*c*) Is there heat generation in the medium?
(*d*) Is the nodal spacing constant or variable?
(*e*) Is the thermal conductivity of the medium constant or variable?

5–44C Consider a medium in which the finite difference formulation of a general interior node is given in its simplest form as

$$T_{\text{node}} = (T_{\text{left}} + T_{\text{top}} + T_{\text{right}} + T_{\text{bottom}})/4$$

(*a*) Is heat transfer in this medium steady or transient?
(*b*) Is heat transfer one-, two-, or three-dimensional?
(*c*) Is there heat generation in the medium?
(*d*) Is the nodal spacing constant or variable?
(*e*) Is the thermal conductivity of the medium constant or variable?

5–45C What is an irregular boundary? What is a practical way of handling irregular boundary surfaces with the finite difference method?

5–46 Consider steady two-dimensional heat transfer in a long solid body whose cross section is given in the figure. The temperatures at the selected nodes and the thermal conditions at the boundaries are as shown. The thermal conductivity of the body is $k = 45$ W/m · °C, and heat is generated in the body uniformly at a rate of $\dot{g} = 6 \times 10^6$ W/m^3. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 5.0$ cm, determine (*a*) the temperatures at nodes 1, 2, and 3 and (*b*) the rate of heat loss from the bottom surface through a 1-m-long section of the body.

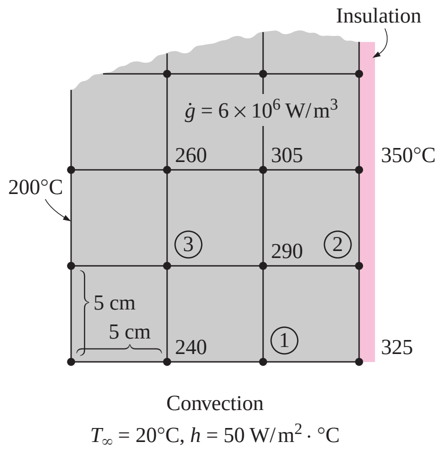


FIGURE P5–46

5–47 Consider steady two-dimensional heat transfer in a long solid body whose cross section is given in the figure. The measured temperatures at selected points of the outer surfaces are as shown. The thermal conductivity of the body is $k = 45$ W/m · °C, and there is no heat generation. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 2.0$ cm, determine the temperatures at the indicated points in the medium. *Hint:* Take advantage of symmetry.

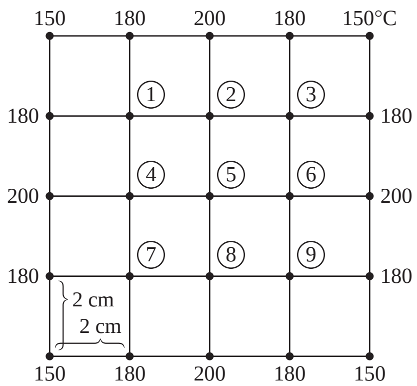


FIGURE P5–47

5–48 Consider steady two-dimensional heat transfer in a long solid bar whose cross section is given in the figure. The measured temperatures at selected points of the outer surfaces are as shown. The thermal conductivity of the body is $k = 20$ W/m · °C, and there is no heat generation. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 1.0$ cm, determine the temperatures at the indicated points in the medium.
Answers: $T_1 = 185$°C, $T_2 = T_3 = T_4 = 190$°C

5–49 Starting with an energy balance on a volume element, obtain the steady two-dimensional finite difference equation for a general interior node in rectangular coordinates for $T(x, y)$ for the case of variable thermal conductivity and uniform heat generation.

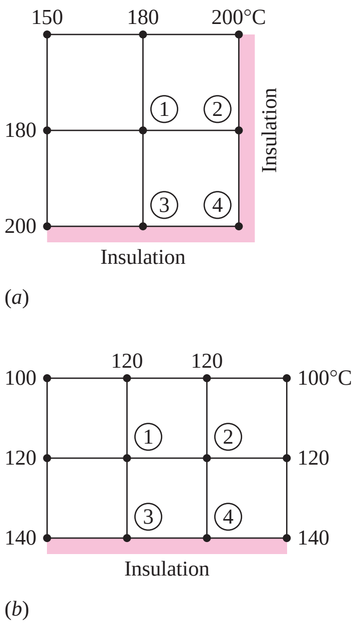


FIGURE P5–48

5–50 Consider steady two-dimensional heat transfer in a long solid body whose cross section is given in the figure. The temperatures at the selected nodes and the thermal conditions on the boundaries are as shown. The thermal conductivity of the body is $k = 180$ W/m · °C, and heat is generated in the body uniformly at a rate of $\dot{g} = 10^7$ W/m^3. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 10$ cm, determine (*a*) the temperatures at nodes 1, 2, 3, and 4 and (*b*) the rate of heat loss from the top surface through a 1-m-long section of the body.

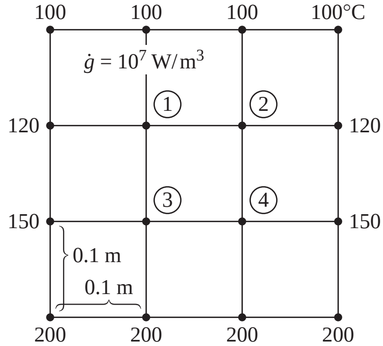


FIGURE P5–50

5–51 Reconsider Problem 5–50. Using EES (or other) software, investigate the effects of the thermal conductivity and the heat generation rate on the temperatures at nodes 1 and 3, and the rate of heat loss from the top surface. Let the thermal conductivity vary from 10 W/m · °C to 400 W/m · °C and the heat generation rate from 10^5 W/m^3 to 10^8 W/m^3. Plot the temperatures at nodes 1 and 3, and the rate of heat loss as functions of thermal conductivity and heat generation rate, and discuss the results.

5–52 Consider steady two-dimensional heat transfer in a long solid bar whose cross section is given in the figure. The measured temperatures at selected points on the outer surfaces are as shown. The thermal conductivity of the body is $k = 20$ W/m · °C, and there is no heat generation. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 1.0$ cm, determine the temperatures at the indicated points in the medium. *Hint:* Take advantage of symmetry.

Answers: (*b*) $T_1 = T_4 = 143$°C, $T_2 = T_3 = 136$°C

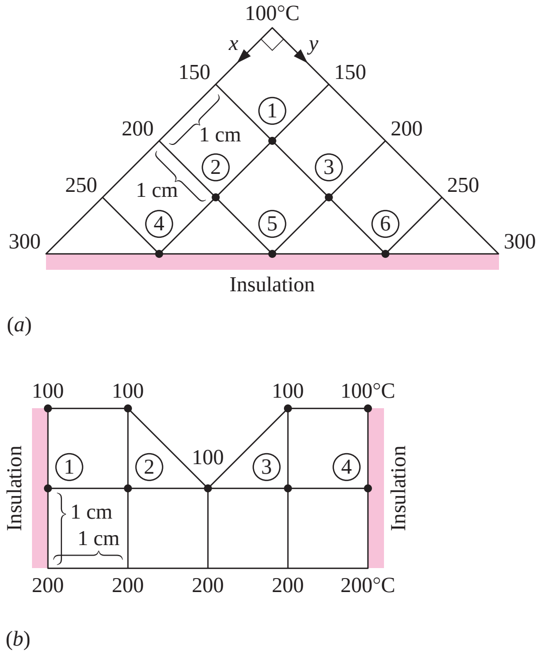


FIGURE P5–52

5–53 Consider steady two-dimensional heat transfer in an L-shaped solid body whose cross section is given in the figure. The thermal conductivity of the body is $k = 45$ W/m · °C, and heat is generated in the body at a rate of $\dot{g} = 5 \times 10^6$ W/m^3.

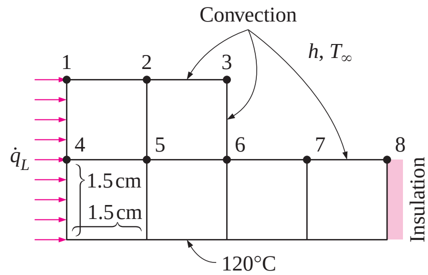


FIGURE P5–53

The right surface of the body is insulated, and the bottom surface is maintained at a uniform temperature of 120°C. The entire top surface is subjected to convection with ambient air at $T_\infty = 30°C$ with a heat transfer coefficient of $h = 55$ W/m$^2 \cdot$ °C, and the left surface is subjected to heat flux at a uniform rate of $\dot{q}_L = 8000$ W/m^2. The nodal network of the problem consists of 13 equally spaced nodes with $\Delta x = \Delta y = 1.5$ cm. Five of the nodes are at the bottom surface and thus their temperatures are known. (*a*) Obtain the finite difference equations at the remaining eight nodes and (*b*) determine the nodal temperatures by solving those equations.

5–54E Consider steady two-dimensional heat transfer in a long solid bar of square cross section in which heat is generated uniformly at a rate of $\dot{g} = 0.19 \times 10^5$ Btu/h $\cdot$ ft^3. The cross section of the bar is 0.4 ft $\times$ 0.4 ft in size, and its thermal conductivity is $k = 16$ Btu/h $\cdot$ ft $\cdot$ °F. All four sides of the bar are subjected to convection with the ambient air at $T_\infty = 70°F$ with a heat transfer coefficient of $h = 7.9$ Btu/h $\cdot$ ft$^2 \cdot$ °F. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 0.2$ ft, determine (*a*) the temperatures at the nine nodes and (*b*) the rate of heat loss from the bar through a 1-ft-long section.

Answer: (*b*) 3040 Btu/h

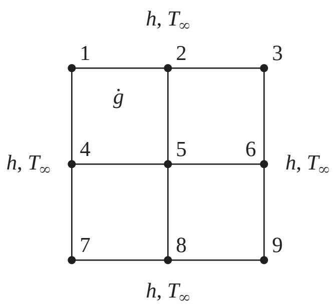


FIGURE P5–54E

5–55 Hot combustion gases of a furnace are flowing through a concrete chimney ($k = 1.4$ W/m $\cdot$ °C) of rectangular cross section. The flow section of the chimney is 20 cm $\times$ 40 cm, and the thickness of the wall is 10 cm. The average temperature of the hot gases in the chimney is $T_i = 280°C$, and the average convection heat transfer coefficient inside the chimney is $h_i = 75$ W/m$^2 \cdot$ °C. The chimney is losing heat from its outer surface to the ambient air at $T_o = 15°C$ by convection with a heat transfer coefficient of $h_o = 18$ W/m$^2 \cdot$ °C and to the sky by radiation. The emissivity of the outer surface of the wall is $\varepsilon = 0.9$, and the effective sky temperature is estimated to be 250 K. Using the finite difference method with $\Delta x = \Delta y = 10$ cm and taking full advantage of symmetry, (*a*) obtain the finite difference formulation of this problem for steady two-dimensional heat transfer, (*b*) determine the temperatures at the nodal points of a cross section, and (*c*) evaluate the rate of heat loss for a 1-m-long section of the chimney.

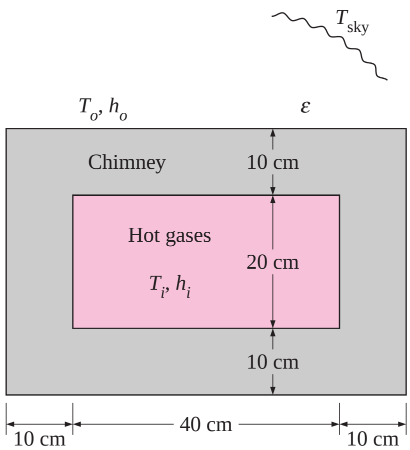


FIGURE P5–55

5–56 Repeat Problem 5–55 by disregarding radiation heat transfer from the outer surfaces of the chimney.

5–57 Reconsider Problem 5–55. Using EES (or other) software, investigate the effects of hot-gas temperature and the outer surface emissivity on the temperatures at the outer corner of the wall and the middle of the inner surface of the right wall, and the rate of heat loss. Let the temperature of the hot gases vary from 200°C to 400°C and the emissivity from 0.1 to 1.0. Plot the temperatures and the rate of heat loss as functions of the temperature of the hot gases and the emissivity, and discuss the results.

5–58 Consider a long concrete dam ($k = 0.6$ W/m $\cdot$ °C, $\alpha_s = 0.7$ m^2/s) of triangular cross section whose exposed surface is subjected to solar heat flux of $\dot{q}_s = 800$ W/m^2 and to convection and radiation to the environment at 25°C with a combined heat transfer coefficient of 30 W/m$^2 \cdot$ °C. The 2-m-high vertical section of the dam is subjected to convection by water at 15°C with a heat transfer coefficient of 150 W/m$^2 \cdot$ °C, and heat transfer through the 2-m-long base is considered to be negligible. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 1$ m and assuming steady two-dimensional heat transfer, determine the temperature of the top, middle, and bottom of the exposed surface of the dam. *Answers:* 21.3°C, 43.2°C, 43.6°C

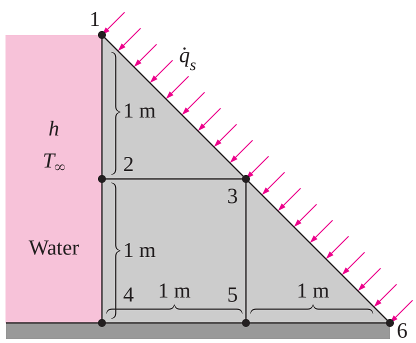


FIGURE P5–58

5–59E Consider steady two-dimensional heat transfer in a V-grooved solid body whose cross section is given in the figure. The top surfaces of the groove are maintained at 32°F while the bottom surface is maintained at 212°F. The side surfaces of the groove are insulated. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 1$ ft and taking advantage of symmetry, determine the temperatures at the middle of the insulated surfaces.

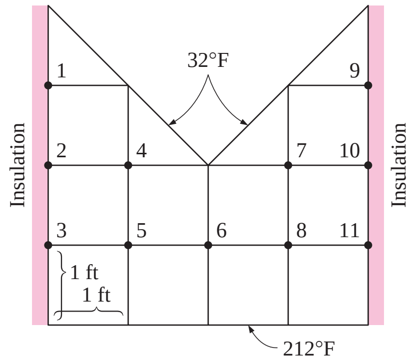


FIGURE P5–59E

5–60 Reconsider Problem 5–59E. Using EES (or other) software, investigate the effects of the temperatures at the top and bottom surfaces on the temperature in the middle of the insulated surface. Let the temperatures at the top and bottom surfaces vary from 32°F to 212°F. Plot the temperature in the middle of the insulated surface as functions of the temperatures at the top and bottom surfaces, and discuss the results.

5–61 Consider a long solid bar whose thermal conductivity is $k = 12$ W/m · °C and whose cross section is given in the figure. The top surface of the bar is maintained at 50°C while the bottom surface is maintained at 120°C. The left surface is insulated and the remaining three surfaces are subjected to convection with ambient air at $T_\infty = 25$°C with a heat transfer coefficient of $h = 30$ W/m^2 · °C. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 10$ cm, (*a*) obtain the finite difference formulation of this problem for steady two-dimensional heat transfer and (*b*) determine the unknown nodal temperatures by solving those equations.

Answers: (*b*) 85.7°C, 86.4°C, 87.6°C

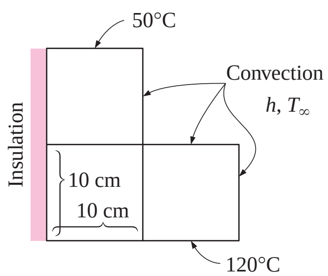


FIGURE P5–61

5–62 Consider a 5-m-long constantan block ($k = 23$ W/m · °C) 30 cm high and 50 cm wide. The block is completely submerged in iced water at 0°C that is well stirred, and the heat transfer coefficient is so high that the temperatures on both sides of the block can be taken to be 0°C. The bottom surface of the bar is covered with a low-conductivity material so that heat transfer through the bottom surface is negligible. The top surface of the block is heated uniformly by a 6-kW resistance heater. Using the finite difference method with a mesh size of $\Delta x = \Delta y = 10$ cm and taking advantage of symmetry, (*a*) obtain the finite difference formulation of this problem for steady two-dimensional heat transfer, (*b*) determine the unknown nodal temperatures by solving those equations, and (*c*) determine the rate of heat transfer from the block to the iced water.

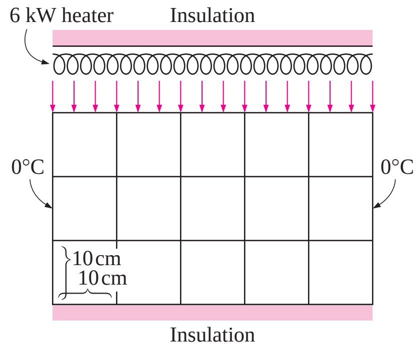


FIGURE P5–62

Transient Heat Conduction

5–63C How does the finite difference formulation of a transient heat conduction problem differ from that of a steady heat conduction problem? What does the term $\rho A \Delta x C(T_m^{i+1} - T_m^i)/\Delta t$ represent in the transient finite difference formulation?

5–64C What are the two basic methods of solution of transient problems based on finite differencing? How do heat transfer terms in the energy balance formulation differ in the two methods?

5–65C The explicit finite difference formulation of a general interior node for transient heat conduction in a plane wall is given by

$$T_{m-1}^i - 2T_m^i + T_{m+1}^i + \frac{\dot{g}_m^i \Delta x^2}{k} = \frac{T_m^{i+1} - T_m^i}{\tau}$$

Obtain the finite difference formulation for the steady case by simplifying the relation above.

5–66C The explicit finite difference formulation of a general interior node for transient two-dimensional heat conduction is given by

$$T_{\text{node}}^{i+1} = \tau(T_{\text{left}}^{i} + T_{\text{top}}^{i} + T_{\text{right}}^{i} + T_{\text{bottom}}^{i}) + (1 - 4\tau)T_{\text{node}}^{i} + \tau \frac{\dot{g}_{\text{node}}^{i} l^2}{k}$$

Obtain the finite difference formulation for the steady case by simplifying the relation above.

5–67C Is there any limitation on the size of the time step Δt in the solution of transient heat conduction problems using (*a*) the explicit method and (*b*) the implicit method?

5–68C Express the general stability criterion for the explicit method of solution of transient heat conduction problems.

5–69C Consider transient one-dimensional heat conduction in a plane wall that is to be solved by the explicit method. If both sides of the wall are at specified temperatures, express the stability criterion for this problem in its simplest form.

5–70C Consider transient one-dimensional heat conduction in a plane wall that is to be solved by the explicit method. If both sides of the wall are subjected to specified heat flux, express the stability criterion for this problem in its simplest form.

5–71C Consider transient two-dimensional heat conduction in a rectangular region that is to be solved by the explicit method. If all boundaries of the region are either insulated or at specified temperatures, express the stability criterion for this problem in its simplest form.

5–72C The implicit method is unconditionally stable and thus any value of time step Δt can be used in the solution of transient heat conduction problems. To minimize the computation time, someone suggests using a very large value of Δt since there is no danger of instability. Do you agree with this suggestion? Explain.

5–73 Consider transient heat conduction in a plane wall whose left surface (node 0) is maintained at 50°C while the right surface (node 6) is subjected to a solar heat flux of 600 W/m^2. The wall is initially at a uniform temperature of 50°C. Express the explicit finite difference formulation of the boundary nodes 0 and 6 for the case of no heat generation. Also, obtain the finite difference formulation for the total amount of heat transfer at the left boundary during the first three time steps.

5–74 Consider transient heat conduction in a plane wall with variable heat generation and constant thermal conductivity. The nodal network of the medium consists of nodes 0, 1, 2, 3, and 4 with a uniform nodal spacing of Δx. The wall is initially at a specified temperature. Using the energy balance approach, obtain the explicit finite difference formulation of the boundary nodes for the case of uniform heat flux $\dot{q}_0$ at the left boundary (node 0) and convection at the right boundary (node 4) with a convection coefficient of h and an ambient temperature of T_∞. Do not simplify.

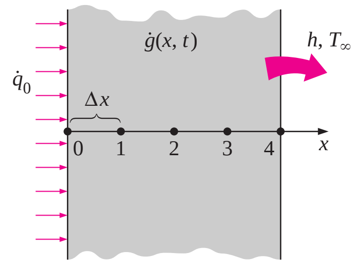


FIGURE P5–74

5–75 Repeat Problem 5–74 for the case of implicit formulation.

5–76 Consider transient heat conduction in a plane wall with variable heat generation and constant thermal conductivity. The nodal network of the medium consists of nodes 0, 1, 2, 3, 4, and 5 with a uniform nodal spacing of Δx. The wall is initially at a specified temperature. Using the energy balance approach, obtain the explicit finite difference formulation of the boundary nodes for the case of insulation at the left boundary (node 0) and radiation at the right boundary (node 5) with an emissivity of ε and surrounding temperature of T_{surr}.

5–77 Consider transient heat conduction in a plane wall with variable heat generation and constant thermal conductivity. The nodal network of the medium consists of nodes 0, 1, 2, 3, and 4 with a uniform nodal spacing of Δx. The wall is initially at a specified temperature. The temperature at the right boundary (node 4) is specified. Using the energy balance approach, obtain the explicit finite difference formulation of the boundary node 0 for the case of combined convection, radiation, and heat flux at the left boundary with an emissivity of ε, convection coefficient of h, ambient temperature of T_∞, surrounding temperature of T_{surr}, and uniform heat flux of $\dot{q}_0$ toward the wall. Also, obtain the finite difference formulation for the total amount of heat transfer at the right boundary for the first 20 time steps.

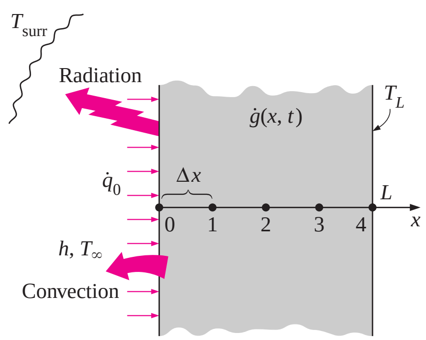


FIGURE P5–77

5–78 Starting with an energy balance on a volume element, obtain the two-dimensional transient explicit finite difference equation for a general interior node in rectangular coordinates for $T(x, y, t)$ for the case of constant thermal conductivity and no heat generation.

5–79 Starting with an energy balance on a volume element, obtain the two-dimensional transient implicit finite difference equation for a general interior node in rectangular coordinates for $T(x, y, t)$ for the case of constant thermal conductivity and no heat generation.

5–80 Starting with an energy balance on a disk volume element, derive the one-dimensional transient explicit finite difference equation for a general interior node for $T(z, t)$ in a cylinder whose side surface is insulated for the case of constant thermal conductivity with uniform heat generation.

5–81 Consider one-dimensional transient heat conduction in a composite plane wall that consists of two layers A and B with perfect contact at the interface. The wall involves no heat generation and initially is at a specified temperature. The nodal network of the medium consists of nodes 0, 1 (at the interface), and 2 with a uniform nodal spacing of Δx. Using the energy balance approach, obtain the explicit finite difference formulation of this problem for the case of insulation at the left boundary (node 0) and radiation at the right boundary (node 2) with an emissivity of ε and surrounding temperature of T_{surr}.

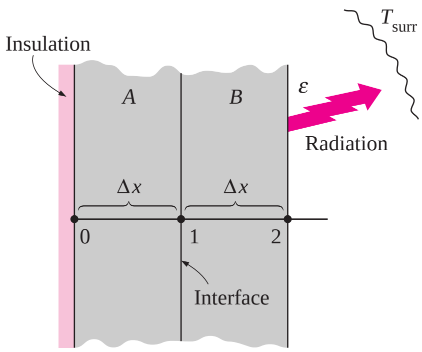


FIGURE P5–81

5–82 Consider transient one-dimensional heat conduction in a pin fin of constant diameter D with constant thermal conductivity. The fin is losing heat by convection to the ambient air at T_∞ with a heat transfer coefficient of h and by radiation to the surrounding surfaces at an average temperature of T_{surr}. The nodal network of the fin consists of nodes 0 (at the base), 1 (in the middle), and 2 (at the fin tip) with a uniform nodal spacing of Δx. Using the energy balance approach, obtain the explicit finite difference formulation of this problem for the case of a specified temperature at the fin base and negligible heat transfer at the fin tip.

5–83 Repeat Problem 5–82 for the case of implicit formulation.

5–84 Consider a large uranium plate of thickness $L = 8$ cm, thermal conductivity $k = 28$ W/m · °C, and thermal diffusivity $\alpha = 12.5 \times 10^{-6}$ m^2/s that is initially at a uniform temperature of 100°C. Heat is generated uniformly in the plate at a constant rate of $\dot{g} = 10^6$ W/m^3. At time $t = 0$, the left side of the plate is insulated while the other side is subjected to convection with an environment at $T_\infty = 20$°C with a heat transfer coefficient of $h = 35$ W/m^2 · °C. Using the explicit finite difference approach with a uniform nodal spacing of $\Delta x = 2$ cm, determine (*a*) the temperature distribution in the plate after 5 min and (*b*) how long it will take for steady conditions to be reached in the plate.

5–85 Reconsider Problem 5–84. Using EES (or other) software, investigate the effect of the cooling time on the temperatures of the left and right sides of the plate. Let the time vary from 5 min to 60 min. Plot the temperatures at the left and right surfaces as a function of time, and discuss the results.

5–86 Consider a house whose south wall consists of a 30-cm-thick Trombe wall whose thermal conductivity is $k = 0.70$ W/m · °C and whose thermal diffusivity is $\alpha = 0.44 \times 10^{-6}$ m^2/s. The variations of the ambient temperature T_{out} and the solar heat flux $\dot{q}_{solar}$ incident on a south-facing vertical surface throughout the day for a typical day in February are given in the table in 3-h intervals. The Trombe wall has single glazing with an absorptivity-transmissivity product of $\kappa = 0.76$ (that is, 76 percent of the solar energy incident is absorbed by the exposed surface of the Trombe wall), and the average combined heat transfer coefficient for heat loss from the Trombe wall to the ambient is determined to be $h_{out} = 3.4$ W/m^2 · °C. The interior of the house is maintained at $T_{in} = 20$°C at all times, and the heat transfer coefficient at the interior surface of the Trombe wall is $h_{in} = 9.1$ W/m^2 · °C. Also, the vents on the Trombe wall are kept closed, and thus the only heat transfer between the air in the house and the Trombe wall is through the

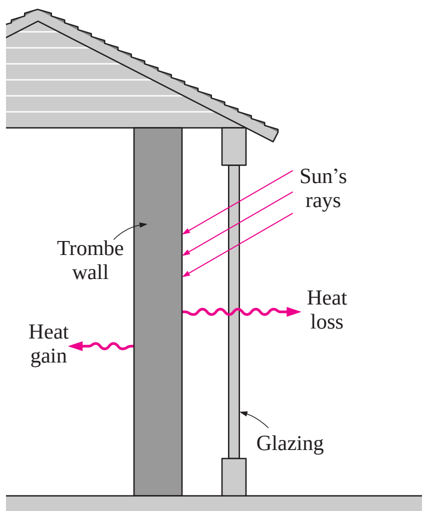


FIGURE P5–86

TABLE P5–86

The hourly variations of the monthly average ambient temperature and solar heat flux incident on a vertical surface

Time of Day	Ambient Temperature, °C	Solar Insolation, W/m²
7 AM–10 AM	0	375
10 AM–1 PM	4	750
1 PM–4 PM	6	580
4 PM–7 PM	1	95
7 PM–10 PM	−2	0
10 PM–1 AM	−3	0
1 AM–4 AM	−4	0
4 AM–7 AM	4	0

interior surface of the wall. Assuming the temperature of the Trombe wall to vary linearly between 20°C at the interior surface and 0°C at the exterior surface at 7 AM and using the explicit finite difference method with a uniform nodal spacing of Δx = 5 cm, determine the temperature distribution along the thickness of the Trombe wall after 6, 12, 18, 24, 30, 36, 42, and 48 hours and plot the results. Also, determine the net amount of heat transferred to the house from the Trombe wall during the first day if the wall is 2.8 m high and 7 m long.

5–87 Consider two-dimensional transient heat transfer in an L-shaped solid bar that is initially at a uniform temperature of 140°C and whose cross section is given in the figure. The thermal conductivity and diffusivity of the body are k = 15 W/m · °C and $\alpha = 3.2 \times 10^{-6}$ m²/s, respectively, and heat is generated in the body at a rate of $\dot{g} = 2 \times 10^7$ W/m³. The right surface of the body is insulated, and the bottom surface is maintained at a uniform temperature of 140°C at all times. At time t = 0, the entire top surface is subjected to convection with ambient air at T_∞ = 25°C with a heat transfer coefficient of h = 80 W/m² · °C, and the left surface is subjected to uniform heat flux at a rate of $\dot{q}_L$ = 8000 W/m². The nodal network of the problem consists of 13 equally spaced nodes with $\Delta x = \Delta y$ = 1.5 cm. Using the explicit method, determine the temperature at the top corner (node 3) of the body after 2, 5, and 30 min.

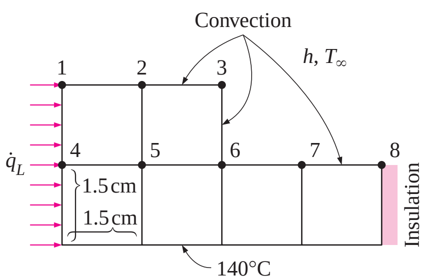


FIGURE P5–87

5–88 Reconsider Problem 5–87. Using EES (or other) software, plot the temperature at the top corner as a function of heating time varies from 2 min to 30 min, and discuss the results.

5–89 Consider a long solid bar (k = 28 W/m · °C and $\alpha = 12 \times 10^{-6}$ m²/s) of square cross section that is initially at a uniform temperature of 20°C. The cross section of the bar is 20 cm × 20 cm in size, and heat is generated in it uniformly at a rate of $\dot{g} = 8 \times 10^5$ W/m³. All four sides of the bar are subjected to convection to the ambient air at T_∞ = 30°C with a heat transfer coefficient of h = 45 W/m² · °C. Using the explicit finite difference method with a mesh size of $\Delta x = \Delta y$ = 10 cm, determine the centerline temperature of the bar (*a*) after 10 min and (*b*) after steady conditions are established.

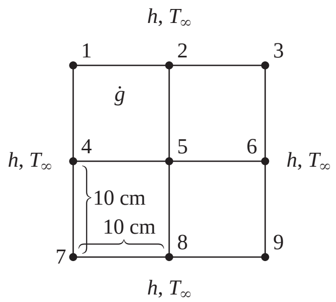


FIGURE P5–89

5–90E Consider a house whose windows are made of 0.375-in.-thick glass (k = 0.48 Btu/h · ft · °F and $\alpha = 4.2 \times 10^{-6}$ ft²/s). Initially, the entire house, including the walls and the windows, is at the outdoor temperature of T_o = 35°F. It is observed that the windows are fogged because the indoor temperature is below the dew-point temperature of 54°F. Now the heater is turned on and the air temperature in the house is raised to T_i = 72°F at a rate of 2°F rise per minute. The heat transfer coefficients at the inner and outer surfaces of the wall can be taken to be h_i = 1.2 and h_o = 2.6 Btu/h · ft² · °F, respectively, and the outdoor temperature can be assumed to remain constant. Using the explicit finite difference method with a mesh size of Δx = 0.125 in., determine how long it will take

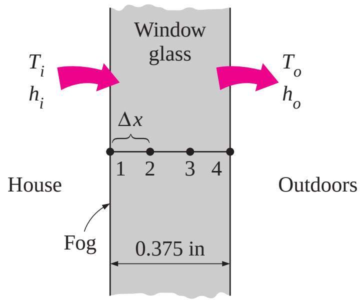


FIGURE P5–90E

for the fog on the windows to clear up (i.e., for the inner surface temperature of the window glass to reach 54°F).

5–91 A common annoyance in cars in winter months is the formation of fog on the glass surfaces that blocks the view. A practical way of solving this problem is to blow hot air or to attach electric resistance heaters to the inner surfaces. Consider the rear window of a car that consists of a 0.4-cm-thick glass (k = 0.84 W/m · °C and $\alpha = 0.39 \times 10^{-6}$ m²/s). Strip heater wires of negligible thickness are attached to the inner surface of the glass, 4 cm apart. Each wire generates heat at a rate of 10 W/m length. Initially the entire car, including its windows, is at the outdoor temperature of $T_o = -3$°C. The heat transfer coefficients at the inner and outer surfaces of the glass can be taken to be h_i = 6 and h_o = 20 W/m² · °C, respectively. Using the explicit finite difference method with a mesh size of Δx = 0.2 cm along the thickness and Δy = 1 cm in the direction normal to the heater wires, determine the temperature distribution throughout the glass 15 min after the strip heaters are turned on. Also, determine the temperature distribution when steady conditions are reached.

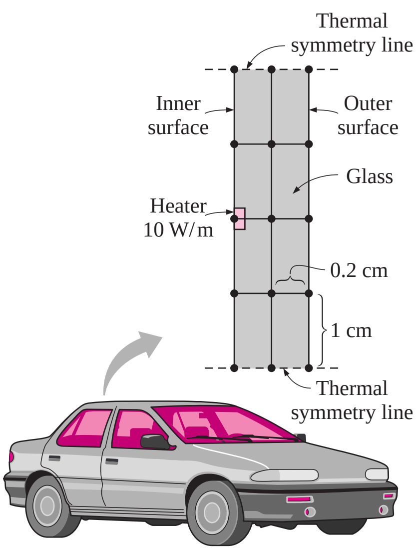


FIGURE P5–91

5–92 Repeat Problem 5–91 using the implicit method with a time step of 1 min.

5–93 The roof of a house consists of a 15-cm-thick concrete slab (k = 1.4 W/m · °C and $\alpha = 0.69 \times 10^{-6}$ m²/s) that is 20 m wide and 20 m long. One evening at 6 PM, the slab is observed to be at a uniform temperature of 18°C. The average ambient air and the night sky temperatures for the entire night are predicted to be 6°C and 260 K, respectively. The convection heat transfer coefficients at the inner and outer surfaces of the roof can be taken to be h_i = 5 and h_o = 12 W/m² · °C, respectively. The house and the interior surfaces of the walls and the floor

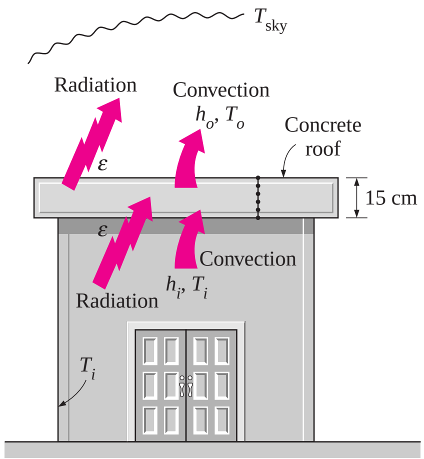


FIGURE P5–93

are maintained at a constant temperature of 20°C during the night, and the emissivity of both surfaces of the concrete roof is 0.9. Considering both radiation and convection heat transfers and using the explicit finite difference method with a time step of Δt = 5 min and a mesh size of Δx = 3 cm, determine the temperatures of the inner and outer surfaces of the roof at 6 AM. Also, determine the average rate of heat transfer through the roof during that night.

5–94 Consider a refrigerator whose outer dimensions are 1.80 m × 0.8 m × 0.7 m. The walls of the refrigerator are constructed of 3-cm-thick urethane insulation (k = 0.026 W/m · ° C and $\alpha = 0.36 \times 10^{-6}$ m²/s) sandwiched between two layers of sheet metal with negligible thickness. The refrigerated space is maintained at 3°C and the average heat transfer coefficients at the inner and outer surfaces of the wall are 6 W/m² · °C and 9 W/m² · °C, respectively. Heat transfer through the bottom surface of the refrigerator is negligible. The kitchen temperature remains constant at about 25°C. Initially, the refrigerator contains 15 kg of food items at an average specific heat of 3.6 kJ/kg · °C. Now a malfunction occurs and the refrigerator stops running for 6 h as a result. Assuming the

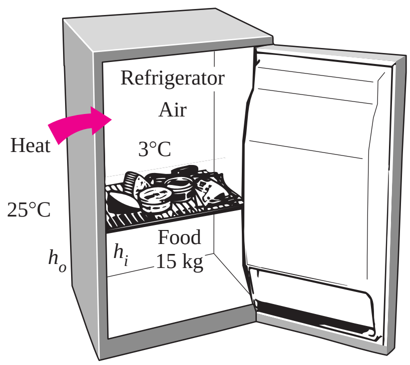


FIGURE P5–94

temperature of the contents of the refrigerator, including the air inside, rises uniformly during this period, predict the temperature inside the refrigerator after 6 h when the repairman arrives. Use the explicit finite difference method with a time step of $\Delta t = 1$ min and a mesh size of $\Delta x = 1$ cm and disregard corner effects (i.e., assume one-dimensional heat transfer in the walls).

5–95 Reconsider Problem 5–94. Using EES (or other) software, plot the temperature inside the refrigerator as a function of heating time as time varies from 1 h to 10 h, and discuss the results.

Special Topic: Controlling the Numerical Error

5–96C Why do the results obtained using a numerical method differ from the exact results obtained analytically? What are the causes of this difference?

5–97C What is the cause of the discretization error? How does the global discretization error differ from the local discretization error?

5–98C Can the global (accumulated) discretization error be less than the local error during a step? Explain.

5–99C How is the finite difference formulation for the first derivative related to the Taylor series expansion of the solution function?

5–100C Explain why the local discretization error of the finite difference method is proportional to the square of the step size. Also explain why the global discretization error is proportional to the step size itself.

5–101C What causes the round-off error? What kind of calculations are most susceptible to round-off error?

5–102C What happens to the discretization and the round-off errors as the step size is decreased?

5–103C Suggest some practical ways of reducing the round-off error.

5–104C What is a practical way of checking if the round-off error has been significant in calculations?

5–105C What is a practical way of checking if the discretization error has been significant in calculations?

Review Problems

5–106 Starting with an energy balance on the volume element, obtain the steady three-dimensional finite difference equation for a general interior node in rectangular coordinates for $T(x, y, z)$ for the case of constant thermal conductivity and uniform heat generation.

5–107 Starting with an energy balance on the volume element, obtain the three-dimensional transient explicit finite difference equation for a general interior node in rectangular coordinates for $T(x, y, z, t)$ for the case of constant thermal conductivity and no heat generation.

5–108 Consider steady one-dimensional heat conduction in a plane wall with variable heat generation and constant thermal conductivity. The nodal network of the medium consists of nodes 0, 1, 2, and 3 with a uniform nodal spacing of Δx. The temperature at the left boundary (node 0) is specified. Using the energy balance approach, obtain the finite difference formulation of boundary node 3 at the right boundary for the case of combined convection and radiation with an emissivity of ε, convection coefficient of h, ambient temperature of T_∞, and surrounding temperature of T_{surr}. Also, obtain the finite difference formulation for the rate of heat transfer at the left boundary.

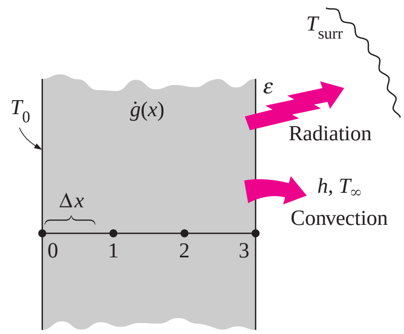


FIGURE P5–108

5–109 Consider one-dimensional transient heat conduction in a plane wall with variable heat generation and variable thermal conductivity. The nodal network of the medium consists of nodes 0, 1, and 2 with a uniform nodal spacing of Δx. Using the energy balance approach, obtain the explicit finite difference formulation of this problem for the case of specified heat flux $\dot{q}_0$ and convection at the left boundary (node 0) with a convection coefficient of h and ambient temperature of T_∞, and radiation at the right boundary (node 2) with an emissivity of ε and surrounding temperature of T_{surr}.

5–110 Repeat Problem 5–109 for the case of implicit formulation.

5–111 Consider steady one-dimensional heat conduction in a pin fin of constant diameter D with constant thermal conductivity. The fin is losing heat by convection with the ambient air

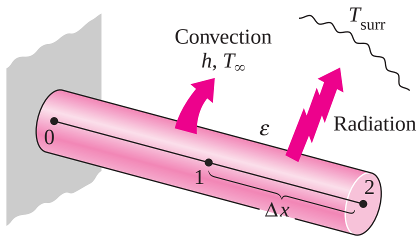


FIGURE P5–111

at T_∞ (in °C) with a convection coefficient of h, and by radiation to the surrounding surfaces at an average temperature of T_{surr} (in K). The nodal network of the fin consists of nodes 0 (at the base), 1 (in the middle), and 2 (at the fin tip) with a uniform nodal spacing of Δx. Using the energy balance approach, obtain the finite difference formulation of this problem for the case of a specified temperature at the fin base and convection and radiation heat transfer at the fin tip.

5–112 Starting with an energy balance on the volume element, obtain the two-dimensional transient explicit finite difference equation for a general interior node in rectangular coordinates for $T(x, y, t)$ for the case of constant thermal conductivity and uniform heat generation.

5–113 Starting with an energy balance on a disk volume element, derive the one-dimensional transient implicit finite difference equation for a general interior node for $T(z, t)$ in a cylinder whose side surface is subjected to convection with a convection coefficient of h and an ambient temperature of T_∞ for the case of constant thermal conductivity with uniform heat generation.

5–114E The roof of a house consists of a 5-in.-thick concrete slab ($k = 0.81$ Btu/h · ft · °F and $\alpha = 7.4 \times 10^{-6}$ ft²/s) that is 45 ft wide and 55 ft long. One evening at 6 PM, the slab is observed to be at a uniform temperature of 70°F. The ambient air temperature is predicted to be at about 50°F from 6 PM to 10 PM, 42°F from 10 PM to 2 AM, and 38°F from 2 AM to 6 AM, while the night sky temperature is expected to be about 445 R for the entire night. The convection heat transfer coefficients at the inner and outer surfaces of the roof can be taken to be $h_i = 0.9$ and $h_o = 2.1$ Btu/h · ft² · °F, respectively. The house and the interior surfaces of the walls and the floor are maintained at a constant temperature of 70°F during the night, and the emissivity of both surfaces of the concrete roof is 0.9. Considering both radiation and convection heat transfers and using the explicit finite difference method with a mesh size of $\Delta x = 1$ in. and a time step of $\Delta t = 5$ min, determine the temperatures of the inner and outer surfaces of the roof at 6 AM. Also, determine the average rate of heat transfer through the roof during that night.

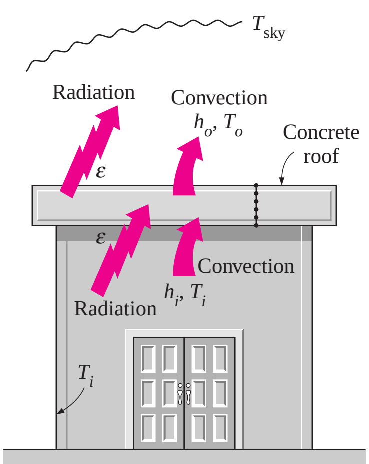


FIGURE P5–114E

5–115 Solar radiation incident on a large body of clean water ($k = 0.61$ W/m · °C and $\alpha = 0.15 \times 10^{-6}$ m²/s) such as a lake, a river, or a pond is mostly absorbed by water, and the amount of absorption varies with depth. For solar radiation incident at a 45° angle on a 1-m-deep large pond whose bottom surface is black (zero reflectivity), for example, 2.8 percent of the solar energy is reflected back to the atmosphere, 37.9 percent is absorbed by the bottom surface, and the remaining 59.3 percent is absorbed by the water body. If the pond is considered to be four layers of equal thickness (0.25 m in this case), it can be shown that 47.3 percent of the incident solar energy is absorbed by the top layer, 6.1 percent by the upper mid layer, 3.6 percent by the lower mid layer, and 2.4 percent by the bottom layer [for more information see Çengel and Özişik, *Solar Energy*, 33, no. 6 (1984), pp. 581–591]. The radiation absorbed by the water can be treated conveniently as heat generation in the heat transfer analysis of the pond.

Consider a large 1-m-deep pond that is initially at a uniform temperature of 15°C throughout. Solar energy is incident on the pond surface at 45° at an average rate of 500 W/m² for a period of 4 h. Assuming no convection currents in the water and using the explicit finite difference method with a mesh size of $\Delta x = 0.25$ m and a time step of $\Delta t = 15$ min, determine the temperature distribution in the pond under the most favorable conditions (i.e., no heat losses from the top or bottom surfaces of the pond). The solar energy absorbed by the bottom surface of the pond can be treated as a heat flux to the water at that surface in this case.

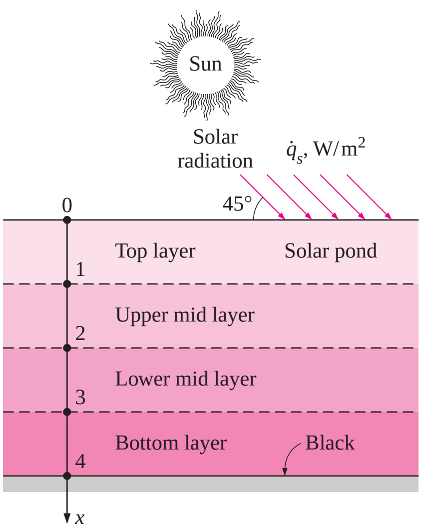


FIGURE P5–115

5–116 Reconsider Problem 5–115. The absorption of solar radiation in that case can be expressed more accurately as a fourth-degree polynomial as

$$\dot{g}(x) = \dot{q}_s(0.859 - 3.415x + 6.704x^2 - 6.339x^3 + 2.278x^4),\ \text{W/m}^3$$

where $\dot{q}_s$ is the solar flux incident on the surface of the pond in W/m^2 and x is the distance from the free surface of the pond in m. Solve Problem 5–115 using this relation for the absorption of solar radiation.

5–117 A hot surface at 120°C is to be cooled by attaching 8 cm long, 0.8 cm in diameter aluminum pin fins (k = 237 W/m · °C and $\alpha = 97.1 \times 10^{-6}$ m^2/s) to it with a center-to-center distance of 1.6 cm. The temperature of the surrounding medium is 15°C, and the heat transfer coefficient on the surfaces is 35 W/m^2 · °C. Initially, the fins are at a uniform temperature of 30°C, and at time t = 0, the temperature of the hot surface is raised to 120°C. Assuming one-dimensional heat conduction along the fin and taking the nodal spacing to be Δx = 2 cm and a time step to be Δt = 0.5 s, determine the nodal temperatures after 5 min by using the explicit finite difference method. Also, determine how long it will take for steady conditions to be reached.

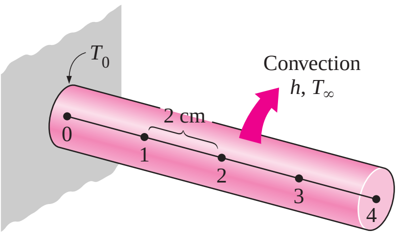


FIGURE P5–117

5–118E Consider a large plane wall of thickness L = 0.3 ft and thermal conductivity k = 1.2 Btu/h · ft · °F in space. The wall is covered with a material having an emissivity of ε = 0.80 and a solar absorptivity of α_s = 0.45. The inner surface of the wall is maintained at 520 R at all times, while the outer surface is exposed to solar radiation that is incident at a rate of $\dot{q}_s$ = 300 Btu/h · ft^2. The outer surface is also losing heat by radiation to deep space at 0 R. Using a uniform nodal spacing of Δx = 0.1 ft, (*a*) obtain the finite difference formulation for steady one-dimensional heat conduction and (*b*) determine the nodal temperatures by solving those equations.

Answers: (*b*) 522 R, 525 R, 527 R

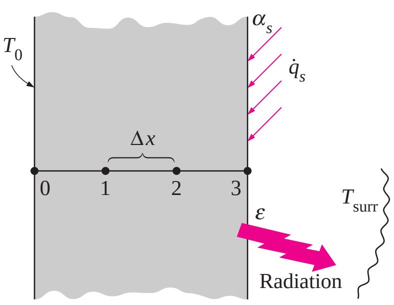


FIGURE P5–118E

5–119 Frozen food items can be defrosted by simply leaving them on the counter, but it takes too long. The process can be speeded up considerably for flat items such as steaks by placing them on a large piece of highly conducting metal, called the defrosting plate, which serves as a fin. The increased surface area enhances heat transfer and thus reduces the defrosting time.

Consider two 1.5-cm-thick frozen steaks at −18°C that resemble a 15-cm-diameter circular object when placed next to each other. The steaks are now placed on a 1-cm-thick black-anodized circular aluminum defrosting plate (k = 237 W/m · °C, $\alpha = 97.1 \times 10^{-6}$ m^2/s, and ε = 0.90) whose outer diameter is 30 cm. The properties of the frozen steaks are ρ = 970 kg/m^3, C_p = 1.55 kJ/kg · °C, k = 1.40 W/m · °C, $\alpha = 0.93 \times 10^{-6}$ m^2/s, and ε = 0.95, and the heat of fusion is h_{if} = 187 kJ/kg. The steaks can be considered to be defrosted when their average temperature is 0°C and all of the ice in the steaks is melted. Initially, the defrosting plate is at the room temperature of 20°C, and the wooden countertop it is placed on can be treated as insulation. Also, the surrounding surfaces can be taken to be at the same temperature as the ambient air, and the convection heat transfer coefficient for all exposed surfaces can be taken to be 12 W/m^2 · °C. Heat transfer from the lateral surfaces of the steaks and the defrosting plate can be neglected. Assuming one-dimensional heat conduction in both the steaks and the defrosting plate and using the explicit finite difference method, determine how long it will take to defrost the steaks. Use four nodes with a nodal spacing of Δx = 0.5 cm for the steaks, and three nodes with a nodal spacing of Δr = 3.75 cm for the exposed portion of the defrosting plate. Also, use a time step of Δt = 5 s. *Hint:* First, determine the total amount of heat transfer needed to defrost the steaks, and then determine how long it will take to transfer that much heat.

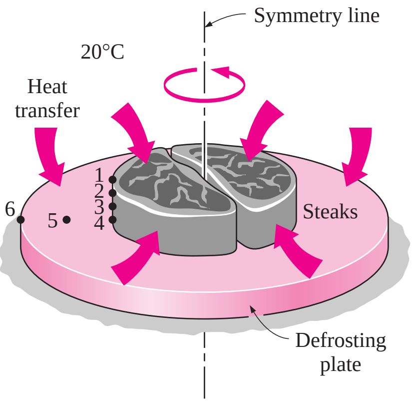


FIGURE P5–119

5–120 Repeat Problem 5–119 for a copper defrosting plate using a time step of $\Delta t = 3$ s.

Design and Essay Problems

5–121 Write a two-page essay on the finite element method, and explain why it is used in most commercial engineering software packages. Also explain how it compares to the finite difference method.

5–122 Numerous professional software packages are available in the market for performing heat transfer analysis, and they are widely advertised in professional magazines such as the *Mechanical Engineering* magazine published by the American Society of Mechanical Engineers (ASME). Your company decides to purchase such a software package and asks you to prepare a report on the available packages, their costs, capabilities, ease of use, and compatibility with the available hardware, and other software as well as the reputation of the software company, their history, financial health, customer support, training, and future prospects, among other things. After a preliminary investigation, select the top three packages and prepare a full report on them.

5–123 Design a defrosting plate to speed up defrosting of flat food items such as frozen steaks and packaged vegetables and evaluate its performance using the finite difference method (see Prob. 5–119). Compare your design to the defrosting plates currently available on the market. The plate must perform well, and it must be suitable for purchase and use as a household utensil, durable, easy to clean, easy to manufacture, and affordable. The frozen food is expected to be at an initial temperature of -18°C at the beginning of the thawing process and 0°C at the end with all the ice melted. Specify the material, shape, size, and thickness of the proposed plate. Justify your recommendations by calculations. Take the ambient and surrounding surface temperatures to be 20°C and the convection heat transfer coefficient to be 15 W/m$^2 \cdot$ °C in your analysis. For a typical case, determine the defrosting time with and without the plate.

5–124 Design a fire-resistant safety box whose outer dimensions are 0.5 m $\times$ 0.5 m $\times$ 0.5 m that will protect its combustible contents from fire which may last up to 2 h. Assume the box will be exposed to an environment at an average temperature of 700°C with a combined heat transfer coefficient of 70 W/m$^2 \cdot$ °C and the temperature inside the box must be below 150°C at the end of 2 h. The cavity of the box must be as large as possible while meeting the design constraints, and the insulation material selected must withstand the high temperatures to which it will be exposed. Cost, durability, and strength are also important considerations in the selection of insulation materials.

CHAPTER

6

FUNDAMENTALS OF CONVECTION

So far, we have considered *conduction,* which is the mechanism of heat transfer through a solid or a quiescent fluid. We now consider *convection,* which is the mechanism of heat transfer through a fluid in the presence of bulk fluid motion.

Convection is classified as *natural* (or *free*) and *forced convection,* depending on how the fluid motion is initiated. In forced convection, the fluid is forced to flow over a surface or in a pipe by external means such as a pump or a fan. In natural convection, any fluid motion is caused by natural means such as the buoyancy effect, which manifests itself as the rise of warmer fluid and the fall of the cooler fluid. Convection is also classified as *external* and *internal,* depending on whether the fluid is forced to flow over a surface or in a channel.

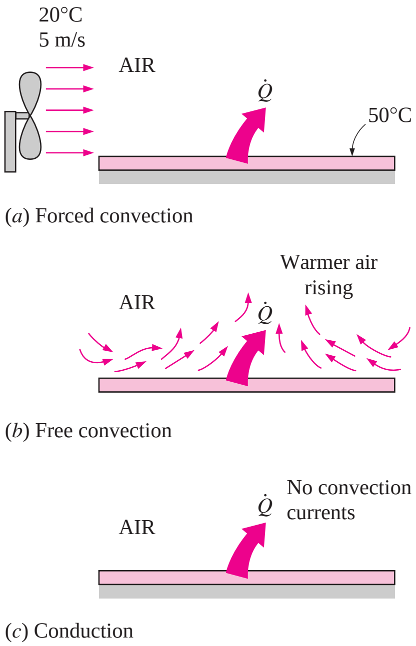
We start this chapter with a general physical description of the convection mechanism. We then discuss the *velocity* and *thermal boundary layers,* and *laminar and turbulent flows.* We continue with the discussion of the dimensionless *Reynolds, Prandtl,* and *Nusselt numbers,* and their physical significance. Next we derive the *convection equations* of on the basis of mass, momentum, and energy conservation, and obtain solutions for *flow over a flat plate.* We then nondimensionalize the convection equations, and obtain functional forms of friction and convection coefficients. Finally, we present analogies between momentum and heat transfer.

CONTENTS

6–1 ▪ PHYSICAL MECHANISM OF CONVECTION

We mentioned earlier that there are three basic mechanisms of heat transfer: conduction, convection, and radiation. Conduction and convection are similar in that both mechanisms require the presence of a material medium. But they are different in that convection requires the presence of fluid motion.

Heat transfer through a solid is always by conduction, since the molecules of a solid remain at relatively fixed positions. Heat transfer through a liquid or gas, however, can be by conduction or convection, depending on the presence of any bulk fluid motion. Heat transfer through a fluid is by convection in the presence of bulk fluid motion and by conduction in the absence of it. Therefore, conduction in a fluid can be viewed as the limiting case of convection, corresponding to the case of quiescent fluid (Fig. 6–1).



(*a*) Forced convection

(*b*) Free convection

(*c*) Conduction

FIGURE 6–1
Heat transfer from a hot surface to the surrounding fluid by convection and conduction.

Convection heat transfer is complicated by the fact that it involves fluid motion as well as heat conduction. The fluid motion enhances heat transfer, since it brings hotter and cooler chunks of fluid into contact, initiating higher rates of conduction at a greater number of sites in a fluid. Therefore, the rate of heat transfer through a fluid is much higher by convection than it is by conduction. In fact, the higher the fluid velocity, the higher the rate of heat transfer.

To clarify this point further, consider steady heat transfer through a fluid contained between two parallel plates maintained at different temperatures, as shown in Figure 6–2. The temperatures of the fluid and the plate will be the same at the points of contact because of the continuity of temperature. Assuming no fluid motion, the energy of the hotter fluid molecules near the hot plate will be transferred to the adjacent cooler fluid molecules. This energy will then be transferred to the next layer of the cooler fluid molecules. This energy will then be transferred to the next layer of the cooler fluid, and so on, until it is finally transferred to the other plate. This is what happens during conduction through a fluid. Now let us use a syringe to draw some fluid near the hot plate and inject it near the cold plate repeatedly. You can imagine that this will speed up the heat transfer process considerably, since some energy is carried to the other side as a result of fluid motion.

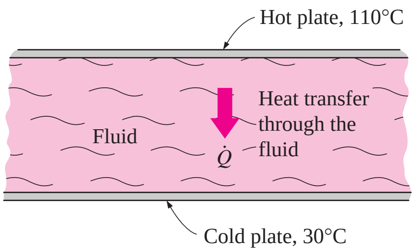


FIGURE 6–2
Heat transfer through a fluid sandwiched between two parallel plates.

Consider the cooling of a hot iron block with a fan blowing air over its top surface, as shown in Figure 6–3. We know that heat will be transferred from the hot block to the surrounding cooler air, and the block will eventually cool. We also know that the block will cool faster if the fan is switched to a higher speed. Replacing air by water will enhance the convection heat transfer even more.

Experience shows that convection heat transfer strongly depends on the fluid properties *dynamic viscosity* μ, *thermal conductivity k, density* ρ, and *specific heat* C_p, as well as the *fluid velocity* $\mathcal{V}$. It also depends on the *geometry* and the *roughness* of the solid surface, in addition to the *type of fluid flow* (such as being streamlined or turbulent). Thus, we expect the convection heat transfer relations to be rather complex because of the dependence of convection on so many variables. This is not surprising, since convection is the most complex mechanism of heat transfer.

Despite the complexity of convection, the rate of convection heat transfer is observed to be proportional to the temperature difference and is conveniently expressed by **Newton's law of cooling** as

$$\dot{q}_{\text{conv}} = h(T_s - T_\infty) \qquad (\text{W/m}^2) \tag{6-1}$$

or

$$\dot{Q}_{\text{conv}} = hA_s(T_s - T_\infty) \qquad (\text{W}) \tag{6-2}$$

where

h = convection heat transfer coefficient, $\text{W/m}^2 \cdot {}^\circ\text{C}$
A_s = heat transfer surface area, m^2
T_s = temperature of the surface, °C
T_∞ = temperature of the fluid sufficiently far from the surface, °C

Judging from its units, the **convection heat transfer coefficient** h can be defined as *the rate of heat transfer between a solid surface and a fluid per unit surface area per unit temperature difference.*

You should not be deceived by the simple appearance of this relation, because the convection heat transfer coefficient h depends on the several of the mentioned variables, and thus is difficult to determine.

When a fluid is forced to flow over a solid surface that is nonporous (i.e., impermeable to the fluid), it is observed that the fluid in motion comes to a complete stop at the surface and assumes a zero velocity relative to the surface. That is, the fluid layer in direct contact with a solid surface "sticks" to the surface and there is no slip. In fluid flow, this phenomenon is known as the **no-slip condition,** and it is due to the viscosity of the fluid (Fig. 6–4).

The no-slip condition is responsible for the development of the velocity profile for flow. Because of the friction between the fluid layers, the layer that sticks to the wall slows the adjacent fluid layer, which slows the next layer, and so on. A consequence of the no-slip condition is that all velocity profiles must have zero values at the points of contact between a fluid and a solid. The only exception to the no-slip condition occurs in extremely rarified gases.

A similar phenomenon occurs for the temperature. When two bodies at different temperatures are brought into contact, heat transfer occurs until both bodies assume the same temperature at the point of contact. Therefore, a fluid and a solid surface will have the same temperature at the point of contact. This is known as **no-temperature-jump condition.**

An implication of the no-slip and the no-temperature jump conditions is that heat transfer from the solid surface to the fluid layer adjacent to the surface is by *pure conduction,* since the fluid layer is motionless, and can be expressed as

$$\dot{q}_{\text{conv}} = \dot{q}_{\text{cond}} = -k_{\text{fluid}} \left.\frac{\partial T}{\partial y}\right|_{y=0} \qquad (\text{W/m}^2) \tag{6-3}$$

where T represents the temperature distribution in the fluid and $(\partial T/\partial y)_{y=0}$ is the *temperature gradient* at the surface. This heat is then *convected away* from the surface as a result of fluid motion. Note that convection heat transfer from a solid surface to a fluid is merely the conduction heat transfer from the solid surface to the fluid layer adjacent to the surface. Therefore, we can equate Eqs. 6-1 and 6-3 for the heat flux to obtain

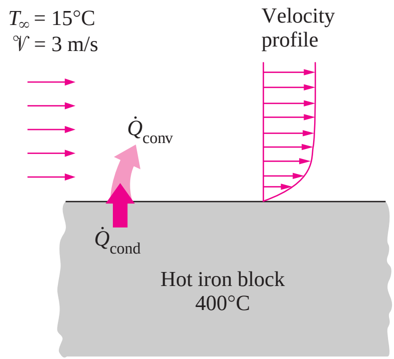


FIGURE 6–3
The cooling of a hot block by forced convection.

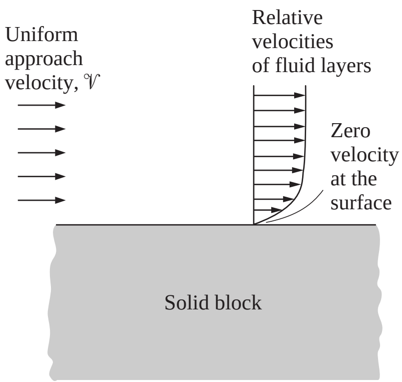


FIGURE 6–4
A fluid flowing over a stationary surface comes to a complete stop at the surface because of the no-slip condition.

$$h = \frac{-k_{\text{fluid}}(\partial T/\partial y)_{y=0}}{T_s - T_\infty} \qquad (\text{W/m}^2 \cdot {}^\circ\text{C}) \tag{6-4}$$

for the determination of the *convection heat transfer coefficient* when the temperature distribution within the fluid is known.

The convection heat transfer coefficient, in general, varies along the flow (or x-) direction. The *average* or *mean* convection heat transfer coefficient for a surface in such cases is determined by properly averaging the *local* convection heat transfer coefficients over the entire surface.

Nusselt Number

In convection studies, it is common practice to nondimensionalize the governing equations and combine the variables, which group together into *dimensionless numbers* in order to reduce the number of total variables. It is also common practice to nondimensionalize the heat transfer coefficient h with the Nusselt number, defined as

$$\text{Nu} = \frac{hL_c}{k} \tag{6-5}$$

where k is the thermal conductivity of the fluid and L_c is the *characteristic length.* The Nusselt number is named after Wilhelm Nusselt, who made significant contributions to convective heat transfer in the first half of the twentieth century, and it is viewed as the *dimensionless convection heat transfer coefficient.*

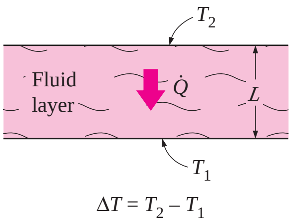


FIGURE 6–5
Heat transfer through a fluid layer of thickness L and temperature difference ΔT.

To understand the physical significance of the Nusselt number, consider a fluid layer of thickness L and temperature difference $\Delta T = T_2 - T_1$, as shown in Fig. 6–5. Heat transfer through the fluid layer will be by *convection* when the fluid involves some motion and by *conduction* when the fluid layer is motionless. Heat flux (the rate of heat transfer per unit time per unit surface area) in either case will be

$$\dot{q}_{\text{conv}} = h\Delta T \tag{6-6}$$

and

$$\dot{q}_{\text{cond}} = k\frac{\Delta T}{L} \tag{6-7}$$

Taking their ratio gives

$$\frac{\dot{q}_{\text{conv}}}{\dot{q}_{\text{cond}}} = \frac{h\Delta T}{k\Delta T/L} = \frac{hL}{k} = \text{Nu} \tag{6-8}$$

which is the Nusselt number. Therefore, the Nusselt number represents the enhancement of heat transfer through a fluid layer as a result of convection relative to conduction across the same fluid layer. The larger the Nusselt number, the more effective the convection. A Nusselt number of Nu $= 1$ for a fluid layer represents heat transfer across the layer by pure conduction.

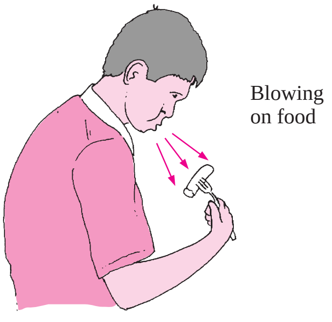


FIGURE 6–6
We resort to forced convection whenever we need to increase the rate of heat transfer.

We use forced convection in daily life more often than you might think (Fig. 6–6). We resort to forced convection whenever we want to increase the rate of heat transfer from a hot object. For example, we turn on the fan on hot summer days to help our body cool more effectively. The higher the fan speed, the better we feel. We *stir* our soup and *blow* on a hot slice of pizza to make them cool faster. The air on *windy* winter days feels much colder than it actually is. The simplest solution to heating problems in electronics packaging is to use a large enough fan.

6–2 ▪ CLASSIFICATION OF FLUID FLOWS

Convection heat transfer is closely tied with fluid mechanics, which is the science that deals with the behavior of fluids at rest or in motion, and the interaction of fluids with solids or other fluids at the boundaries. There are a wide variety of fluid flow problems encountered in practice, and it is usually convenient to classify them on the basis of some common characteristics to make it feasible to study them in groups. There are many ways to classify the fluid flow problems, and below we present some general categories.

Viscous versus Inviscid Flow

When two fluid layers move relative to each other, a friction force develops between them and the slower layer tries to slow down the faster layer. This internal resistance to flow is called the **viscosity,** which is a measure of internal stickiness of the fluid. Viscosity is caused by cohesive forces between the molecules in liquids, and by the molecular collisions in gases. There is no fluid with zero viscosity, and thus all fluid flows involve viscous effects to some degree. Flows in which the effects of viscosity are significant are called **viscous flows.** The effects of viscosity are very small in some flows, and neglecting those effects greatly simplifies the analysis without much loss in accuracy. Such idealized flows of zero-viscosity fluids are called frictionless or **inviscid flows.**

Internal versus External Flow

A fluid flow is classified as being internal and external, depending on whether the fluid is forced to flow in a confined channel or over a surface. The flow of an unbounded fluid over a surface such as a plate, a wire, or a pipe is **external flow.** The flow in a pipe or duct is **internal flow** if the fluid is completely bounded by solid surfaces. Water flow in a pipe, for example, is internal flow, and air flow over an exposed pipe during a windy day is external flow (Fig. 6–7). The flow of liquids in a pipe is called *open-channel flow* if the pipe is partially filled with the liquid and there is a free surface. The flow of water in rivers and irrigation ditches are examples of such flows.

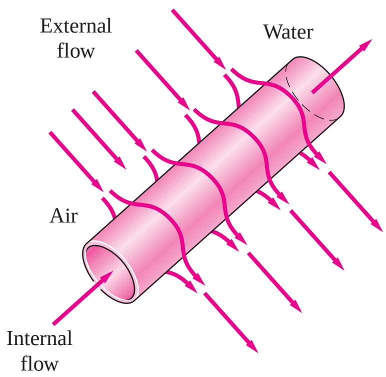


FIGURE 6–7
Internal flow of water in a pipe and the external flow of air over the same pipe.

Compressible versus Incompressible Flow

A fluid flow is classified as being *compressible* or *incompressible,* depending on the density variation of the fluid during flow. The densities of liquids are essentially constant, and thus the flow of liquids is typically incompressible. Therefore, liquids are usually classified as *incompressible substances.* A pressure of 210 atm, for example, will cause the density of liquid water at 1 atm to change by just 1 percent. Gases, on the other hand, are highly compressible. A pressure change of just 0.01 atm, for example, will cause a change of 1 percent in the density of atmospheric air. However, gas flows can be treated as incompressible if the density changes are under about 5 percent, which is usually the case when the flow velocity is less than 30 percent of the velocity of sound in that gas (i.e., the Mach number of flow is less than 0.3). The velocity of sound in air at room temperature is 346 m/s. Therefore, the compressibility effects of air can be neglected at speeds under 100 m/s. Note that the flow of a gas is not necessarily a compressible flow.

Laminar versus Turbulent Flow

Some flows are smooth and orderly while others are rather chaotic. The highly ordered fluid motion characterized by smooth streamlines is called **laminar.** The flow of high-viscosity fluids such as oils at low velocities is typically laminar. The highly disordered fluid motion that typically occurs at high velocities characterized by velocity fluctuations is called **turbulent.** The flow of low-viscosity fluids such as air at high velocities is typically turbulent. The flow regime greatly influences the heat transfer rates and the required power for pumping.

Natural (or Unforced) versus Forced Flow

A fluid flow is said to be natural or forced, depending on how the fluid motion is initiated. In **forced flow,** a fluid is forced to flow over a surface or in a pipe by external means such as a pump or a fan. In **natural flows,** any fluid motion is due to a natural means such as the buoyancy effect, which manifests itself as the rise of the warmer (and thus lighter) fluid and the fall of cooler (and thus denser) fluid. This thermosiphoning effect is commonly used to replace pumps in solar water heating systems by placing the water tank sufficiently above the solar collectors (Fig. 6–8).

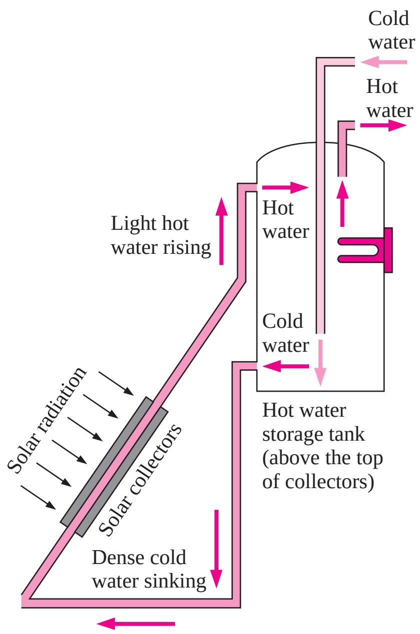


FIGURE 6–8
Natural circulation of water in a solar water heater by thermosiphoning.

Steady versus Unsteady (Transient) Flow

The terms *steady* and *uniform* are used frequently in engineering, and thus it is important to have a clear understanding of their meanings. The term **steady** implies *no change with time.* The opposite of steady is **unsteady,** or **transient.** The term *uniform,* however, implies *no change with location* over a specified region.

Many devices such as turbines, compressors, boilers, condensers, and heat exchangers operate for long periods of time under the same conditions, and they are classified as *steady-flow devices.* During steady flow, the fluid properties can change from point to point within a device, but at any fixed point they remain constant.

One-, Two-, and Three-Dimensional Flows

A flow field is best characterized by the velocity distribution, and thus a flow is said to be one-, two-, or three-dimensional if the flow velocity $\mathcal{V}$ varies in one, two, or three primary dimensions, respectively. A typical fluid flow involves a three-dimensional geometry and the velocity may vary in all three dimensions rendering the flow three-dimensional [$\mathcal{V}(x, y, z)$ in rectangular or $\mathcal{V}(r, \theta, z)$ in cylindrical coordinates]. However, the variation of velocity in certain direction can be small relative to the variation in other directions, and can be ignored with negligible error. In such cases, the flow can be modeled conveniently as being one- or two-dimensional, which is easier to analyze.

When the entrance effects are disregarded, fluid flow in a circular pipe is *one-dimensional* since the velocity varies in the radial r direction but not in the angular θ- or axial z-directions (Fig. 6–9). That is, the velocity profile is the same at any axial z-location, and it is symmetric about the axis of the pipe. Note that even in this simplest flow, the velocity cannot be uniform across the cross section of the pipe because of the no-slip condition. However, for convenience in calculations, the velocity can be assumed to be constant and thus *uniform* at a cross section. Fluid flow in a pipe usually approximated as *one-dimensional uniform flow.*

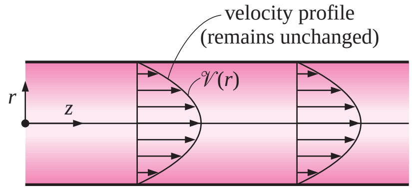


FIGURE 6–9
One-dimensional flow in a circular pipe.

6–3 ▪ VELOCITY BOUNDARY LAYER

Consider the parallel flow of a fluid over a *flat plate,* as shown in Fig. 6–10. Surfaces that are slightly contoured such as turbine blades can also be approximated as flat plates with reasonable accuracy. The x-coordinate is measured along the plate surface from the *leading edge* of the plate in the direction of the flow, and y is measured from the surface in the normal direction. The fluid approaches the plate in the x-direction with a uniform upstream velocity of $\mathcal{V}$, which is practically identical to the free-stream velocity u_∞ over the plate away from the surface (this would not be the case for cross flow over blunt bodies such as a cylinder).

For the sake of discussion, we can consider the fluid to consist of adjacent layers piled on top of each other. The velocity of the particles in the first fluid layer adjacent to the plate becomes zero because of the no-slip condition. This motionless layer slows down the particles of the neighboring fluid layer as a result of friction between the particles of these two adjoining fluid layers at different velocities. This fluid layer then slows down the molecules of the next layer, and so on. Thus, the presence of the plate is felt up to some normal distance δ from the plate beyond which the free-stream velocity u_∞ remains essentially unchanged. As a result, the x-component of the fluid velocity, u, will vary from 0 at $y = 0$ to nearly u_∞ at $y = \delta$ (Fig. 6–11).

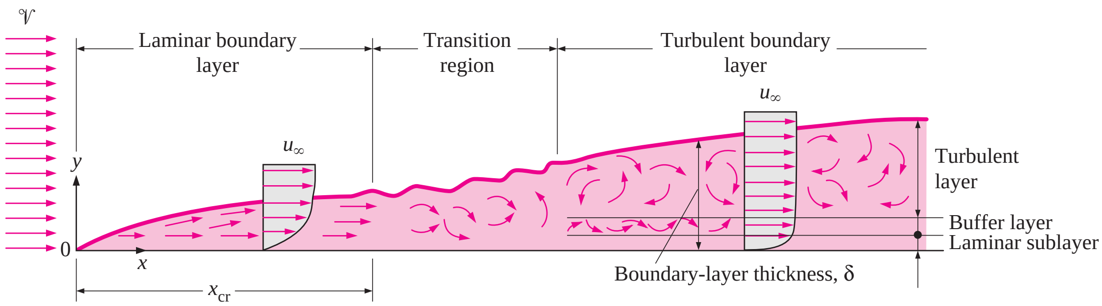


FIGURE 6–10
The development of the boundary layer for flow over a flat plate, and the different flow regimes.

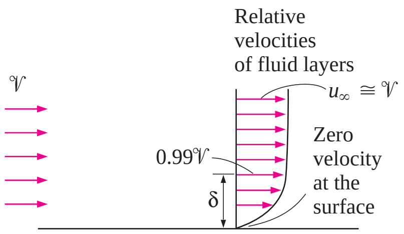


FIGURE 6–11
The development of a boundary layer on a surface is due to the no-slip condition.

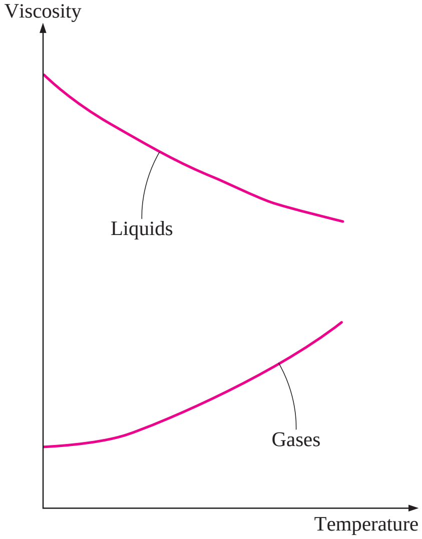


FIGURE 6–12
The viscosity of liquids decreases and the viscosity of gases increases with temperature.

The region of the flow above the plate bounded by δ in which the effects of the viscous shearing forces caused by fluid viscosity are felt is called the **velocity boundary layer.** The *boundary layer thickness,* δ, is typically defined as the distance y from the surface at which $u = 0.99u_\infty$.

The hypothetical line of $u = 0.99u_\infty$ divides the flow over a plate into two regions: the **boundary layer region,** in which the viscous effects and the velocity changes are significant, and the **inviscid flow region,** in which the frictional effects are negligible and the velocity remains essentially constant.

Surface Shear Stress

Consider the flow of a fluid over the surface of a plate. The fluid layer in contact with the surface will try to drag the plate along via friction, exerting a *friction force* on it. Likewise, a faster fluid layer will try to drag the adjacent slower layer and exert a friction force because of the friction between the two layers. Friction force per unit area is called **shear stress,** and is denoted by τ. Experimental studies indicate that the shear stress for most fluids is proportional to the *velocity gradient,* and the shear stress at the wall surface is as

$$\tau_s = \mu \left.\frac{\partial u}{\partial y}\right|_{y=0} \qquad (\text{N/m}^2) \tag{6-9}$$

where the constant of proportionality μ is called the **dynamic viscosity** of the fluid, whose unit is kg/m · s (or equivalently, N · s/m^2, or Pa · s, or poise = 0.1 Pa · s).

The fluids that that obey the linear relationship above are called **Newtonian fluids,** after Sir Isaac Newton who expressed it first in 1687. Most common fluids such as water, air, gasoline, and oils are Newtonian fluids. Blood and liquid plastics are examples of non-Newtonian fluids. In this text we will consider Newtonian fluids only.

In fluid flow and heat transfer studies, the ratio of dynamic viscosity to density appears frequently. For convenience, this ratio is given the name **kinematic viscosity** ν and is expressed as $\nu = \mu/\rho$. Two common units of kinematic viscosity are m^2/s and *stoke* (1 stoke = 1 cm^2/s = 0.0001 m^2/s).

The viscosity of a fluid is a measure of its *resistance to flow,* and it is a strong function of temperature. The viscosities of liquids *decrease* with temperature, whereas the viscosities of gases *increase* with temperature (Fig. 6–12). The viscosities of some fluids at 20°C are listed in Table 6–1. Note that the viscosities of different fluids differ by several orders of magnitude.

The determination of the surface shear stress τ_s from Eq. 6-9 is not practical since it requires a knowledge of the flow velocity profile. A more practical approach in external flow is to relate τ_s to the upstream velocity $\mathcal{V}$ as

$$\tau_s = C_f \frac{\rho \mathcal{V}^2}{2} \qquad (\text{N/m}^2) \tag{6-10}$$

where C_f is the dimensionless **friction coefficient,** whose value in most cases is determined experimentally, and ρ is the density of the fluid. Note that the friction coefficient, in general, will vary with location along the surface. Once the average friction coefficient over a given surface is available, the friction force over the entire surface is determined from

$$F_f = C_f A_s \frac{\rho \mathcal{V}^2}{2} \qquad \text{(N)} \tag{6-11}$$

where A_s is the surface area.

The friction coefficient is an important parameter in heat transfer studies since it is directly related to the heat transfer coefficient and the power requirements of the pump or fan.

6–4 ▪ THERMAL BOUNDARY LAYER

We have seen that a velocity boundary layer develops when a fluid flows over a surface as a result of the fluid layer adjacent to the surface assuming the surface velocity (i.e., zero velocity relative to the surface). Also, we defined the velocity boundary layer as the region in which the fluid velocity varies from zero to $0.99u_\infty$. Likewise, a *thermal boundary layer* develops when a fluid at a specified temperature flows over a surface that is at a different temperature, as shown in Fig. 6–13.

Consider the flow of a fluid at a uniform temperature of T_∞ over an isothermal flat plate at temperature T_s. The fluid particles in the layer adjacent to the surface will reach thermal equilibrium with the plate and assume the surface temperature T_s. These fluid particles will then exchange energy with the particles in the adjoining-fluid layer, and so on. As a result, a temperature profile will develop in the flow field that ranges from T_s at the surface to T_∞ sufficiently far from the surface. The flow region over the surface in which the temperature variation in the direction normal to the surface is significant is the **thermal boundary layer.** The *thickness* of the thermal boundary layer δ_t at any location along the surface is defined as *the distance from the surface at which the temperature difference* $T - T_s$ *equals* $0.99(T_\infty - T_s)$. Note that for the special case of $T_s = 0$, we have $T = 0.99T_\infty$ at the outer edge of the thermal boundary layer, which is analogous to $u = 0.99u_\infty$ for the velocity boundary layer.

The thickness of the thermal boundary layer increases in the flow direction, since the effects of heat transfer are felt at greater distances from the surface further down stream.

The convection heat transfer rate anywhere along the surface is directly related to the temperature gradient at that location. Therefore, the shape of the temperature profile in the thermal boundary layer dictates the convection heat transfer between a solid surface and the fluid flowing over it. In flow over a heated (or cooled) surface, both velocity and thermal boundary layers will develop simultaneously. Noting that the fluid velocity will have a strong influence on the temperature profile, the development of the velocity boundary layer relative to the thermal boundary layer will have a strong effect on the convection heat transfer.

Prandtl Number

The relative thickness of the velocity and the thermal boundary layers is best described by the *dimensionless* parameter **Prandtl number,** defined as

$$\text{Pr} = \frac{\text{Molecular diffusivity of momentum}}{\text{Molecular diffusivity of heat}} = \frac{\nu}{\alpha} = \frac{\mu C_p}{k} \tag{6-12}$$

TABLE 6–1

Dynamic viscosities of some fluids at 1 atm and 20°C (unless otherwise stated)

Fluid	Dynamic viscosity μ, kg/m · s
Glycerin:	
−20°C	134.0
0°C	12.1
20°C	1.49
40°C	0.27
Engine oil:	
SAE 10W	0.10
SAE 10W30	0.17
SAE 30	0.29
SAE 50	0.86
Mercury	0.0015
Ethyl alcohol	0.0012
Water:	
0°C	0.0018
20°C	0.0010
100°C (liquid)	0.0003
100°C (vapor)	0.000013
Blood, 37°C	0.0004
Gasoline	0.00029
Ammonia	0.00022
Air	0.000018
Hydrogen, 0°C	0.000009

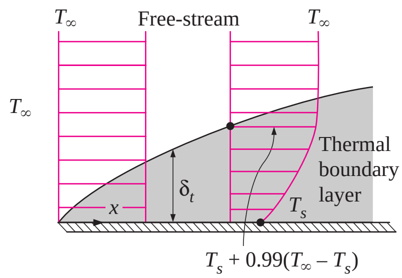


FIGURE 6–13

Thermal boundary layer on a flat plate (the fluid is hotter than the plate surface).

TABLE 6–2

Typical ranges of Prandtl numbers for common fluids

Fluid	Pr
Liquid metals	0.004–0.030
Gases	0.7–1.0
Water	1.7–13.7
Light organic fluids	5–50
Oils	50–100,000
Glycerin	2000–100,000

It is named after Ludwig Prandtl, who introduced the concept of boundary layer in 1904 and made significant contributions to boundary layer theory. The Prandtl numbers of fluids range from less than 0.01 for liquid metals to more than 100,000 for heavy oils (Table 6–2). Note that the Prandtl number is in the order of 10 for water.

The Prandtl numbers of gases are about 1, which indicates that both momentum and heat dissipate through the fluid at about the same rate. Heat diffuses very quickly in liquid metals ($Pr \ll 1$) and very slowly in oils ($Pr \gg 1$) relative to momentum. Consequently the thermal boundary layer is much thicker for liquid metals and much thinner for oils relative to the velocity boundary layer.

6–5 ▪ LAMINAR AND TURBULENT FLOWS

If you have been around smokers, you probably noticed that the cigarette smoke rises in a smooth plume for the first few centimeters and then starts fluctuating randomly in all directions as it continues its journey toward the lungs of others (Fig. 6–14). Likewise, a careful inspection of flow in a pipe reveals that the fluid flow is streamlined at low velocities but turns chaotic as the velocity is increased above a critical value, as shown in Figure 6–15. The flow regime in the first case is said to be **laminar,** characterized by *smooth streamlines* and *highly-ordered motion,* and **turbulent** in the second case, where it is characterized by *velocity fluctuations* and *highly-disordered motion.* The **transition** from laminar to turbulent flow does not occur suddenly; rather, it occurs over some region in which the flow fluctuates between laminar and turbulent flows before it becomes fully turbulent.

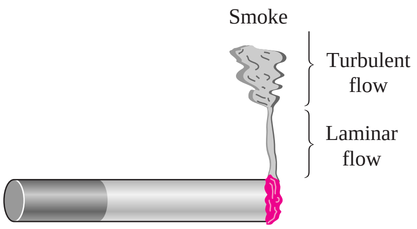


FIGURE 6–14
Laminar and turbulent flow regimes of cigarette smoke.

We can verify the existence of these laminar, transition, and turbulent flow regimes by injecting some dye streak into the flow in a glass tube, as the British scientist Osborn Reynolds (1842–1912) did over a century ago. We will observe that the dye streak will form a *straight and smooth line* at low velocities when the flow is laminar (we may see some blurring because of molecular diffusion), will have *bursts of fluctuations* in the transition regime, and will *zigzag rapidly and randomly* when the flow becomes fully turbulent. These zigzags and the dispersion of the dye are indicative of the fluctuations in the main flow and the rapid mixing of fluid particles from adjacent layers.

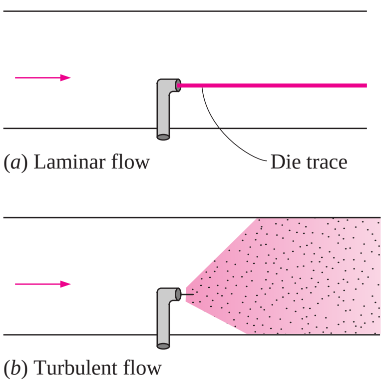


FIGURE 6–15
The behavior of colored fluid injected into the flow in laminar and turbulent flows in a tube.

Typical velocity profiles in laminar and turbulent flow are also given in Figure 6–10. Note that the velocity profile is approximately parabolic in laminar flow and becomes flatter in turbulent flow, with a sharp drop near the surface. The turbulent boundary layer can be considered to consist of three layers. The very thin layer next to the wall where the viscous effects are dominant is the **laminar sublayer.** The velocity profile in this layer is nearly linear, and the flow is streamlined. Next to the laminar sublayer is the **buffer layer,** in which the turbulent effects are significant but not dominant of the diffusion effects, and next to it is the **turbulent layer,** in which the turbulent effects dominate.

The *intense mixing* of the fluid in turbulent flow as a result of rapid fluctuations enhances heat and momentum transfer between fluid particles, which increases the friction force on the surface and the convection heat transfer rate. It also causes the boundary layer to enlarge. Both the friction and heat transfer coefficients reach maximum values when the flow becomes *fully turbulent.* So it will come as no surprise that a special effort is made in the design of heat transfer coefficients associated with turbulent flow. The enhancement in heat transfer in turbulent flow does not come for free, however. It may be necessary to use a larger pump to overcome the larger friction forces accompanying the higher heat transfer rate.

Reynolds Number

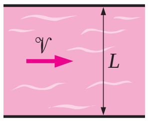
The transition from laminar to turbulent flow depends on the *surface geometry, surface roughness, free-stream velocity, surface temperature,* and *type of fluid,* among other things. After exhaustive experiments in the 1880s, Osborn Reynolds discovered that the flow regime depends mainly on the ratio of the *inertia forces* to *viscous forces* in the fluid. This ratio is called the **Reynolds number,** which is a *dimensionless* quantity, and is expressed for external flow as (Fig. 6–16)

$$\text{Re} = \frac{\text{Inertia forces}}{\text{Viscous}} = \frac{\mathcal{V} L_c}{\nu} = \frac{\rho \mathcal{V} L_c}{\mu} \tag{6-13}$$

where $\mathcal{V}$ is the upstream velocity (equivalent to the free-stream velocity u_∞ for a flat plate), L_c is the characteristic length of the geometry, and $\nu = \mu/\rho$ is the kinematic viscosity of the fluid. For a flat plate, the characteristic length is the distance x from the leading edge. Note that kinematic viscosity has the unit m^2/s, which is identical to the unit of thermal diffusivity, and can be viewed as *viscous diffusivity* or *diffusivity for momentum.*

At *large* Reynolds numbers, the inertia forces, which are proportional to the density and the velocity of the fluid, are large relative to the viscous forces, and thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid. At *small* Reynolds numbers, however, the viscous forces are large enough to overcome the inertia forces and to keep the fluid "in line." Thus the flow is *turbulent* in the first case and *laminar* in the second.

The Reynolds number at which the flow becomes turbulent is called the **critical Reynolds number.** The value of the critical Reynolds number is different for different geometries. For flow over a flat plate, the generally accepted value of the critical Reynolds number is $\text{Re}_{cr} = \mathcal{V} x_{cr}/\nu = u_\infty x_{cr}/\nu = 5 \times 10^5$, where x_{cr} is the distance from the leading edge of the plate at which transition from laminar to turbulent flow occurs. The value of Re_{cr} may change substantially, however, depending on the level of turbulence in the free stream.

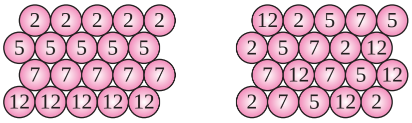


$$\text{Re} = \frac{\text{Inertia forces}}{\text{Viscous forces}} = \frac{\rho \mathcal{V}^2/L}{\mu \mathcal{V}/L^2} = \frac{\rho \mathcal{V} L}{\mu} = \frac{\mathcal{V} L}{\nu}$$

FIGURE 6–16
The Reynolds number can be viewed as the ratio of the inertia forces to viscous forces acting on a fluid volume element.

6–6 ▪ HEAT AND MOMENTUM TRANSFER IN TURBULENT FLOW

Most flows encountered in engineering practice are turbulent, and thus it is important to understand how turbulence affects wall shear stress and heat transfer. Turbulent flow is characterized by random and rapid fluctuations of groups of fluid particles, called *eddies,* throughout the boundary layer. These fluctuations provide an additional mechanism for momentum and heat transfer. In laminar flow, fluid particles flow in an orderly manner along streamlines, and both momentum and heat are transferred across streamlines by molecular diffusion. In turbulent flow, the transverse motion of eddies transport momentum and heat to other regions of flow before they mix with the rest of the fluid and lose their identity, greatly enhancing momentum and heat



(a) Before turbulence

(b) After turbulence

FIGURE 6–17

The intense mixing in turbulent flow brings fluid particles at different temperatures into close contact, and thus enhances heat transfer.

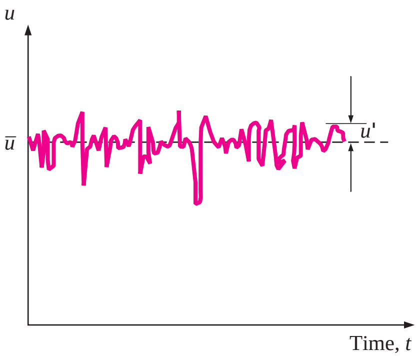


FIGURE 6–18

Fluctuations of the velocity component u with time at a specified location in turbulent flow.

transfer. As a result, turbulent flow is associated with much higher values of friction and heat transfer coefficients (Fig. 6–17).

Even when the mean flow is steady, the eddying motion in turbulent flow causes significant fluctuations in the values of velocity, temperature, pressure, and even density (in compressible flow). Figure 6–18 shows the variation of the instantaneous velocity component u with time at a specified location, as can be measured with a hot-wire anemometer probe or other sensitive device. We observe that the instantaneous values of the velocity fluctuate about a mean value, which suggests that the velocity can be expressed as the sum of a *mean value* $\bar{u}$ and a *fluctuating component* u',

$$u = \bar{u} + u' \tag{6-14}$$

This is also the case for other properties such as the velocity component v in the y direction, and thus $v = \bar{v} + v'$, $P = \bar{P} + P'$, and $T = \bar{T} + T'$. The mean value of a property at some location is determined by averaging it over a time interval that is sufficiently large so that the net effect of fluctuations is zero. Therefore, the time average of fluctuating components is zero, e.g., $\bar{u}' = 0$. The magnitude of u' is usually just a few percent of $\bar{u}$, but the high frequencies of eddies (in the order of a thousand per second) makes them very effective for the transport of momentum and thermal energy. In *steady* turbulent flow, the mean values of properties (indicated by an overbar) are independent of time.

Consider the upward eddy motion of a fluid during flow over a surface. The mass flow rate of fluid per unit area normal to flow is $\rho v'$. Noting that $h = C_pT$ represents the energy of the fluid and T' is the eddy temperature relative to the mean value, the rate of thermal energy transport by turbulent eddies is $\dot{q}_t = \rho C_p v'T'$. By a similar argument on momentum transfer, the turbulent shear stress can be shown to be $\tau_t = -\rho\overline{u'v'}$. Note that $\overline{u'v'} \neq 0$ even though $\overline{u'} = 0$ and $\overline{v'} = 0$, and experimental results show that $\overline{u'v'}$ is a negative quantity. Terms such as $-\rho\overline{u'v'}$ are called **Reynolds stresses.**

The random eddy motion of groups of particles resembles the random motion of molecules in a gas—colliding with each other after traveling a certain distance and exchanging momentum and heat in the process. Therefore, momentum and heat transport by eddies in turbulent boundary layers is analogous to the molecular momentum and heat diffusion. Then turbulent wall shear stress and turbulent heat transfer can be expressed in an analogous manner as

$$\tau_t = -\rho\overline{u'v'} = \mu_t\frac{\partial\bar{u}}{\partial y} \quad \text{and} \quad \dot{q}_t = \rho C_p\overline{v'T'} = -k_t\frac{\partial\bar{T}}{\partial y} \tag{6-15}$$

where μ_t is called the **turbulent viscosity,** which accounts for momentum transport by turbulent eddies, and k_t is called the **turbulent thermal conductivity,** which accounts for thermal energy transport by turbulent eddies. Then the total shear stress and total heat flux can be expressed conveniently as

$$\tau_{\text{total}} = (\mu + \mu_t)\frac{\partial\bar{u}}{\partial y} = \rho(\nu + \varepsilon_M)\frac{\partial\bar{u}}{\partial y} \tag{6-16}$$

and

$$\dot{q}_{\text{total}} = -(k + k_t)\frac{\partial\bar{T}}{\partial y} = -\rho C_p(\alpha + \varepsilon_H)\frac{\partial\bar{T}}{\partial y} \tag{6-17}$$

where $\varepsilon_M = \mu_t/\rho$ is the **eddy diffusivity of momentum** and $\varepsilon_H = k_t/\rho C_p$ is the **eddy diffusivity of heat.**

Eddy motion and thus eddy diffusivities are much larger than their molecular counterparts in the core region of a turbulent boundary layer. The eddy motion loses its intensity close to the wall, and diminishes at the wall because of the no-slip condition. Therefore, the velocity and temperature profiles are nearly uniform in the core region of a turbulent boundary layer, but very steep in the thin layer adjacent to the wall, resulting in large velocity and temperature gradients at the wall surface. So it is no surprise that the wall shear stress and wall heat flux are much larger in turbulent flow than they are in laminar flow (Fig. 6–19).

Note that molecular diffusivities ν and α (as well as μ and k) are fluid properties, and their values can be found listed in fluid handbooks. Eddy diffusivities ε_M and ε_H (as well as μ_t and k_t), however are *not* fluid properties and their values depend on flow conditions. Eddy diffusivities ε_M and ε_H decrease towards the wall, becoming zero at the wall.

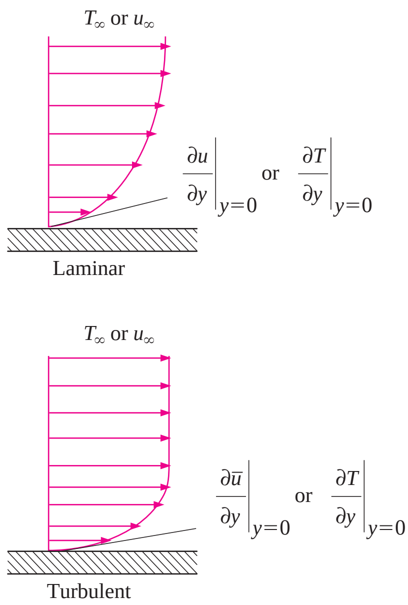


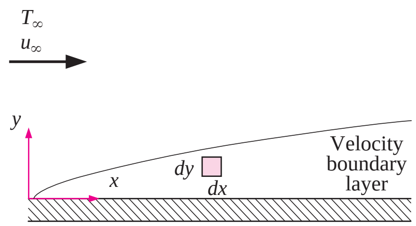
FIGURE 6–19
The velocity and temperature gradients at the wall, and thus the wall shear stress and heat transfer rate, are much larger for turbulent flow than they are for laminar flow (T is shown relative to T_s).

6–7 ▪ DERIVATION OF DIFFERENTIAL CONVECTION EQUATIONS*

In this section we derive the governing equations of fluid flow in the boundary layers. To keep the analysis at a manageable level, we assume the flow to be steady and two-dimensional, and the fluid to be Newtonian with constant properties (density, viscosity, thermal conductivity, etc.).

Consider the parallel flow of a fluid over a surface. We take the flow direction along the surface to be x and the direction normal to the surface to be y, and we choose a differential volume element of length dx, height dy, and unit depth in the z-direction (normal to the paper) for analysis (Fig. 6–20). The fluid flows over the surface with a uniform free-stream velocity u_∞, but the velocity within boundary layer is two-dimensional: the x-component of the velocity is u, and the y-component is v. Note that $u = u(x, y)$ and $v = v(x, y)$ in steady two-dimensional flow.

Next we apply three fundamental laws to this fluid element: Conservation of mass, conservation of momentum, and conservation of energy to obtain the continuity, momentum, and energy equations for laminar flow in boundary layers.



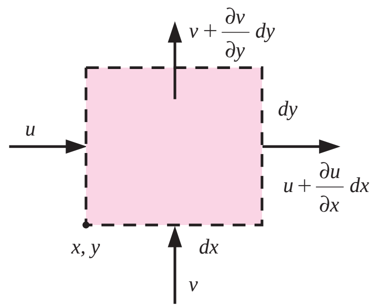


FIGURE 6–20
Differential control volume used in the derivation of mass balance in velocity boundary layer in two-dimensional flow over a surface.

Conservation of Mass Equation

The conservation of mass principle is simply a statement that mass cannot be created or destroyed, and all the mass must be accounted for during an analysis. In steady flow, the amount of mass within the control volume remains constant, and thus the conservation of mass can be expressed as

$$\begin{pmatrix}\text{Rate of mass flow}\\\text{into the control volume}\end{pmatrix} = \begin{pmatrix}\text{Rate of mass flow}\\\text{out of the control volume}\end{pmatrix} \tag{6-18}$$

*This and the upcoming sections of this chapter deal with theoretical aspects of convection, and can be skipped and be used as a reference if desired without a loss in continuity.

Noting that mass flow rate is equal to the product of density, mean velocity, and cross-sectional area normal to flow, the rate at which fluid enters the control volume from the left surface is $\rho u(dy \cdot 1)$. The rate at which the fluid leaves the control volume from the right surface can be expressed as

$$\rho\left(u + \frac{\partial u}{\partial x}dx\right)(dy \cdot 1) \tag{6-19}$$

Repeating this for the y direction and substituting the results into Eq. 6-18, we obtain

$$\rho u(dy \cdot 1) + \rho v(dx \cdot 1) = \rho\left(u + \frac{\partial u}{\partial x}dx\right)(dy \cdot 1) + \rho\left(v + \frac{\partial v}{\partial y}dy\right)(dx \cdot 1) \tag{6-20}$$

Simplifying and dividing by $dx \cdot dy \cdot 1$ gives

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{6-21}$$

This is the *conservation of mass* relation, also known as the **continuity equation,** or **mass balance** for steady two-dimensional flow of a fluid with constant density.

Conservation of Momentum Equations

The differential forms of the equations of motion in the velocity boundary layer are obtained by applying Newton's second law of motion to a differential control volume element in the boundary layer. Newton's second law is an expression for the conservation of momentum, and can be stated as *the net force acting on the control volume is equal to the mass times the acceleration of the fluid element within the control volume, which is also equal to the net rate of momentum outflow from the control volume.*

The forces acting on the control volume consist of *body forces* that act throughout the entire body of the control volume (such as gravity, electric, and magnetic forces) and are proportional to the volume of the body, and *surface forces* that act on the control surface (such as the pressure forces due to hydrostatic pressure and shear stresses due to viscous effects) and are proportional to the surface area. The surface forces appear as the control volume is isolated from its surroundings for analysis, and the effect of the detached body is replaced by a force at that location. Note that pressure represents the compressive force applied on the fluid element by the surrounding fluid, and is always directed to the surface.

We express Newton's second law of motion for the control volume as

$$(\text{Mass})\begin{pmatrix}\text{Acceleration}\\ \text{in a specified direction}\end{pmatrix} = \begin{pmatrix}\text{Net force (body and surface)}\\ \text{acting in that direction}\end{pmatrix} \tag{6-22}$$

or

$$\delta m \cdot a_x = F_{\text{surface},\,x} + F_{\text{body},\,x} \tag{6-23}$$

where the mass of the fluid element within the control volume is

$$\delta m = \rho(dx \cdot dy \cdot 1) \tag{6-24}$$

Noting that flow is steady and two-dimensional and thus $u = u(x, y)$, the total differential of u is

$$du = \frac{\partial u}{\partial x} dx + \frac{\partial u}{\partial y} dy \tag{6-25}$$

Then the acceleration of the fluid element in the x direction becomes

$$a_x = \frac{du}{dt} = \frac{\partial u}{\partial x}\frac{dx}{dt} + \frac{\partial u}{\partial y}\frac{dy}{dt} = u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} \tag{6-26}$$

You may be tempted to think that acceleration is zero in steady flow since acceleration is the rate of change of velocity with time, and in steady flow there is no change with time. Well, a garden hose nozzle will tell us that this understanding is not correct. Even in steady flow and thus constant mass flow rate, water will accelerate through the nozzle (Fig. 6–21). *Steady* simply means no change with time at a specified location (and thus $\partial u/\partial t = 0$), but the value of a quantity may change from one location to another (and thus $\partial u/\partial x$ and $\partial u/\partial y$ may be different from zero). In the case of a nozzle, the velocity of water remains constant at a specified point, but it changes from inlet to the exit (water accelerates along the nozzle, which is the reason for attaching a nozzle to the garden hose in the first place).

The forces acting on a surface are due to pressure and viscous effects. In two-dimensional flow, the *viscous stress* at any point on an imaginary surface within the fluid can be resolved into two perpendicular components: one normal to the surface called *normal stress* (which should not be confused with pressure) and another along the surface called *shear stress.* The normal stress is related to the velocity gradients $\partial u/\partial x$ and $\partial v/\partial y$, that are much smaller than $\partial u/\partial y$, to which shear stress is related. Neglecting the normal stresses for simplicity, the surface forces acting on the control volume in the x-direction will be as shown in Fig. 6–22. Then the net surface force acting in the x-direction becomes

$$F_{\text{surface},\,x} = \left(\frac{\partial \tau}{\partial y} dy\right)(dx \cdot 1) - \left(\frac{\partial P}{\partial x} dx\right)(dy \cdot 1) = \left(\frac{\partial \tau}{\partial y} - \frac{\partial P}{\partial x}\right)(dx \cdot dy \cdot 1)$$
$$= \left(\mu\frac{\partial^2 u}{\partial y^2} - \frac{\partial P}{\partial x}\right)(dx \cdot dy \cdot 1) \tag{6-27}$$

since $\tau = \mu(\partial u/\partial y)$. Substituting Eqs. 6-21, 6-23, and 6-24 into Eq. 6-20 and dividing by $dx \cdot dy \cdot 1$ gives

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = \mu\frac{\partial^2 u}{\partial y^2} - \frac{\partial P}{\partial x} \tag{6-28}$$

This is the relation for the **conservation of momentum** in the x-direction, and is known as the ***x*-momentum equation.** Note that we would obtain the same result if we used momentum flow rates for the left-hand side of this equation instead of mass times acceleration. If there is a body force acting in the x-direction, it can be added to the right side of the equation provided that it is expressed per unit volume of the fluid.

In a boundary layer, the velocity component in the flow direction is much larger than that in the normal direction, and thus $u \gg v$, and $\partial v/\partial x$ and $\partial v/\partial y$ are

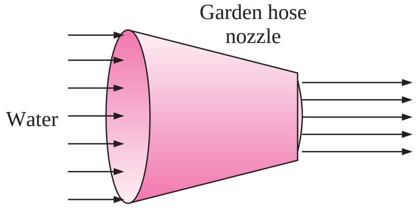


FIGURE 6–21
During steady flow, a fluid may not accelerate in time at a fixed point, but it may accelerate in space.

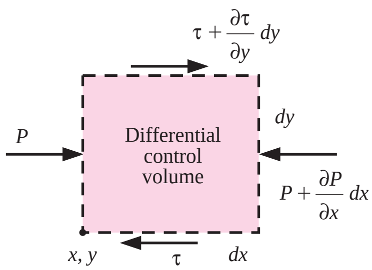
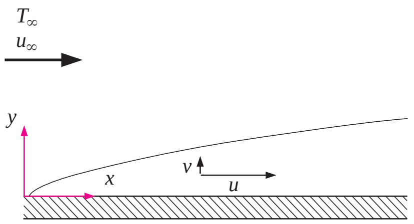


FIGURE 6–22
Differential control volume used in the derivation of x-momentum equation in velocity boundary layer in two-dimensional flow over a surface.



1) Velocity components:
$v \ll u$
2) Velocity grandients:
$\frac{\partial v}{\partial x} \approx 0, \frac{\partial v}{\partial y} \approx 0$
$\frac{\partial u}{\partial x} \ll \frac{\partial u}{\partial y}$
3) Temperature gradients:
$\frac{\partial T}{\partial x} \ll \frac{\partial T}{\partial y}$

FIGURE 6–23
Boundary layer approximations.

negligible. Also, u varies greatly with y in the normal direction from zero at the wall surface to nearly the free-stream value across the relatively thin boundary layer, while the variation of u with x along the flow is typically small. Therefore, $\partial u/\partial y \gg \partial u/\partial x$. Similarly, if the fluid and the wall are at different temperatures and the fluid is heated or cooled during flow, heat conduction will occur primarily in the direction normal to the surface, and thus $\partial T/\partial y \gg \partial T/\partial x$. That is, the velocity and temperature gradients normal to the surface are much greater than those along the surface. These simplifications are known as the **boundary layer approximations.** These approximations greatly simplify the analysis usually with little loss in accuracy, and make it possible to obtain analytical solutions for certain types of flow problems (Fig. 6–23).

When gravity effects and other body forces are negligible and the boundary layer approximations are valid, applying Newton's second law of motion on the volume element in the y-direction gives the *y-momentum equation* to be

$$\frac{\partial P}{\partial y} = 0 \tag{6-29}$$

That is, *the variation of pressure in the direction normal to the surface is negligible,* and thus $P = P(x)$ and $\partial P/\partial x = dP/dx$. Then it follows that for a given x, the pressure in the boundary layer is equal to the pressure in the free stream, and the pressure determined by a separate analysis of fluid flow in the free stream (which is typically easier because of the absence of viscous effects) can readily be used in the boundary layer analysis.

The velocity components in the free stream region of a flat plate are $u = u_\infty$ = constant and $v = 0$. Substituting these into the x-momentum equations (Eq. 6-28) gives $\partial P/\partial x = 0$. Therefore, for flow over a flat plate, the pressure remains constant over the entire plate (both inside and outside the boundary layer).

Conservation of Energy Equation

The energy balance for any system undergoing any process is expressed as $E_{in} - E_{out} = \Delta E_{system}$, which states that the change in the energy content of a system during a process is equal to the difference between the energy input and the energy output. During a *steady-flow process,* the total energy content of a control volume remains constant (and thus $\Delta E_{system} = 0$), and the amount of energy entering a control volume in all forms must be equal to the amount of energy leaving it. Then the rate form of the general energy equation reduces for a steady-flow process to $\dot{E}_{in} - \dot{E}_{out} = 0$.

Noting that energy can be transferred by heat, work, and mass only, the energy balance for a steady-flow control volume can be written explicitly as

$$(\dot{E}_{in} - \dot{E}_{out})_{by\ heat} + (\dot{E}_{in} - \dot{E}_{out})_{by\ work} + (\dot{E}_{in} - \dot{E}_{out})_{by\ mass} = 0 \tag{6-30}$$

The total energy of a flowing fluid stream per unit mass is $e_{stream} = h + \text{ke} + \text{pe}$ where h is the enthalpy (which is the sum of internal energy and flow energy), $\text{pe} = gz$ is the potential energy, and $\text{ke} = \mathcal{V}^2/2 = (u^2 + v^2)/2$ is the kinetic energy of the fluid per unit mass. The kinetic and potential energies are usually very small relative to enthalpy, and therefore it is common practice to neglect them (besides, it can be shown that if kinetic energy is included in the analysis below, all the terms due to this inclusion cancel each other). We assume the density ρ, specific heat C_p, viscosity μ, and the thermal conductivity k of the fluid to be constant. Then the energy of the fluid per unit mass can be expressed as $e_{stream} = h = C_pT$.

Energy is a scalar quantity, and thus energy interactions in all directions can be combined in one equation. Noting that mass flow rate of the fluid entering the control volume from the left is $\rho u(dy \cdot 1)$, the rate of energy transfer to the control volume by mass in the x-direction is, from Fig. 6–24,

$$
\begin{aligned}
(\dot{E}_{in} - \dot{E}_{out})_{by\ mass,\ x} &= (\dot{m}e_{stream})_x - \left[(\dot{m}e_{stream})_x + \frac{\partial(\dot{m}e_{stream})_x}{\partial x}dx\right] \\
&= -\frac{\partial[\rho u(dy \cdot 1)C_pT]}{\partial x}dx = -\rho C_p\left(u\frac{\partial T}{\partial x} + T\frac{\partial u}{\partial x}\right)dx\,dy \qquad \text{(6-31)}
\end{aligned}
$$

Repeating this for the y-direction and adding the results, the net rate of energy transfer to the control volume by mass is determined to be

$$
\begin{aligned}
(\dot{E}_{in} - \dot{E}_{out})_{by\ mass} &= -\rho C_p\left(u\frac{\partial T}{\partial x} + T\frac{\partial u}{\partial x}\right)dx\,dy - \rho C_p\left(v\frac{\partial T}{\partial y} + T\frac{\partial v}{\partial y}\right)dx\,dy \\
&= -\rho C_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right)dx\,dy \qquad \text{(6-32)}
\end{aligned}
$$

since $\partial u/\partial x + \partial v/\partial y = 0$ from the continuity equation.

The net rate of heat conduction to the volume element in the x-direction is

$$
\begin{aligned}
(\dot{E}_{in} - \dot{E}_{out})_{by\ heat,\ x} &= \dot{Q}_x - \left(\dot{Q}_x + \frac{\partial \dot{Q}_x}{\partial x}dx\right) \\
&= -\frac{\partial}{\partial x}\left(-k(dy \cdot 1)\frac{\partial T}{\partial x}\right)dx = k\frac{\partial^2 T}{\partial x^2}dx\,dy \qquad \text{(6-33)}
\end{aligned}
$$

Repeating this for the y-direction and adding the results, the net rate of energy transfer to the control volume by heat conduction becomes

$$
(\dot{E}_{in} - \dot{E}_{out})_{by\ heat} = k\frac{\partial^2 T}{\partial x^2}dx\,dy + k\frac{\partial^2 T}{\partial y^2}dx\,dy = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right)dx\,dy \qquad \text{(6-34)}
$$

Another mechanism of energy transfer to and from the fluid in the control volume is the work done by the body and surface forces. The work done by a body force is determined by multiplying this force by the velocity in the direction of the force and the volume of the fluid element, and this work needs to be considered only in the presence of significant gravitational, electric, or magnetic effects. The surface forces consist of the forces due to fluid pressure and the viscous shear stresses. The work done by pressure (the flow work) is already accounted for in the analysis above by using enthalpy for the microscopic energy of the fluid instead of internal energy. The shear stresses that result from viscous effects are usually very small, and can be neglected in many cases. This is especially the case for applications that involve low or moderate velocities.

Then the energy equation for the steady two-dimensional flow of a fluid with constant properties and negligible shear stresses is obtained by substituting Eqs. 6-32 and 6-34 into 6-30 to be

$$
\rho C_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \qquad \text{(6-35)}
$$

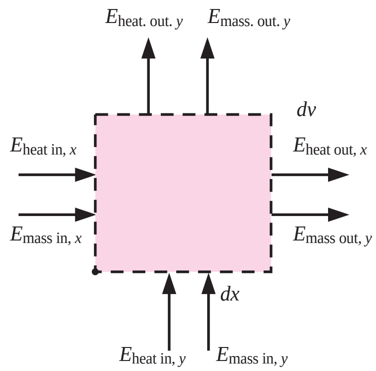


FIGURE 6–24
The energy transfers by heat and mass flow associated with a differential control volume in the thermal boundary layer in steady two-dimensional flow.

which states that *the net energy convected by the fluid out of the control volume is equal to the net energy transferred into the control volume by heat conduction.*

When the viscous shear stresses are not negligible, their effect is accounted for by expressing the energy equation as

$$\rho C_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) + \mu\Phi \tag{6-36}$$

where the *viscous dissipation function* Φ is obtained after a lengthy analysis (see an advanced book such as the one by *Schlichting* (Ref. 9) for details) to be

$$\Phi = 2\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2\right] + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 \tag{6-37}$$

Viscous dissipation may play a dominant role in high-speed flows, especially when the viscosity of the fluid is high (like the flow of oil in journal bearings). This manifests itself as a significant rise in fluid temperature due to the conversion of the kinetic energy of the fluid to thermal energy. Viscous dissipation is also significant for high-speed flights of aircraft.

For the special case of a stationary fluid, $u = v = 0$ and the energy equation reduces, as expected, to the steady two-dimensional heat conduction equation,

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} = 0 \tag{6-38}$$

EXAMPLE 6–1 Temperature Rise of Oil in a Journal Bearing

The flow of oil in a journal bearing can be approximated as parallel flow between two large plates with one plate moving and the other stationary. Such flows are known as Couette flow.

Consider two large isothermal plates separated by 2-mm-thick oil film. The upper plates moves at a constant velocity of 12 m/s, while the lower plate is stationary. Both plates are maintained at 20°C. (*a*) Obtain relations for the velocity and temperature distributions in the oil. (*b*) Determine the maximum temperature in the oil and the heat flux from the oil to each plate (Fig. 6–25).

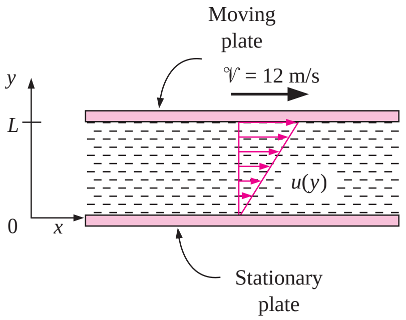


FIGURE 6–25
Schematic for Example 6–1.

SOLUTION Parallel flow of oil between two plates is considered. The velocity and temperature distributions, the maximum temperature, and the total heat transfer rate are to be determined.

Assumptions **1** Steady operating conditions exist. **2** Oil is an incompressible substance with constant properties. **3** Body forces such as gravity are negligible. **4** The plates are large so that there is no variation in the *z* direction.

Properties The properties of oil at 20°C are (Table A-10):

$$k = 0.145 \text{ W/m}\cdot\text{K} \quad \text{and} \quad \mu = 0.800 \text{ kg/m}\cdot\text{s} = 0.800 \text{ N}\cdot\text{s/m}^2$$

Analysis (*a*) We take the *x-axis* to be the flow direction, and *y* to be the normal direction. This is parallel flow between two plates, and thus $v = 0$. Then the continuity equation (Eq. 6-21) reduces to

Continuity:
$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \rightarrow \frac{\partial u}{\partial x} = 0 \quad \rightarrow u = u(y)$$