Chapter 8: Heat Exchangers

Section 8.1: Introduction to Heat Exchangers

8.1-1 (8-1 in text) Dry air at $T_{a,in}$ = 30°C, and atmospheric pressure is blown at $\dot{V}_a$ = 1.0 m^3/s through a cross-flow heat exchanger in which refrigerant R134a is evaporating at a constant pressure of p_R = 345 kPa. The air exits the heat exchanger at $T_{a,out}$ = 13°C. The tubes and fins of the heat exchanger are both made of copper. The tubes have an outer diameter of $D_{out,t}$ = 1.64 cm and th_t = 1.5 mm tube wall thickness. The fins are circular with a spacing that leads to 275 fins per meter, an outer diameter of $D_{out,f}$ = 3.1 cm and a thickness of th_f = 0.25 mm. The heat transfer coefficient between the R134a and the inner tube wall is estimated to be $\overline{h}_R$ = 2,500 W/m^2-K. The heat transfer coefficient between the air and tubes and fins is estimated to be $\overline{h}_a$ = 70 W/m^2-K. The total length of finned tubes is L = 110 m.

a) Determine the rate of heat transfer from the air.

b) Determine the value of the heat exchanger conductance for this heat exchanger.

8.1-2 (8-2 in text) The cross-flow heat exchanger described in Problem 8.1-1 (8-1 in text) has geometry similar to that for compact heat exchanger 'fc_tubes_sCF-70-58J'. The frontal area of the heat exchanger is A_f = 0.5 m^2 and the length of the heat exchanger in the flow direction is W = 0.25 m.

a.) Use the compact heat exchanger library to estimate the air-side conductance and the overall heat exchanger conductance assuming that the heat transfer coefficient between the R134a and the inner tube wall is $\overline{h}_R$ = 2,500 W/m^2-K.

b.) Compare the result to the value determined in Problem 8.1-1 (8-1).

8.1-3 (8-3 in text) A decision has been made to use chilled water, rather than R134a in the heat exchanger described in Problems 8.1-1 and 8.1-2 (8-1 and 8-2 in the text). The mass flow rate of chilled water has been chosen so that the temperature rise of the water is ΔT_w = 4°C as it passes through the heat exchanger. The water-side is configured so that the chilled water flows through N_c = 10 parallel circuits.

a.) Estimate the overall heat transfer conductance and compare the result to your answers from Problems 8.1-1 and 8.1-2 (8-1 and 8-2).

b.) Estimate how much the overall heat transfer coefficient can be expected to drop over time due to fouling of the closed chilled water loop.

Section 8.2: The Log-Mean Temperature Difference Method

8.2-1 (8-4 in text) In Problem 8.1-1 (8-1 in text), the inlet volumetric flowrate and the inlet and outlet temperatures of the air were known and therefore it was possible to determine the heat transfer rate without a heat exchanger analysis. However, you have just learned that the outlet air temperature was measured with a thermocouple in only one location in the duct and it is not necessarily an accurate measurement of the mixed average outlet air temperature. Use the log-mean temperature difference method to estimate the average air outlet temperature.

8.2-2 (8-5 in text) Table P8.2-2 provides heat transfer data from a manufacturer's catalog for a counterflow oil cooler. The table provides the heat transfer rate for three different oil flow rates (expressed in gpm, gallons per minute). The values in the table are the heat transfer rate between the oil and water in units of Btu/min-ETD where ETD is the entering temperature difference in °F. The density and specific heat of the oil are ρ_o = 830 kg/m3 and c_o = 2.3 kJ/kg-K, respectively. The water enters at $\dot{V}_w$ = 35 gallons per minute at $T_{w,in}$ = 180°F and atmospheric pressure. The oil enters at $T_{o,in}$ = 240°F.

Table P8.2-2: Heat transfer data (heat transfer rate/ETD) for different models and oil flow rates

	Oil flow rate		
	1 gpm	**3 gpm**	**5 gpm**
Model 1	2.5 Btu/min-deg. F	4.9 Btu/min-deg. F	
Model 2	2.9 Btu/min-deg. F	6.1 Btu/min-deg. F	8.1 Btu/min-deg. F
Model 3	3.1 Btu/min-deg. F	6.8 Btu/min-deg. F	9.7 Btu/min-deg. F

a.) Determine the outlet oil temperature, the log mean temperature difference, and the overall conductance for Model 2 at oil flow rates 1, 3 and 5 gallons/min.

b) Plot the conductance as a function of the oil flow rate and provide an explanation for the observed variation.

c) Estimate the oil outlet temperature if the oil enters the heat exchanger at $T_{o,in}$ = 225°F with a flow rate of $\dot{V}_o$ = 4 gpm and the water enters with temperature $T_{w,in}$ = 180°F at a flow rate of $\dot{V}_w$ = 35 gpm.

Section 8.3: The Effectiveness-NTU Method

8.3-1 (8-6 in text) The plant where you work includes a process that results in a stream of hot combustion products at moderate temperature $T_{hg,in}$ = 150°C with mass flow rate $\dot{m}_{hg}$ = 0.25 kg/s. The properties of the combustion products can be assumed to be the same as those for air. You would like to recover the energy associated with this flow in order to reduce the cost of heating the plant and therefore you are evaluating the use of the air-to-air heat exchanger shown in Figure P8.3-1.

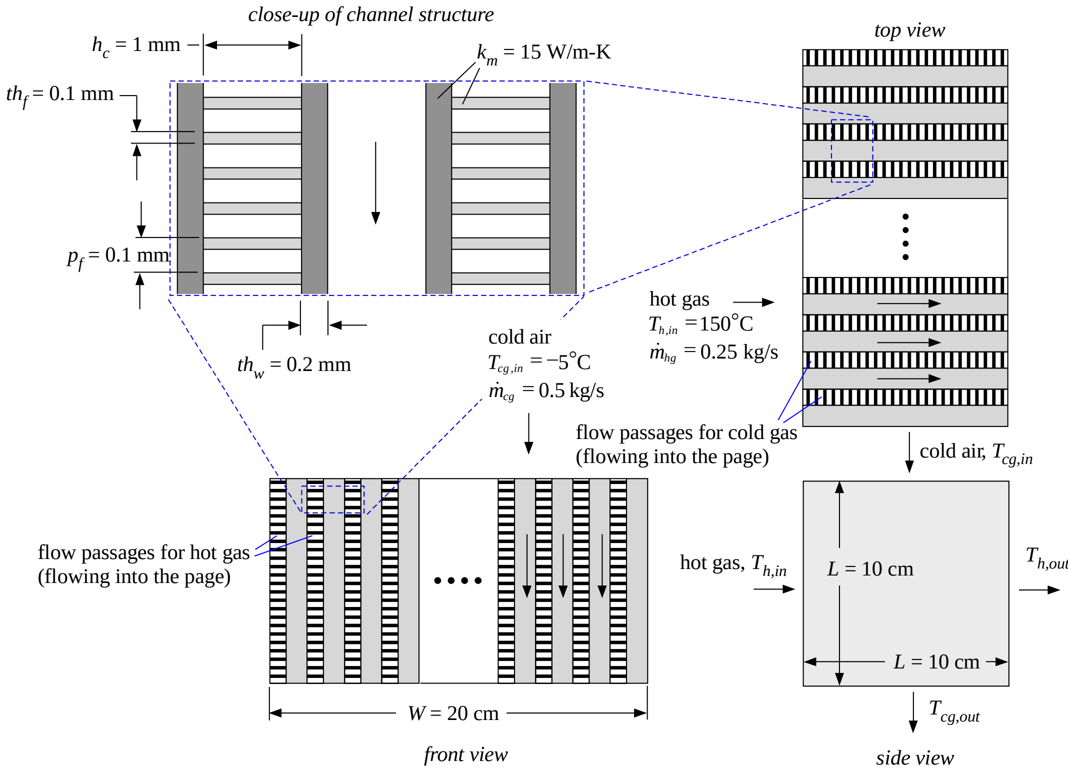


Figure P8.3-1: Air-to-air heat exchanger.

The air-to-air heat exchanger is a cross-flow configuration. The length of the heat exchanger parallel to the two flow directions is L = 10 cm. The width of the heat exchanger in the direction perpendicular to the two flow directions is W = 20 cm. Cold air enters the heat exchanger from outdoors at $T_{cg,in}$ = -5°C with mass flow rate $\dot{m}_{cg}$ = 0.50 kg/s and is heated by the combustion products to $T_{cg,out}$. The hot and cold air flows through channels that are rectangular (both sides of the heat exchanger have the same geometry). The width of the channels is h_c = 1.0 mm. There are fins placed in the channel in order to provide structural support and also increase the surface area for heat transfer. The fins can be assumed to run the complete length of the heat exchanger and are 100% efficient. The fins are spaced with pitch, p_f = 0.5 mm and the fins are th_f = 0.10 mm thick. The thickness of the metal separating the cold channels from the hot channels is th_w = 0.20 mm and the conductivity of this metal is k_m = 15 W/m-K. Both the hot and cold flows are at nominally atmospheric pressure. The fouling factor associated with the flow of combustion gas through the heat exchanger is R''_f = 0.0018 K-m^2/W. There is no fouling associated with the flow of outdoor air through the heat exchanger.

a.) Compute the heat transfer coefficient between the hot air and the channel walls and the cold gas and the channel walls. Use the inlet temperatures of the air flows to compute the properties.
b.) Compute the total conductance of the heat exchanger.
c.) Determine the heat transferred in the heat exchanger and the temperature of the cold gas leaving the heat exchanger.
d.) Blowers are required to force the hot and cold flows through the heat exchanger. Assume that you have blowers that are η_{blower} = 0.65 efficient. Estimate the total blower power required to operate the energy recovery unit.
e.) If you pay *ec* = 0.08\$/kW-hr for electricity (to run the blowers) and 1.50\$/therm for gas (to heat the plant) then estimate the net savings associated with the energy recovery system (neglect capital investment cost) for *time* = 1 year; this is the savings associated with the heat transferred in the heat exchanger less the cost to run the blower for a year. Assume that the plant runs continuously.
f.) Plot the net savings per year as a function of the mass flow rate of the cold air that is being heated. Your plot should show a maximum; explain why this maximum exists.
g.) Determine the optimal values of the mass flow rate of combustion gas and cold gas ($\dot{m}_{hg}$ and $\dot{m}_{cg}$) which maximize the net savings per year. You should use the Min/Max capability in EES to accomplish this. What is the maximum savings/year? This is the most you could afford to pay for the blowers and heat exchanger if you wanted a 1 year pay-back.

8.3-2 (8-7 in text) A gas turbine engine is used onboard a small ship to drive the propulsion system. The engine consists of a compressor, turbine, and combustor as shown in Figure P8.3-2(a). Ambient air is pulled through the gas turbine engine with a mass flow rate of $\dot{m} = 0.1$ kg/s and enters the compressor at T_1 = 20°C and P_1 = 1 atm. The exit pressure of the compressor is P_2 = 3.5 atm and T_2 = 167°C. The air enters a combustor where it is heated to T_3 = 810°C with very little loss of pressure so that P_3 = 3.5 atm. The hot air leaving the combustor enters a turbine where it is expanded to P_4 = 1 atm. The temperature of the air leaving the turbine is T_4 = 522°C. You may assume that the turbine and compressor are well-insulated and that the specific heat capacity of air is constant and equal to c = 1000 J/kg-K. The difference between the power produced by the turbine and required by the compressor is used to drive the ship.

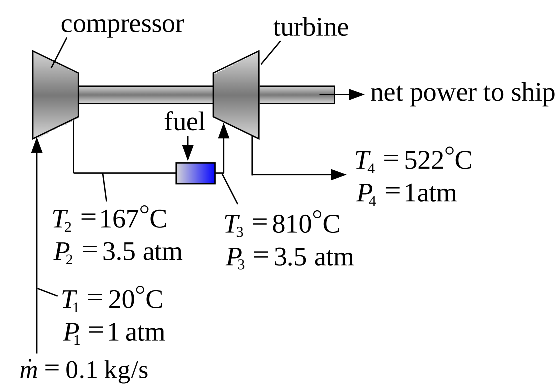


Figure P8.3-2(a): Un-recuperated gas turbine engine.

a.) Determine the efficiency of the gas turbine engine (the ratio of the net power to the ship to the heat transferred in the combustor).
b.) The combustor runs on fuel with a heating value of HV = 44x10^6 J/kg and a mission lasts t = 2 days. What is the mass of fuel that the ship must carry?

In order to reduce the amount of fuel required, you have been asked to look at the option of adding a recuperative heat exchanger to the gas turbine cycle, as shown in Figure P8.3-2(b). You are considering the air-to-air heat exchanger that was evaluated in Problem 8.3-1 (8-6 in the text) and is shown in Figure P8.3-1 (Figure P8-6 in the text). The air-to-air heat exchanger is a cross-flow configuration. The length of the heat exchanger parallel to the two flow directions is L = 10 cm. The width of the heat exchanger in the direction perpendicular to the two flow directions is W = 20 cm but this can easily be adjusted by adding additional plates. Air enters the heat exchanger from the compressor and is heated by the air leaving the turbine. The hot and cold air flows through channels that are rectangular (both sides of the heat exchanger have the same geometry). The width of the channels is h_c = 1.0 mm. There are fins placed in the channel in order to provide structural support and also increase the surface area for heat transfer. The fins can be assumed to run the complete length of the heat exchanger and are 100% efficient. The fins are spaced with pitch, p_f = 0.5 mm and the fins are th_f = 0.10 mm thick. The thickness of the metal separating the cold channels from the hot channels is th_w = 0.20 mm and the conductivity of this metal is k_m = 15 W/m-K. The hot and cold flows are at nominally atmospheric pressure and the compressor discharge pressure, respectively.

The fouling factor associated with the flow of the combustion products leaving the turbine is R''_f = 0.0018 K-m^2/W. There is no fouling associated with the flow of the air leaving the compressor.

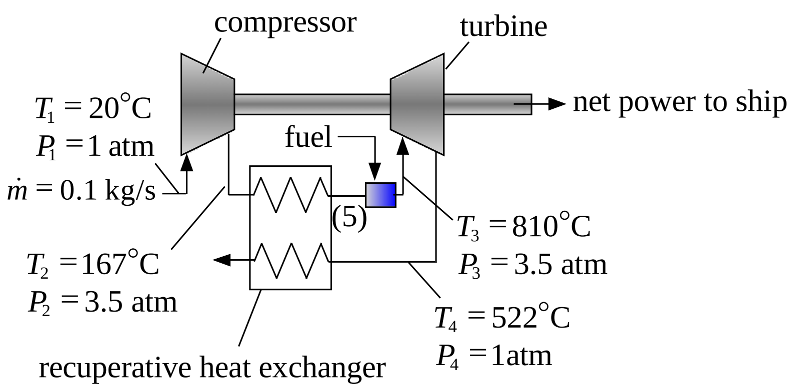


Figure P8.3-2(b): Recuperated gas turbine engine.

c.) Compute the heat transfer coefficient between the hot air from the turbine and the channel walls and the colder air from the compressor and the channel walls. You may use the inlet temperatures of the air flows to compute the properties.
d.) Compute the total conductance of the heat exchanger.
e.) Determine the heat transferred in the heat exchanger and the temperature of the air entering the combustor.
f.) What is the efficiency of the recuperated gas turbine engine?
g.) What is the mass of fuel that must be carried by the ship for the 2 day mission if it uses a recuperated gas turbine engine?
h.) The density of the metal separating the cold channels from the hot channels is ρ_m = 8000 kg/m^3 and the density of the fins is ρ_f = 7500 kg/m^3. What is the mass of the heat exchanger?
i.) What is the net savings in mass associated with using the air-to-air recuperated gas turbine engine for the 2 day mission.
j.) Plot the net savings in mass as a function of the width of the heat exchanger (W). Your plot should show a maximum; explain why.

8.3-4 (8-8 in text) Buildings that have high ventilation rates can significantly reduce their heating load by recovering energy from the exhaust air stream. One way in which this can be done is by use of a run-around loop shown in Figure P8.3-4. As shown in the figure, a run-around loop consists of two conventional liquid to air cross-flow heat exchangers. An ethylene glycol solution with 35% mass percent glycol is pumped at a rate $\dot{m}_g$ = 1 kg/s through both heat exchangers. The specific heat of this glycol solution is c_g = 3.58 kJ/kg-K. (Note that the properties of glycol solutions can be determined using the brineprop2 function in EES). During winter operation, the glycol solution is heated by the warm air exiting in the exhaust duct. The warm glycol solution is then used to preheat cold air entering from outdoors through the ventilation duct.

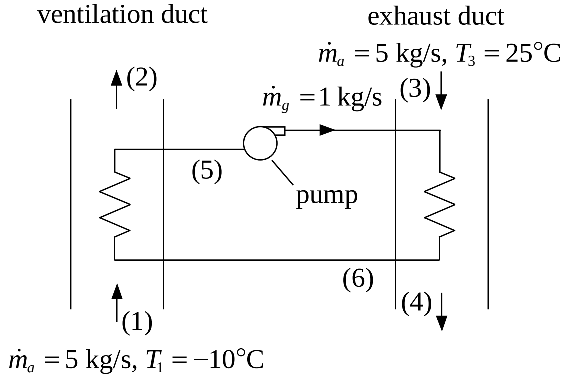


Figure P8.3-4: Run-Around loop for energy recovery

In the present case, outdoor air is blown into the building at a rate of $\dot{m}_a = 5$ kg/s from outdoors. The outdoor temperature is T_1 = -10°C. The building is tightly constructed so the exhaust air flow rate may be assumed to be equal to the ventilation air flow rate ($\dot{m}_a = 5$ kg/s). The air leaving the building through the exhaust duct is at T_3 = 25°C. The cross-flow heat exchangers in the exhaust and ventilation streams are identical, each having a finned coil configuration and an estimated conductance UA = 10 kW/K.

a.) Determine the effectiveness of the ventilation and exhaust heat exchangers.
b.) Determine the temperatures of the glycol solution at states (5) and (6).
c.) Determine the overall effectiveness of the run-around loop.
d.) It has been suggested that the performance of the run-around loop can be improved by optimizing the glycol flow rate. Plot the run-around loop overall effectiveness as a function of the glycol solution flow rate for 0.1 kg/s < $\dot{m}_g$ < 4 kg/s. Assume that the conductance of the heat exchangers vary with glycol solution flow rate to the 0.4 power based on a value of UA = 10 kW/K at $\dot{m}_g$ = 1 kg/s. What flow rate do you recommend?

Problem 8.3-9 (8-9 in text)

A concentric tube heat exchanger is built and operated as shown in Figure P8.3-9. The hot stream is a heat transfer fluid with specific heat capacity c_H = 2.5 kJ/kg-K. The hot stream enters at the center of the annulus at $T_{H,in}$ = 110°C with mass flow rate $\dot{m}_H = 0.64$ kg/s and then splits and an equal amount flows in both directions. The cold stream has specific heat capacity c_C = 4.0 kJ/kg-K. The cold enters the center pipe at $T_{C,in}$ = 10°C with mass flow rate $\dot{m}_C = 0.2$ kg/s. The outlet temperature of the hot-fluid that flows towards the left is $T_{H,out.x=0}$ = 45°C. The two sections of the heat exchanger have the same conductance.

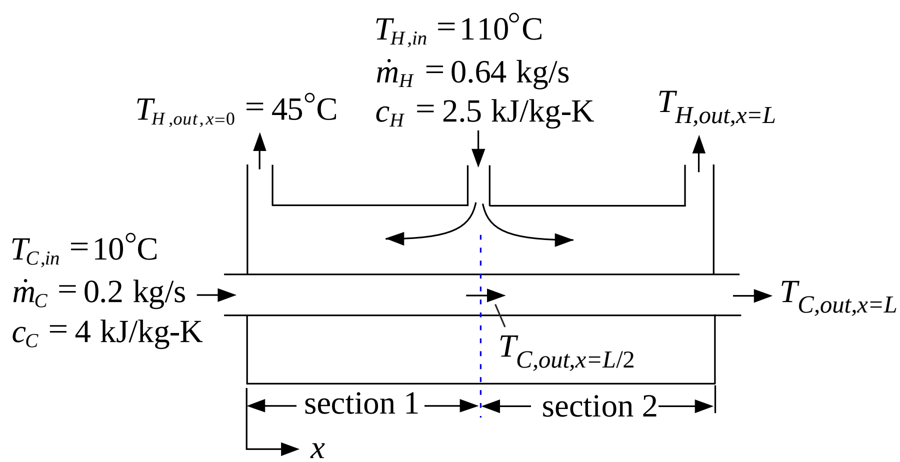


Figure P8.3-9: Concentric tube heat exchanger.

a.) Determine the temperature of the cold stream at the midpoint of the center tube and the temperature of the cold stream leaving the heat exchanger.
b.) Calculate the overall effectiveness of this heat exchanger considering both sections.
c.) How will the overall effectiveness be affected if the inlet temperature is increased to 400°C. (Assume that the properties of the heat transfer fluid are independent of temperature.)
d.) Is the overall effectiveness of this heat exchanger higher, lower, or the same as a counter-flow heat exchanger having the same inlet conditions? Justify your answer.

8.3-12 (8-10 in text) The power delivered to the wheels of a vehicle ($\dot{w}$) as a function of vehicle speed (V) is given by: $\dot{w} = -0.3937\left[\text{hp}\right] + 0.6300\left[\frac{\text{hp}}{\text{mph}}\right]V + 0.01417\left[\frac{\text{hp}}{\text{mph}^2}\right]V^2$ where power is in horsepower and velocity is in mph. The amount of heat rejected from the engine block ($\dot{q}_b$) is approximately equal to the amount of power delivered to the wheel (the rest of the energy from the fuel leaves with the exhaust gas). The heat is removed from the engine by pumping water through the engine block with a mass flow rate of $\dot{m}$ = 0.80 kg/s. The thermal communication between the engine block and the cooling water is very good, therefore you may assume that the water will exit the engine block at the engine block temperature (T_b). For the purpose of this problem you may model the water as having constant properties that are consistent with liquid water at 70°C. The heat is rejected from the water to the surrounding air using a radiator, as shown in Figure P8.3-12. When the car is moving, air is forced through the radiator due to the dynamic pressure associated with the relative motion of the car with respect to the air. That is, the air is forced through the radiator by a pressure difference that is equal to equal to $\rho_a V^2/2$ where ρ_a is the density of air. Assume that the temperature of the ambient air is T_∞ = 35°C and model the air in the radiator assuming that it has constant properties consistent with this temperature.

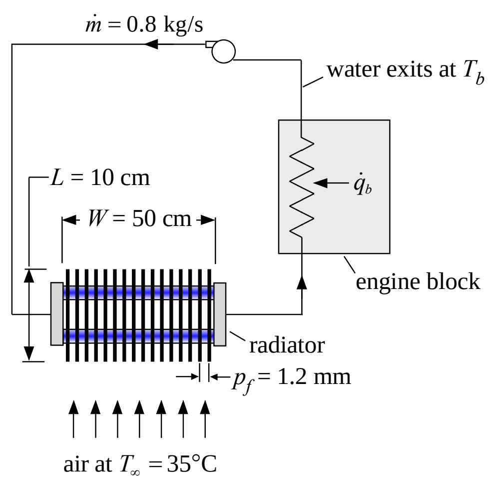


Figure P8.3-12: Engine block and radiator.

The radiator has a plate-fin geometry. There are a series of tubes installed in closely spaced metal plates that serve as fins. The fin pitch is p_f = 1.2 mm and therefore there are W/p_f plates available for heat transfer. The heat exchanger core has overall width W = 50 cm, height H = 30 cm (into the page), and length (in the flow direction) of L = 10 cm. For the purpose of modeling the air-side of the core you may assume that the air flow is consistent with an internal flow through rectangular ducts with dimension H x p_f. Assume that the fins are 100% efficient and neglect the thermal resistance between the fluid and the internal surface of the tubes. Also neglect convection from the external surfaces of the tubes as well as the reduction in the area of the plates associated with the presence of the tubes.

a.) Using the information above, develop an EES model that will allow you to predict the engine block temperature as a function of vehicle velocity. Prepare a plot showing T_b

vs V and explain the shape of the plot (if necessary, produce additional plots to help with your explanation). If the maximum allowable temperature for the engine block is 100°C (in order to prevent vaporization of the water) then what range of vehicle speeds are allowed? You should see both a minimum and maximum limit.

Its not easy to overcome the maximum speed limit identified (a); however, to overcome the minimum speed limit (so that you can pull up to a stop sign without your car overheating) you decide to add a fan. The fan can provide at most 500 cfm ($\dot{V}_o$ - the open circuit flow) and can produce at most 2.0 inch H_2O (Δp_{dh} - the dead-head pressure). The transition from open circuit to dead-head is linear. The fan curve is given by:

$$\Delta p_{fan} = \Delta p_{dh}\left(1-\frac{\dot{V}}{\dot{V}_o}\right)$$

b.) Modify your code to simulate the situation where the air is provided by the fan rather than the vehicle motion. Overlay a plot showing T_b vs V for this configuration on the one from (a); have you successfully overcome the lower speed limitation?

8.3-14 (8-11 in text) A parallel-flow heat exchanger has a total conductance UA = 10 W/K. The hot fluid enters at $T_{h,in}$ = 400 K and has a capacity rate $\dot{C}_h$ = 10 W/K. The cold fluid enters at $T_{c,in}$ = 300 K and has a capacity rate $\dot{C}_c$ = 5 W/K.

a.) Determine the number of transfer units (NTU), effectiveness (ε), heat transfer rate ($\dot{q}$), and exit temperatures ($T_{h,out}$ and $T_{c,out}$) for the heat exchanger.

b.) Sketch the temperature distribution within the heat exchanger.

c.) Sketch the temperature distribution within the heat exchanger if the conductance of the heat exchanger is very large; that is, what is the temperature distribution in the limit that $UA \rightarrow \infty$.

d.) Sketch how the hot exit temperature will change as the total conductance (UA) is varied, with all other quantities held constant at the values listed in the problem statement. Be sure to indicate how your plot behaves as UA approaches zero and as UA approaches infinity.

8.3-15 (8-12 in text) A heat exchanger has a core geometry that corresponds to finned circular tube core 'fc_tubes_s80_38T' in the compact heat exchanger library. The frontal area of the core has dimensions W = 7.75 inch and H = 7.75 inch. The length of the core is L = 1.5 inch. The core is integrated with a fan that has a head-flow curve given by: $\Delta p = a - b\dot{V}_a$ where Δp is the pressure rise across the fan, $\dot{V}_a$ is the volumetric flow rate of air, a = 0.3927 inH2O and b = 0.0021 inH2O/cfm are the coefficients of the fan curve. The manufacturer has tested the heat exchanger with atmospheric air at $T_{a,in}$ = 20ºC and water at $T_{w,in}$ = 75ºC, p_w = 65 psia flowing through the tubes. The test data are shown in Table P8.3-15. The tubes are plumbed in series (i.e., all of the water flows through each tube) and the tube thickness is th_t = 0.035 inch.

Table P8.3-15: Manufacturer's data for heat exchanger.

Water flow rate	Water outlet temperature
0.13 gpm	44.3ºC
0.25 gpm	51.1ºC
0.5 gpm	60.1ºC
1 gpm	66.9ºC
2 gpm	70.8ºC
4 gpm	72.9ºC

a.) Develop a model using the effectiveness-NTU technique that can predict the outlet temperature of the water for a water flow rate of the water $\dot{V}_w$.

b.) Plot the outlet temperature of the water as a function of the water flow rate and overlay the manufacturer's data onto your plot.

8.6-2 (8-13 in text) A Joule-Thomson refrigeration cycle is illustrated in Figure P8.6-2.

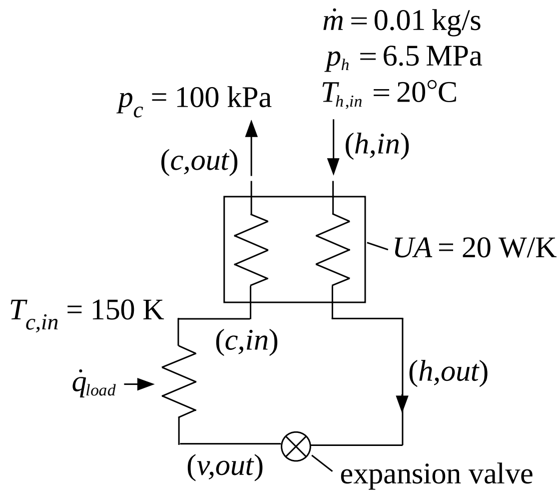


Figure P8.6-2: Joule-Thomson refrigeration cycle.

The system uses pure argon as the working fluid. High pressure argon enters a counterflow heat exchanger with mass flow rate $\dot{m}$ = 0.01 kg/s at $T_{h,in}$ = 20ºC and p_h = 6.5 MPa. The argon flows through the heat exchanger where it is pre-cooled by the low pressure argon returning from the cold end of the cycle. The high pressure argon leaving the heat exchanger enters an expansion valve where it is expanded to p_c = 100 kPa. The argon passes through a load heat exchanger where it accepts a refrigeration load, $\dot{q}_{load}$, and it heated to $T_{c,in}$ = 150 K. The conductance of the heat exchanger is UA= 20 W/K. Neglect pressure loss in the heat exchanger on both the hot and cold sides of the heat exchanger.

a.) Use the effectiveness-*NTU* method to estimate the effectiveness of the heat exchanger and the rate of heat transferred from the hot to the cold stream in the heat exchanger. Calculate the specific heat capacity of the high- and low-pressure argon using the average of the hot and cold inlet temperatures.

b.) Determine the refrigeration load provided by the cycle.

c.) Prepare a plot of refrigeration load as a function of cold inlet temperature for 85 K < $T_{c,in}$ < 290 K and various values of the conductance. A negative refrigeration load is not physically possible (without some external cooling); therefore, terminate your plots at $\dot{q}_{load}$ = 0 W.

d.) Instead of using the effectiveness-*NTU* method, divide the heat exchanger into sub-heat exchangers as discussed in Section 8.6.3. What is the heat transferred in the heat exchanger for the conditions listed in the problem statement?

e.) Determine the refrigeration load associated with your prediction from (d).

f.) Overlay on your plot from (c) the refrigeration load as a function of cold inlet temperature for the same values of the conductance.

8.6-3 (8-14 in text) A counter-flow heat exchanger has a total conductance of UA = 130 W/K. Air flows on the hot side. The air enters at $T_{h,in}$ = 500 K with pressure p_h = 1 atm and mass flow rate $\dot{m}_h$ = 0.08 kg/s. Carbon dioxide flows on the cold side. The CO_2 enters at $T_{c,in}$ = 300 K with pressure p_c = 80 atm and mass flow rate $\dot{m}_c$ = 0.02 kg/s.

a.) Plot the specific heat capacity of air at 1 atm and carbon dioxide at 80 atm and comment on whether the ε-*NTU* solution can be applied to this heat exchanger.

b.) Prepare a solution to this problem by numerically integrating the governing equations using the Euler technique, as discussed in Section 8.6.2.

c.) Using your solution from (b), plot the temperature of the carbon dioxide and air as a function of the dimensionless axial position (x/L).

d.) Plot the rate of heat transfer predicted by the model as a function of the number of integration steps.

e.) Prepare a solution to this problem by sub-dividing the heat exchanger into sub-heat exchangers, as discussed in Section 8.6.3.

f.) Overlay on your plot from (c) the temperature distribution predicted by your model from (e).

g.) Overlay on your plot from (d) the rate of heat transfer predicted by your model from (e) as a function of the number of sub-heat exchangers.

Section 8.10: Regenerators

8.10-1 (8-15 in text) A solar heating system is shown in Figure P8.10-1.

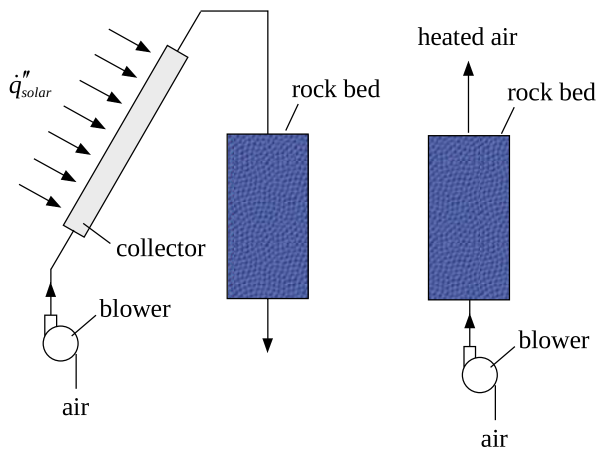


Figure P8.10-1: Solar heating system during charging and during discharging.

During the day, the solar heat is not required and therefore air is blown through a series of solar collectors where it is heated as shown in Figure P8.10-1. The thermal energy is stored in a large rock bed regenerator. The rock bed is L= 20 ft long in the flow direction and W = 10 ft x W = 10 ft in cross-sectional area. The bed is filled with D_p = 0.5 inch diameter rocks with density ρ_r = 100 lb_m/ft^3 and specific heat capacity c_r = 0.2 Btu/lb_m-°F. The charging process goes on for t_{charge} =12 hr. There are N_{col} = 40 solar collectors, each with area 8 ft x 4 ft. During the charging process, atmospheric air at T_{indoor} = 70°F enters the collectors where it is heated by the solar irradiation $\dot{q}''_{solar}$ = 750 W/m^2. The efficiency of the collector is given by: $\eta_{collector}$ = 0.75 - 0.0015 [K^{-1}] ($T_{r,in}$ - $T_{outdoor}$) where $T_{outdoor}$ = 10°F and $T_{r,in}$ is the temperature of the air leaving the collector and entering the regenerator. The collector efficiency is the ratio of the energy transferred to the air to the energy incident on the collector. During the night, the energy that was stored in the rock bed is used to provide heating, as shown in Figure P8.10-1. Air at T_{indoor} = 70°F enters the rock bed where it is heated. The hot air is provided to the building. The blower used during both the charging and discharging process has an efficiency of η_b = 0.6 and a pressure/flow curve that goes linearly from Δp_{dh} = 0.5 inch of water at zero flow to $\dot{V}_{open}$ = 1800 cfm at zero pressure rise. Neglect the pressure drop across the collectors and assume that the pressure drop that must be overcome by the blower is related to the flow through the rock bed. The porosity of the rock bed is ϕ = 0.35 and assume that the rock bed is well-insulated.

a.) What is the temperature of the air entering the rock bed during the charging process and the mass flow rate of air during the charging and discharging process?
b.) What is the amount of heat transfer from the rock bed to the air during the discharge process?
c.) There are 100 heating days per year in this location. What is the total amount of heating energy saved over a 10 year period?
d.) If the cost of natural gas is gc = 3.5\$/therm then what is the total heating cost saved over a 10 year period? (Neglect the time value of money for this analysis.)

e.) The cost of the solar collectors is cc = 45\$/ft^2 and the cost of the rock bed is rc = 40\$/ton. The cost of the electrical energy required to run the blowers is ec = 0.12\$/kW-hr. Determine the net savings associated with owning the equipment over a 10 year period.

f.) Plot the net savings as a function of the number of solar collectors. You should see that an optimal number of collectors exists. Provide an explanation for this observation.

g.) Plot the net savings as a function of the length of the rock bed (with N_{col} = 40). You should see that an optimal length of the rock bed. Explain this fact.

h.) Determine the optimal number of collectors and rock bed length.

Chapter 8: Heat Exchangers

Section 8.1: Introduction to Heat Exchangers

8.1-1 (8-1 in text) Dry air at $T_{a,in}$ = 30°C, and atmospheric pressure is blown at $\dot{V}_a$ = 1.0 m^3/s through a cross-flow heat exchanger in which refrigerant R134a is evaporating at a constant pressure of p_R = 345 kPa. The air exits the heat exchanger at $T_{a,out}$ = 13°C. The tubes and fins of the heat exchanger are both made of copper. The tubes have an outer diameter of $D_{out,t}$ = 1.64 cm and th_t = 1.5 mm tube wall thickness. The fins are circular with a spacing that leads to 275 fins per meter, an outer diameter of $D_{out,f}$ = 3.1 cm and a thickness of th_f = 0.25 mm. The heat transfer coefficient between the R134a and the inner tube wall is estimated to be $\overline{h}_R$ = 2,500 W/m^2-K. The heat transfer coefficient between the air and tubes and fins is estimated to be $\overline{h}_a$ = 70 W/m^2-K. The total length of finned tubes is L = 110 m.

a) Determine the rate of heat transfer from the air.

b) Determine the value of the heat exchanger conductance for this heat exchanger.

8.1-2 (8-2 in text) The cross-flow heat exchanger described in Problem 8.1-1 (8-1 in text) has geometry similar to that for compact heat exchanger 'fc_tubes_sCF-70-58J'. The frontal area of the heat exchanger is A_f = 0.5 m^2 and the length of the heat exchanger in the flow direction is W = 0.25 m.

a.) Use the compact heat exchanger library to estimate the air-side conductance and the overall heat exchanger conductance assuming that the heat transfer coefficient between the R134a and the inner tube wall is $\overline{h}_R$ = 2,500 W/m^2-K.

b.) Compare the result to the value determined in Problem 8.1-1 (8-1).

8.1-3 (8-3 in text) A decision has been made to use chilled water, rather than R134a in the heat exchanger described in Problems 8.1-1 and 8.1-2 (8-1 and 8-2 in the text). The mass flow rate of chilled water has been chosen so that the temperature rise of the water is ΔT_w = 4°C as it passes through the heat exchanger. The water-side is configured so that the chilled water flows through N_c = 10 parallel circuits.

a.) Estimate the overall heat transfer conductance and compare the result to your answers from Problems 8.1-1 and 8.1-2 (8-1 and 8-2).

b.) Estimate how much the overall heat transfer coefficient can be expected to drop over time due to fouling of the closed chilled water loop.

8.1-4 A heat exchanger core consists of finned flat tubes with geometry that is consistent with ff_tubes_s91-0737-S. The frontal area of the core has width W= 5 ft and height H = 3 ft. The length of the core in the flow direction is L = 1.5 ft. The fan that is connected to the core provides a volumetric flow rate of $\dot{V}_{open}$ = 30,000 ft^3/min when there is no pressure rise and a pressure rise of Δp_{dh} = 2 inH_2O when there is no flow. The fan curve is linear between these points. The fan draws in air at T_{in} = 80°F and passes it across the coil.

a.) Determine the total surface area of the coil that is exposed to air.
b.) Determine the volumetric flow rate of air that will be provided to the coil.
c.) Determine the heat transfer coefficient on the air side.

Section 8.2: The Log-Mean Temperature Difference Method

8.2-1 (8-4 in text) In Problem 8.1-1 (8-1 in text), the inlet volumetric flowrate and the inlet and outlet temperatures of the air were known and therefore it was possible to determine the heat transfer rate without a heat exchanger analysis. However, you have just learned that the outlet air temperature was measured with a thermocouple in only one location in the duct and it is not necessarily an accurate measurement of the mixed average outlet air temperature. Use the log-mean temperature difference method to estimate the average air outlet temperature.

8.2-2 (8-5 in text) Table P8.2-2 provides heat transfer data from a manufacturer's catalog for a counterflow oil cooler. The table provides the heat transfer rate for three different oil flow rates (expressed in gpm, gallons per minute). The values in the table are the heat transfer rate between the oil and water in units of Btu/min-ETD where ETD is the entering temperature difference in °F. The density and specific heat of the oil are ρ_o = 830 kg/m3 and c_o = 2.3 kJ/kg-K, respectively. The water enters at $\dot{V}_w$ = 35 gallons per minute at $T_{w,in}$ = 180°F and atmospheric pressure. The oil enters at $T_{o,in}$ = 240°F.

Table P8.2-2: Heat transfer data (heat transfer rate/ETD) for different models and oil flow rates

	Oil flow rate		
	1 gpm	**3 gpm**	**5 gpm**
Model 1	2.5 Btu/min-deg. F	4.9 Btu/min-deg. F	
Model 2	2.9 Btu/min-deg. F	6.1 Btu/min-deg. F	8.1 Btu/min-deg. F
Model 3	3.1 Btu/min-deg. F	6.8 Btu/min-deg. F	9.7 Btu/min-deg. F

a.) Determine the outlet oil temperature, the log mean temperature difference, and the overall conductance for Model 2 at oil flow rates 1, 3 and 5 gallons/min.

b) Plot the conductance as a function of the oil flow rate and provide an explanation for the observed variation.

c) Estimate the oil outlet temperature if the oil enters the heat exchanger at $T_{o,in}$ = 225°F with a flow rate of $\dot{V}_o$ = 4 gpm and the water enters with temperature $T_{w,in}$ = 180°F at a flow rate of $\dot{V}_w$ = 35 gpm.

8.2-3 You are specifying a plate frame heat exchanger with a counter-flow configuration to transfer heat from hot water to cooling water. The hot water enters the heat exchanger at $T_{H,in}$ = 120ºF with a flow rate of $\dot{V}_H$ = 0.85 gal/min. The cooling water enters at $T_{C,in}$ = 35ºF with flow rate $\dot{V}_C$ = 0.65 gal/min. If the hot water must be cooled to $T_{H,out}$ = 75ºF then what is the required conductance of the heat exchanger?

Section 8.3: The Effectiveness-NTU Method

8.3-1 (8-6 in text) The plant where you work includes a process that results in a stream of hot combustion products at moderate temperature $T_{hg,in}$ = 150°C with mass flow rate $\dot{m}_{hg}$ = 0.25 kg/s. The properties of the combustion products can be assumed to be the same as those for air. You would like to recover the energy associated with this flow in order to reduce the cost of heating the plant and therefore you are evaluating the use of the air-to-air heat exchanger shown in Figure P8.3-1.

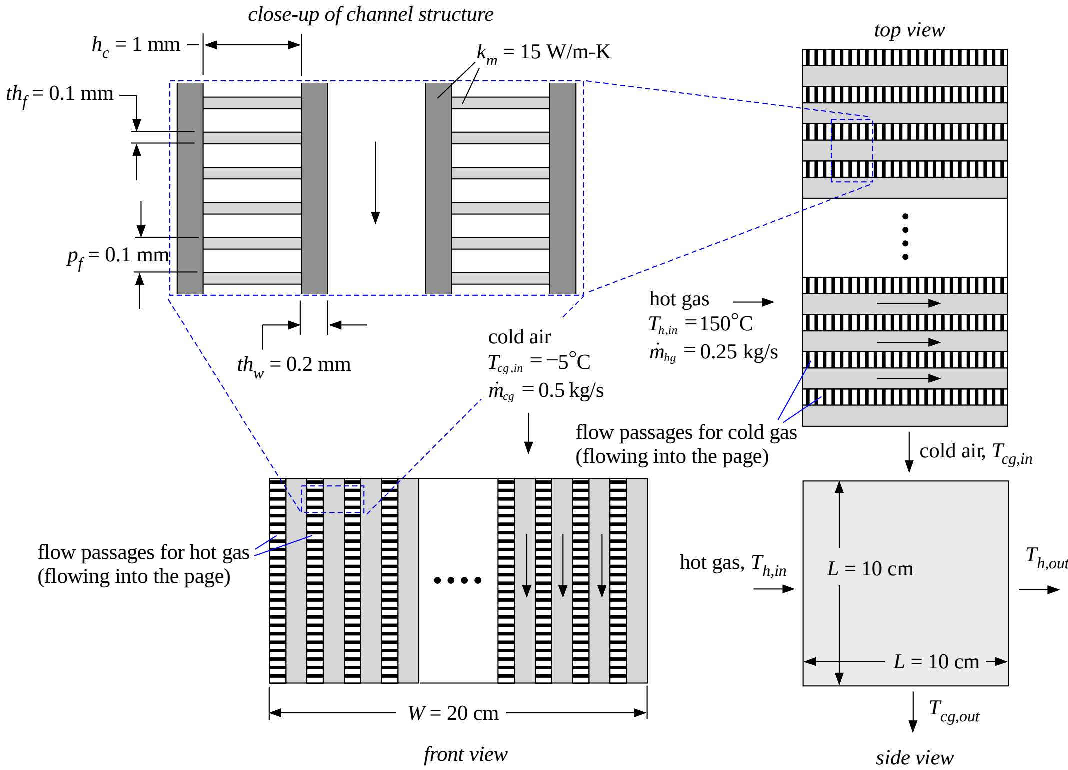


Figure P8.3-1: Air-to-air heat exchanger.

The air-to-air heat exchanger is a cross-flow configuration. The length of the heat exchanger parallel to the two flow directions is L = 10 cm. The width of the heat exchanger in the direction perpendicular to the two flow directions is W = 20 cm. Cold air enters the heat exchanger from outdoors at $T_{cg,in}$ = -5°C with mass flow rate $\dot{m}_{cg}$ = 0.50 kg/s and is heated by the combustion products to $T_{cg,out}$. The hot and cold air flows through channels that are rectangular (both sides of the heat exchanger have the same geometry). The width of the channels is h_c = 1.0 mm. There are fins placed in the channel in order to provide structural support and also increase the surface area for heat transfer. The fins can be assumed to run the complete length of the heat exchanger and are 100% efficient. The fins are spaced with pitch, p_f = 0.5 mm and the fins are th_f = 0.10 mm thick. The thickness of the metal separating the cold channels from the hot channels is th_w = 0.20 mm and the conductivity of this metal is k_m = 15 W/m-K. Both the hot and cold flows are at nominally atmospheric pressure. The fouling factor associated with the flow of combustion gas through the heat exchanger is R''_f = 0.0018 K-m^2/W. There is no fouling associated with the flow of outdoor air through the heat exchanger.

a.) Compute the heat transfer coefficient between the hot air and the channel walls and the cold gas and the channel walls. Use the inlet temperatures of the air flows to compute the properties.
b.) Compute the total conductance of the heat exchanger.
c.) Determine the heat transferred in the heat exchanger and the temperature of the cold gas leaving the heat exchanger.
d.) Blowers are required to force the hot and cold flows through the heat exchanger. Assume that you have blowers that are η_{blower} = 0.65 efficient. Estimate the total blower power required to operate the energy recovery unit.
e.) If you pay *ec* = 0.08$/kW-hr for electricity (to run the blowers) and 1.50$/therm for gas (to heat the plant) then estimate the net savings associated with the energy recovery system (neglect capital investment cost) for *time* = 1 year; this is the savings associated with the heat transferred in the heat exchanger less the cost to run the blower for a year. Assume that the plant runs continuously.
f.) Plot the net savings per year as a function of the mass flow rate of the cold air that is being heated. Your plot should show a maximum; explain why this maximum exists.
g.) Determine the optimal values of the mass flow rate of combustion gas and cold gas ($\dot{m}_{hg}$ and $\dot{m}_{cg}$) which maximize the net savings per year. You should use the Min/Max capability in EES to accomplish this. What is the maximum savings/year? This is the most you could afford to pay for the blowers and heat exchanger if you wanted a 1 year pay-back.

8.3-2 (8-7 in text) A gas turbine engine is used onboard a small ship to drive the propulsion system. The engine consists of a compressor, turbine, and combustor as shown in Figure P8.3-2(a). Ambient air is pulled through the gas turbine engine with a mass flow rate of $\dot{m} = 0.1$ kg/s and enters the compressor at T_1 = 20°C and P_1 = 1 atm. The exit pressure of the compressor is P_2 = 3.5 atm and T_2 = 167°C. The air enters a combustor where it is heated to T_3 = 810°C with very little loss of pressure so that P_3 = 3.5 atm. The hot air leaving the combustor enters a turbine where it is expanded to P_4 = 1 atm. The temperature of the air leaving the turbine is T_4 = 522°C. You may assume that the turbine and compressor are well-insulated and that the specific heat capacity of air is constant and equal to c = 1000 J/kg-K. The difference between the power produced by the turbine and required by the compressor is used to drive the ship.

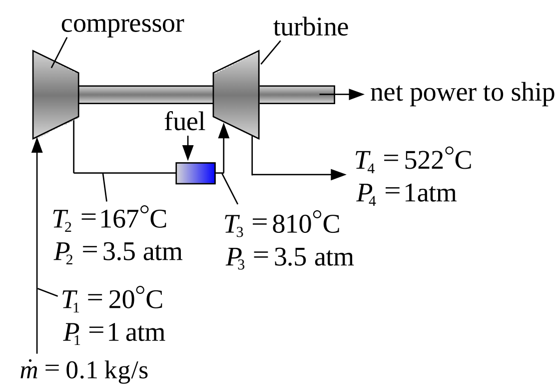


Figure P8.3-2(a): Un-recuperated gas turbine engine.

a.) Determine the efficiency of the gas turbine engine (the ratio of the net power to the ship to the heat transferred in the combustor).

b.) The combustor runs on fuel with a heating value of $HV = 44\text{x}10^6$ J/kg and a mission lasts t = 2 days. What is the mass of fuel that the ship must carry?

In order to reduce the amount of fuel required, you have been asked to look at the option of adding a recuperative heat exchanger to the gas turbine cycle, as shown in Figure P8.3-2(b). You are considering the air-to-air heat exchanger that was evaluated in Problem 8.3-1 (8-6 in the text) and is shown in Figure P8.3-1 (Figure P8-6 in the text). The air-to-air heat exchanger is a cross-flow configuration. The length of the heat exchanger parallel to the two flow directions is L = 10 cm. The width of the heat exchanger in the direction perpendicular to the two flow directions is W = 20 cm but this can easily be adjusted by adding additional plates. Air enters the heat exchanger from the compressor and is heated by the air leaving the turbine. The hot and cold air flows through channels that are rectangular (both sides of the heat exchanger have the same geometry). The width of the channels is h_c = 1.0 mm. There are fins placed in the channel in order to provide structural support and also increase the surface area for heat transfer. The fins can be assumed to run the complete length of the heat exchanger and are 100% efficient. The fins are spaced with pitch, p_f = 0.5 mm and the fins are th_f = 0.10 mm thick. The thickness of the metal separating the cold channels from the hot channels is th_w = 0.20 mm and the conductivity of this metal is k_m = 15 W/m-K. The hot and cold flows are at nominally atmospheric pressure and the compressor discharge pressure, respectively.

The fouling factor associated with the flow of the combustion products leaving the turbine is R''_f = 0.0018 K-m^2/W. There is no fouling associated with the flow of the air leaving the compressor.

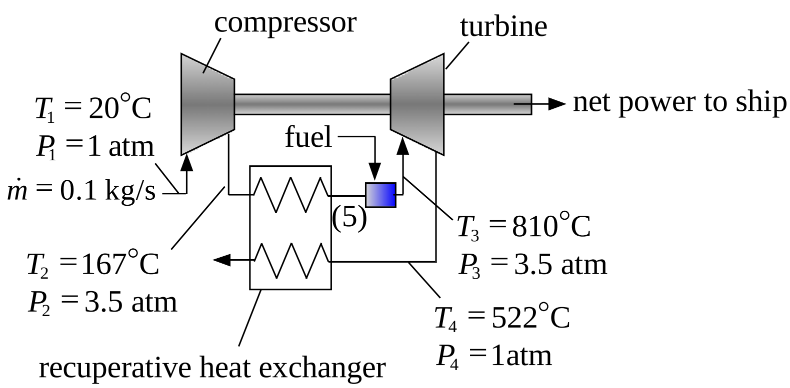


Figure P8.3-2(b): Recuperated gas turbine engine.

c.) Compute the heat transfer coefficient between the hot air from the turbine and the channel walls and the colder air from the compressor and the channel walls. You may use the inlet temperatures of the air flows to compute the properties.

d.) Compute the total conductance of the heat exchanger.

e.) Determine the heat transferred in the heat exchanger and the temperature of the air entering the combustor.

f.) What is the efficiency of the recuperated gas turbine engine?

g.) What is the mass of fuel that must be carried by the ship for the 2 day mission if it uses a recuperated gas turbine engine?

h.) The density of the metal separating the cold channels from the hot channels is ρ_m = 8000 kg/m^3 and the density of the fins is ρ_f = 7500 kg/m^3. What is the mass of the heat exchanger?

i.) What is the net savings in mass associated with using the air-to-air recuperated gas turbine engine for the 2 day mission.

j.) Plot the net savings in mass as a function of the width of the heat exchanger (W). Your plot should show a maximum; explain why.

8.3-20 You are specifying a plate frame heat exchanger with a counter-flow configuration to transfer heat from hot water to cooling water. The hot water enters the heat exchanger at $T_{H,in}$ = 120ºF with a flow rate of $\dot{V}_H$ = 0.85 gal/min. The cooling water enters at $T_{C,in}$ = 35ºF with flow rate $\dot{V}_C$ = 0.65 gal/min. If the hot water must be cooled to $T_{H,out}$ = 75ºF then what is the required conductance of the heat exchanger?

8.3-3 A parallel-flow heat exchanger is shown schematically in Figure P8.3-3(a). The heat exchanger is being used to heat air from $T_{a,in}$ = 10°C using a flow of water at T_w = 80°C. The capacity rate of the air is $\dot{C}_a$ = 0.1 W/K and the capacity rate of the water is $\dot{C}_w$ = 10 W/K. The conductance of the heat exchanger is UA = 0.1 W/K.

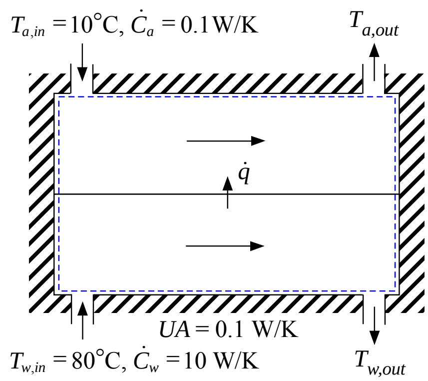


Figure P8.3-3(a): Parallel-flow heat exchanger.

a.) Sketch the temperature distributions in the heat exchanger (i.e., the temperature of the air and the water as a function of position, x); make sure that the qualitative features of your sketch is correct based on the operating conditions.
b.) Determine the heat transfer rate in the heat exchanger ($\dot{q}$) as well as the temperature of the air leaving the heat exchanger ($T_{a,out}$).
c.) Sketch the heat transfer rate ($\dot{q}$) as a function of the heat exchanger conductance (UA, assume that all other conditions remain the same as shown in Figure 1). Make sure that you indicate the heat transfer rate that you expect at UA = 0 and $UA \rightarrow \infty$.
d.) Your boss has heard that it is always better to use a counter-flow as opposed to a parallel-flow heat exchanger and so he insists that you should re-plumb the heat exchanger so that it is in a counter flow configuration; as shown in Figure P8.3-3(b). Do you expect this change to substantially improve the performance (i.e., increase the rate of heat transferred in the heat exchanger)? Clearly justify your result qualitatively.

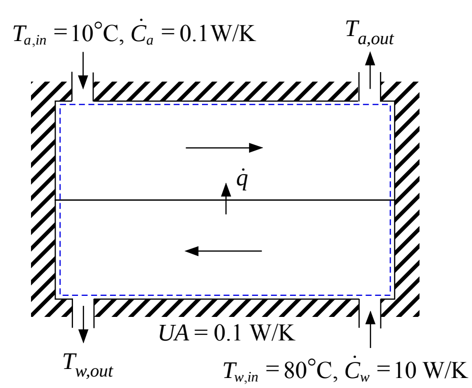


Figure P8.3-3(b): Heat exchanger re-plumbed so that it is in a counter flow configuration.

8.3-4 (8-8 in text) Buildings that have high ventilation rates can significantly reduce their heating load by recovering energy from the exhaust air stream. One way in which this can be done is by use of a run-around loop shown in Figure P8.3-4. As shown in the figure, a run-around loop consists of two conventional liquid to air cross-flow heat exchangers. An ethylene glycol solution with 35% mass percent glycol is pumped at a rate $\dot{m}_g$ = 1 kg/s through both heat exchangers. The specific heat of this glycol solution is c_g = 3.58 kJ/kg-K. (Note that the properties of glycol solutions can be determined using the brineprop2 function in EES). During winter operation, the glycol solution is heated by the warm air exiting in the exhaust duct. The warm glycol solution is then used to preheat cold air entering from outdoors through the ventilation duct.

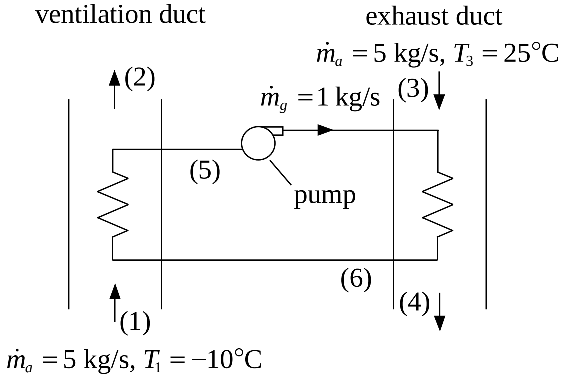


Figure P8.3-4: Run-Around loop for energy recovery

In the present case, outdoor air is blown into the building at a rate of $\dot{m}_a = 5$ kg/s from outdoors. The outdoor temperature is T_1 = -10°C. The building is tightly constructed so the exhaust air flow rate may be assumed to be equal to the ventilation air flow rate ($\dot{m}_a = 5$ kg/s). The air leaving the building through the exhaust duct is at T_3 = 25°C. The cross-flow heat exchangers in the exhaust and ventilation streams are identical, each having a finned coil configuration and an estimated conductance *UA* = 10 kW/K.

a.) Determine the effectiveness of the ventilation and exhaust heat exchangers.
b.) Determine the temperatures of the glycol solution at states (5) and (6).
c.) Determine the overall effectiveness of the run-around loop.
d.) It has been suggested that the performance of the run-around loop can be improved by optimizing the glycol flow rate. Plot the run-around loop overall effectiveness as a function of the glycol solution flow rate for 0.1 kg/s < $\dot{m}_g$ < 4 kg/s. Assume that the conductance of the heat exchangers vary with glycol solution flow rate to the 0.4 power based on a value of *UA* = 10 kW/K at $\dot{m}_g$ = 1 kg/s. What flow rate do you recommend?

8.3-5 A cryosurgical probe uses a Joule-Thomson (ienthalpic) expansion of argon gas to produce cold temperatures, as shown in Figure P8.3-5. The argon is compressed to p_2 = 200 atm and cooled at constant pressure to T_3 = 300 K where it enters a recuperative counter-flow heat exchanger. The gas is cooled to state (4) in the recuperator using the return gas that enters the recuperator at T_6 = 140 K. After leaving the recuperator at state (4), the argon is throttled to a low pressure state (5) where it used to cool tissue. The refrigeration load provided by the cryoprobe ($\dot{q}_{load}$) warms the gas to T_6 = 140 K. The argon returning to the compressor is at a pressure of p_1 = 100 kPa; pressure losses occur in the recuperator, but the pressure difference between states (5) and (6) is negligible.

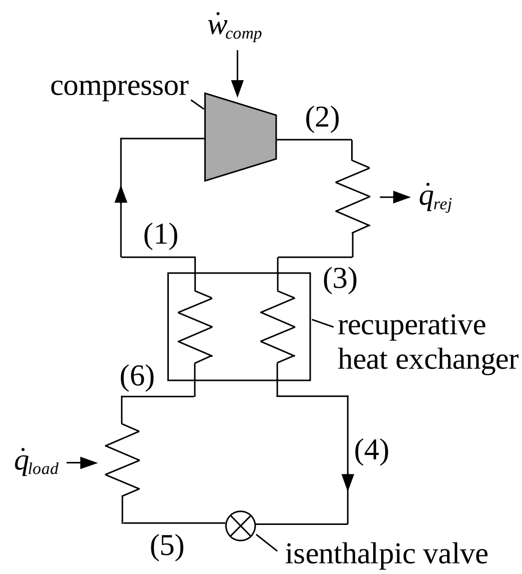
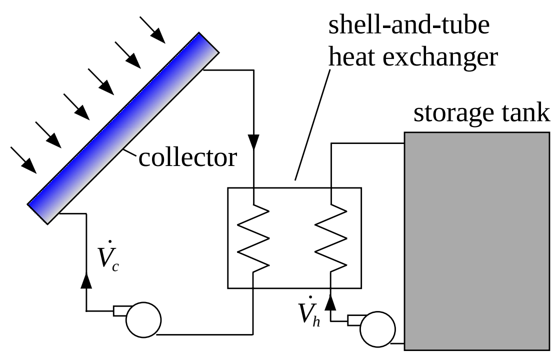


Figure 8.3-5: Joule-Thomson cycle for a cryosurgical probe.

The recuperator is a smooth thin-walled stainless steel concentric tube heat exchanger. The high pressure argon passes through the center tube and the low pressure argon returns through the annulus. The length of the tubes is L = 0.35 m. The inner diameter of the outer tube diameter is D_{out} = 1.75 mm. The wall thickness of the tube is th = 180 μm. The inner diameter of the inner tube is D_{in} = 600 μm. The mass flow rate of argon is $\dot{m}$ = 0.3 g/s.

a.) Determine the conductance of the recuperative heat exchanger.
b.) Determine the heat exchanger effectiveness.
c.) Determine the pressures at states (4) and (6).
d.) Determine the temperatures at states (1), (4), (5), and (6).
e.) Determine the refrigeration load provided by the cryoprobe, $\dot{q}_{load} = \dot{m}\left(i_6 - i_5\right)$ where i is the specific enthalpy.
f.) Plot refrigeration load as a function of mass flow rate. You should see an optimal mass flow rate, why does such an optimal flow exist?
g.) Plot the recuperator effectiveness and the refrigeration load as a function of the inner diameter of the inner tube for 400 μm < D_{in} < 1.1 mm and 0.3 g/s mass flow rate. Is there an optimum inner diameter for the mass flow rate of 0.3 g/s? Why?

8.3-6 A schematic of a solar water heating system is shown in Figure P8.3-6. When solar radiation is available, a 30% (by mass) propylene glycol solution is pumped at volumetric flow rate $\dot{V}_c$ through the collectors and through the tubes in a shell-and-tube heat exchanger. The heat exchanger is comprised of 40 tubes each having a diameter of 0.5 cm in a shell that has a diameter of 4 cm. The copper pipe walls are thin enough to neglect in this analysis. The tubes are 1.25 m in length. A second pump circulates water at volumetric flow rate $\dot{V}_h$ from the storage tank through the shell of the heat exchanger in a counter-flow direction to the glycol solution flow. The heat transfer coefficient between the water and the outer surface of the tubes has been estimated as $\bar{h}_o = 950$ W/m^2-K. A controller is used to control the volumetric flow rate of the glycol and water solutions so that there is always a 10°C temperature difference across the solar collector when solar radiation is available. (An average temperature of 50°C can be assumed to evaluate properties.) The volumetric flow rate of the water pump is always the same as that for the glycol pump. The purpose of this problem is to determine the effectiveness of the heat exchanger as a function of the volumetric flowrate.



P8.3-6 Solar water heating sytem

a.) Calculate and plot the effectiveness of the heat exchanger for volumetric flowrates between 0.04 and 0.40 l/s. Explain the behavior of observed in this plot.

8.3-7 Ammonia is an ideal refrigerant in most respects, although it is an irritant at very low exposure levels and toxic above 1000 ppm. It is also weakly flammable. For these reasons, ammonia has not been used in supermarket refrigeration systems. It is possible, however, to confine the ammonia to the equipment room if a heat exchanger is employed to cool a secondary refrigerant. A 50% propylene glycol solution is heat exchanged with the ammonia, as shown in Figure P8.3-7. The propylene glycol solution is distributed to food cases in the supermarket. The glycol returns from the food cases at T_5 = -9°C and leaves the heat exchanger at T_6 = -15°C. The anticipated load on the food cases is 45 kW. The propylene glycol solution is non-toxic and non-flammable, eliminating the concerns with ammonia used in the supermarket. However, the use of the heat exchanger results in a reduction in the efficiency of the refrigerating equipment and extra pumping power. The ammonia flow rate is controlled with a thermostatic expansion valve to provide saturated vapor leaving the heat exchanger at state (1). The condenser provides saturated liquid at 40°C at state (3). The isentropic efficiency of the compressor is 0.72 and it operates adiabatically. Neglect pressure losses in the refrigeration piping. A shell and tube heat exchanger with 2 shell passes is employed, with the ammonia condensing in the shell and the glycol solution flowing in the tubes.

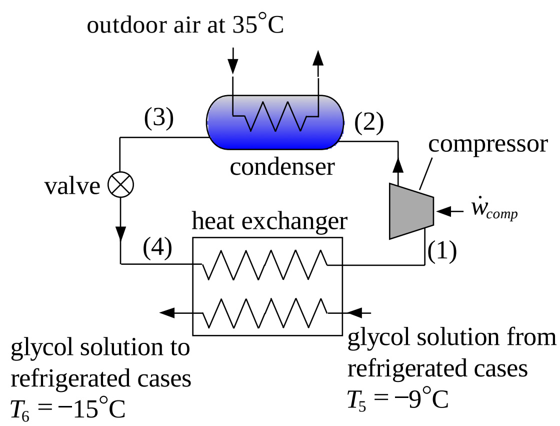


Figure P8.3-7 Supermarket refrigeration system

a.) Prepare a plot of the power required to operate the compressor and the temperature at state (1) as a function of the conductance of the heat exchanger for values ranging from 1.5 kW/K < *UA* < 30 kW/K.

b.) The shell and tube glycol-ammonia heat exchanger considered for this system has 12 parallel circuits of tubing with an inner diameter of 0.0125 m. The major resistance is due to the heat transfer coefficient on the glycol side of the heat exchanger. Calculate the required length of tubing for each circuit and the minimum pump power required to move the glycol solution through the heat exchanger as a function of the *UA*. What *UA* value would you recommend?

8.3-8 Air at atmospheric pressure and $T_{a,in}$ = 55°C flows with a velocity of u_a = 15 m/s through a finned circular tube heat exchanger having a geometry corresponding to compact heat exchanger core fc_tubes_s80-38T. The fins are made of copper. The length of the heat exchanger matrix in the flow direction is L = 0.4 m. R134a is flowing through the tubes at a saturation temperature of T_r = -5°C. The R134a enters the heat exchanger at a quality of x_{in} = 0.45 and exits at a quality of x_{out} = 0.65. The frontal area of the heat exchanger is A_f = 0.2 m^2 and the thickness of the tube wall is th_w = 0.9 mm. The tubes are plumbed in N = 10 parallel circuits.

a.) Determine the heat transfer coefficient on the air side.
b.) Determine the frictional pressure drop on the air side.
c.) Estimate the fin efficiency.
d.) Determine the average heat transfer coefficient on the R134a side.
e.) Determine the effectiveness of the heat exchanger and the rate of heat transfer.

Problem 8.3-9 (8-9 in text)

A concentric tube heat exchanger is built and operated as shown in Figure P8.3-9. The hot stream is a heat transfer fluid with specific heat capacity c_H = 2.5 kJ/kg-K. The hot stream enters at the center of the annulus at $T_{H,in}$ = 110°C with mass flow rate $\dot{m}_H = 0.64$ kg/s and then splits and an equal amount flows in both directions. The cold stream has specific heat capacity c_C = 4.0 kJ/kg-K. The cold enters the center pipe at $T_{C,in}$ = 10°C with mass flow rate $\dot{m}_C = 0.2$ kg/s. The outlet temperature of the hot-fluid that flows towards the left is $T_{H,out.x=0}$ = 45°C. The two sections of the heat exchanger have the same conductance.

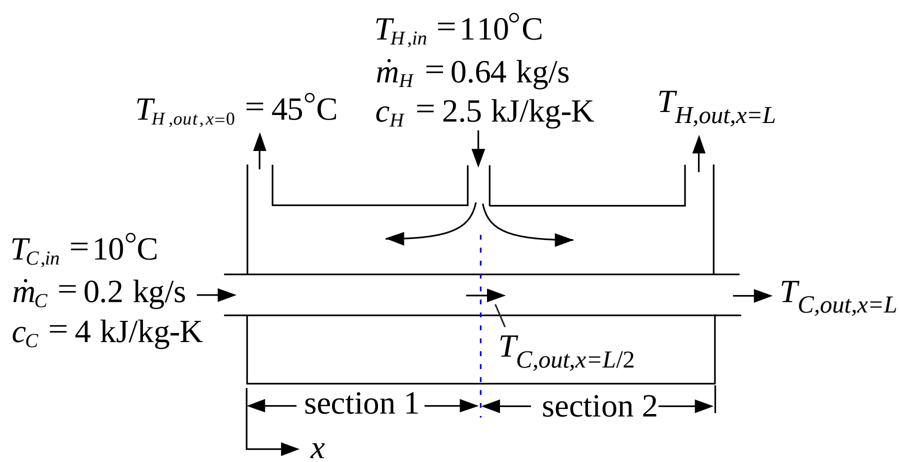


Figure P8.3-9: Concentric tube heat exchanger.

a.) Determine the temperature of the cold stream at the midpoint of the center tube and the temperature of the cold stream leaving the heat exchanger.
b.) Calculate the overall effectiveness of this heat exchanger considering both sections.
c.) How will the overall effectiveness be affected if the inlet temperature is increased to 400°C. (Assume that the properties of the heat transfer fluid are independent of temperature.)
d.) Is the overall effectiveness of this heat exchanger higher, lower, or the same as a counter-flow heat exchanger having the same inlet conditions? Justify your answer.

8.3-10 A factory requires 5000 gallons of hot (95°C) water each day. The water is not consumed; rather, it is used to provide heating in one part of a process and therefore goes down the drain at nominally 75°C. The process currently used to supply hot water to the factory is shown in Figure P8.3-10(a). The factory runs a single, 10-hour shift each day and therefore provides the hot water using a 5000 gallon supply tank. The supply tank is empty at the end of the shift and is subsequently filled with city water at 10°C. The cold city water is heated electrically during the 14 hours between the end of the shift and the beginning of the subsequent shift using a 150 kW heater. Therefore, the tank is full of 95°C water at the start of the shift and is emptied over the course of the next shift.

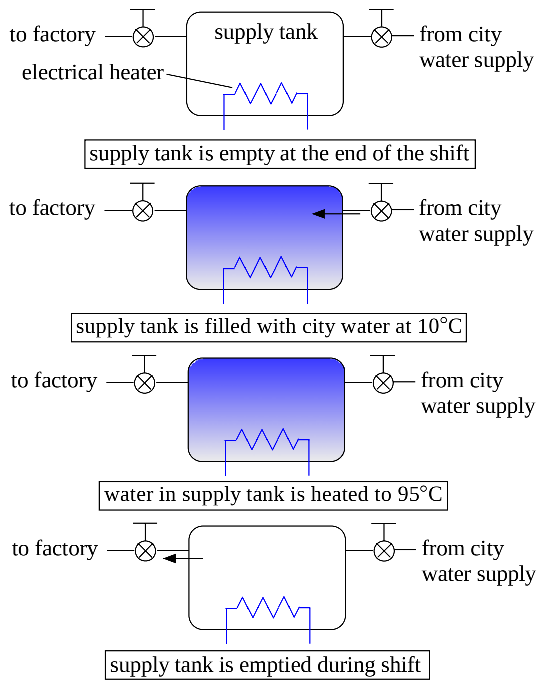


Figure P8.3-10(a): Current hot water system for factory.

You have been asked to replace the current hot water system with an alternative system that utilizes an energy recovery system in order to reduce the cost of the energy required to heat the water. In the proposed system, a second 5000 gallon tank (the drain tank) will be used to store the 75°C water that would otherwise go down the drain, as shown in Figure P8.3-10(b). At the conclusion of the shift, the drain tank will contain 5000 gallons of 75°C water. This water has been contaminated and therefore cannot be re-used directly. Therefore, a two phase water preparation process is required. During the first phase, the water in the drain tank is used to pre-heat the city water using a counterflow heat exchanger. The city water is heated in the counterflow heat exchanger and enters the supply tank at some intermediate temperature that is between 10°C and 75°C, depending on the total conductance (*UA*), of the counterflow heat exchanger. During the second phase of the water preparation process, the electrical heater is activated and used to heat the water to its final temperature, 95°C, so that the next shift can start.

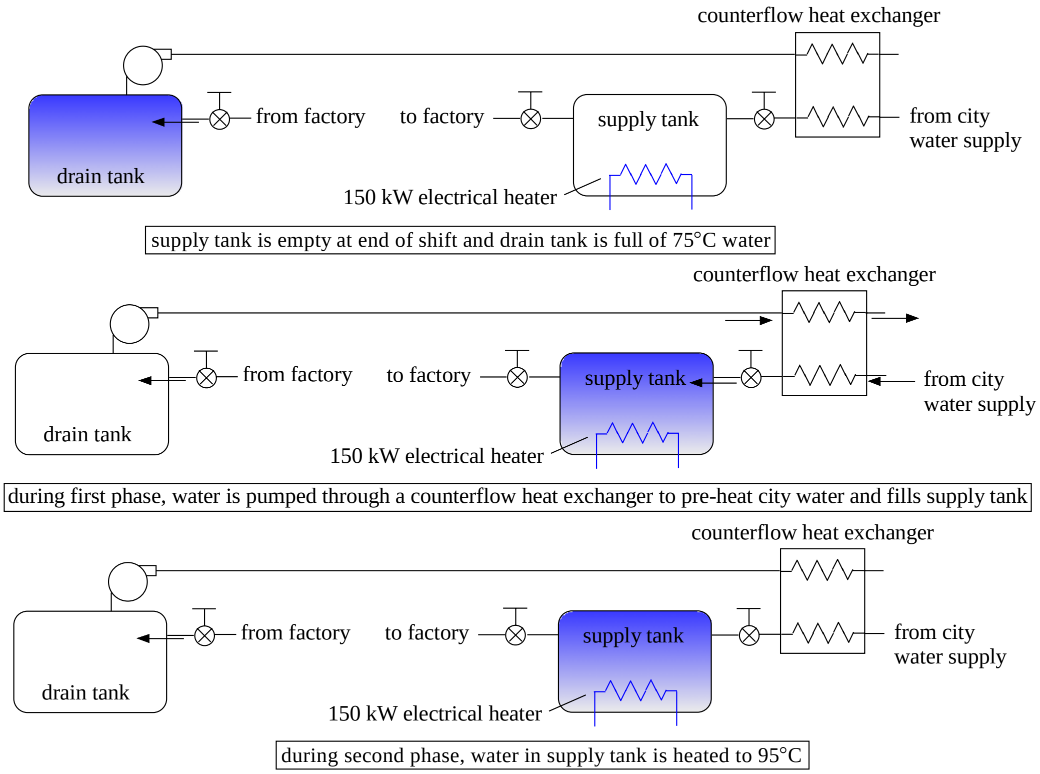


Figure P8.3-10(b): Proposed hot water system for factory.

Assume that the total conductance of the counterflow heat exchanger is UA = 6000 W/K and the duration of the first phase of water preparation is t_{phaseI} = 4 hr. Neglect any energy loss from either the supply or drain tanks.

a.) What is the volumetric flow rate of city water required to completely fill the supply tank and empty the drain tank (gal/min) during the first phase of water preparation?

b.) What is the temperature of the water in the supply tank at the end of the first phase of water preparation (°C)?

c.) If the 150 kW heater is activated at the conclusion of the first phase of water preparation and remains on until the next shift starts (recall that the time between shifts is 14 hr), then what will the final temperature of the water in the supply tank be (°C)?

d.) Prepare a plot showing the final temperature of the water in the supply tank (°C) as a function of duration of the first phase of water preparation (t_{phaseI}). What is the optimal duration of the first phase of water preparation (i.e., what value of t_{phaseI} provides water at exactly 95°C for the next shift)?

e.) If the cost of electricity is 0.10$/kW-hr, then how much money will the proposed system save per day as compared with the original system ($/day)? Make sure you set the duration of the first phase of water preparation to the optimal value you obtained from (d).

f.) The cost to install a new tank and pump and to modify the plumbing is estimated to be $25,000. The cost of the counterflow heat exchanger is proportional to its total conductance; the constant of proportionality is 10$/(W/K). Therefore, the heat exchanger with a total conductance of 6,000 W/K will cost $60,000. Determine the simple payback (years) of the proposed system. That is, neglecting the time value of money, how many years of savings are required in order to exactly equal the initial outlay of capital cost for the system?

g.) Prepare a plot showing the payback (years) as a function of the total conductance of the heat exchanger (W/K). Be sure that you are adjusting both the duration of the first phase of the water preparation process and the cost of the system as you change *UA*. What is the optimal value of *UA*?

8.3-11 Find the length of a concentric tube, counter-flow HX that is needed in order to cool oil entering at a flow rate of 0.1 kg/s from 100ºC to 60ºC using water flowing at 0.1 kg/s that enters at 30ºC. Determine the outlet temperature of the water. The total heat transfer coefficient per unit area, U, is 60 W/m^2-K, based on the inner tube area. The inner tube has a diameter of 2.5 cm. The specific capacity of the oil is c_o = 2.13 kJ/kg-K and the specific heat capacity of the water is c_w = 4.18 kJ/kg-K.

8.3-12 (8-10 in text) The power delivered to the wheels of a vehicle ($\dot{w}$) as a function of vehicle speed (V) is given by: $\dot{w} = -0.3937\left[\text{hp}\right] + 0.6300\left[\frac{\text{hp}}{\text{mph}}\right]V + 0.01417\left[\frac{\text{hp}}{\text{mph}^2}\right]V^2$ where power is in horsepower and velocity is in mph. The amount of heat rejected from the engine block ($\dot{q}_b$) is approximately equal to the amount of power delivered to the wheel (the rest of the energy from the fuel leaves with the exhaust gas). The heat is removed from the engine by pumping water through the engine block with a mass flow rate of $\dot{m}$ = 0.80 kg/s. The thermal communication between the engine block and the cooling water is very good, therefore you may assume that the water will exit the engine block at the engine block temperature (T_b). For the purpose of this problem you may model the water as having constant properties that are consistent with liquid water at 70°C. The heat is rejected from the water to the surrounding air using a radiator, as shown in Figure P8.3-12. When the car is moving, air is forced through the radiator due to the dynamic pressure associated with the relative motion of the car with respect to the air. That is, the air is forced through the radiator by a pressure difference that is equal to equal to $\rho_a V^2/2$ where ρ_a is the density of air. Assume that the temperature of the ambient air is T_∞ = 35°C and model the air in the radiator assuming that it has constant properties consistent with this temperature.

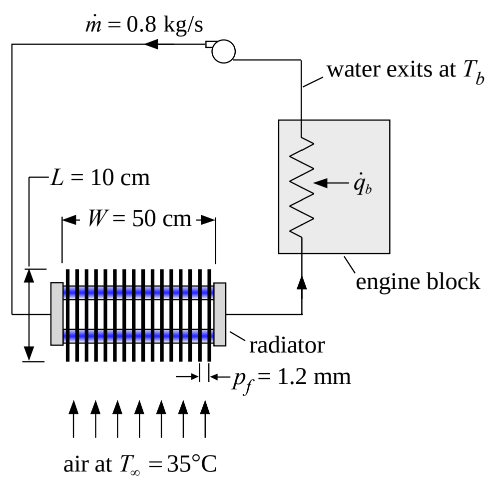


Figure P8.3-12: Engine block and radiator.

The radiator has a plate-fin geometry. There are a series of tubes installed in closely spaced metal plates that serve as fins. The fin pitch is p_f = 1.2 mm and therefore there are W/p_f plates available for heat transfer. The heat exchanger core has overall width W = 50 cm, height H = 30 cm (into the page), and length (in the flow direction) of L = 10 cm. For the purpose of modeling the air-side of the core you may assume that the air flow is consistent with an internal flow through rectangular ducts with dimension H x p_f. Assume that the fins are 100% efficient and neglect the thermal resistance between the fluid and the internal surface of the tubes. Also neglect convection from the external surfaces of the tubes as well as the reduction in the area of the plates associated with the presence of the tubes.

a.) Using the information above, develop an EES model that will allow you to predict the engine block temperature as a function of vehicle velocity. Prepare a plot showing T_b

vs V and explain the shape of the plot (if necessary, produce additional plots to help with your explanation). If the maximum allowable temperature for the engine block is 100°C (in order to prevent vaporization of the water) then what range of vehicle speeds are allowed? You should see both a minimum and maximum limit.

Its not easy to overcome the maximum speed limit identified (a); however, to overcome the minimum speed limit (so that you can pull up to a stop sign without your car overheating) you decide to add a fan. The fan can provide at most 500 cfm ($\dot{V}_o$ - the open circuit flow) and can produce at most 2.0 inch H_2O (Δp_{dh} - the dead-head pressure). The transition from open circuit to dead-head is linear. The fan curve is given by:

$$\Delta p_{fan} = \Delta p_{dh}\left(1 - \frac{\dot{V}}{\dot{V}_o}\right)$$

b.) Modify your code to simulate the situation where the air is provided by the fan rather than the vehicle motion. Overlay a plot showing T_b vs V for this configuration on the one from (a); have you successfully overcome the lower speed limitation?

8.3-13 A counter-flow heat exchanger is operating with a hot fluid inlet temperature of $T_{h,in}$ = 400 K and a cold fluid inlet temperature of $T_{c,in}$ = 300 K. The hot fluid flow has a capacity rate $\dot{C}_h$ = 100 W/K and the cold flow has a capacity rate $\dot{C}_c$ = 50 W/K. The total conductance of the heat exchanger is UA = 200 W/K.

a.) What is the heat transfer rate in the heat exchanger, $\dot{q}$?

b.) What is the exit temperature of the cold fluid, $T_{c,out}$?

c.) Sketch the temperature distribution within the counter-flow heat exchanger.

d.) The heat exchanger is re-plumbed so that the cold and hot fluid flow in the same direction (i.e., in a parallel-flow configuration), what is the exit temperature of the cold fluid? All other aspects of the problem remain the same.

e.) Sketch the temperature distribution within the parallel-flow heat exchanger.

8.3-14 (8-11 in text) A parallel-flow heat exchanger has a total conductance UA = 10 W/K. The hot fluid enters at $T_{h,in}$ = 400 K and has a capacity rate $\dot{C}_h$ = 10 W/K. The cold fluid enters at $T_{c,in}$ = 300 K and has a capacity rate $\dot{C}_c$ = 5 W/K.

a.) Determine the number of transfer units (NTU), effectiveness (ε), heat transfer rate ($\dot{q}$), and exit temperatures ($T_{h,out}$ and $T_{c,out}$) for the heat exchanger.

b.) Sketch the temperature distribution within the heat exchanger.

c.) Sketch the temperature distribution within the heat exchanger if the conductance of the heat exchanger is very large; that is, what is the temperature distribution in the limit that $UA \rightarrow \infty$.

d.) Sketch how the hot exit temperature will change as the total conductance (UA) is varied, with all other quantities held constant at the values listed in the problem statement. Be sure to indicate how your plot behaves as UA approaches zero and as UA approaches infinity.

8.3-15 (8-12 in text) A heat exchanger has a core geometry that corresponds to finned circular tube core 'fc_tubes_s80_38T' in the compact heat exchanger library. The frontal area of the core has dimensions W = 7.75 inch and H = 7.75 inch. The length of the core is L = 1.5 inch. The core is integrated with a fan that has a head-flow curve given by: $\Delta p = a - b\dot{V}_a$ where Δp is the pressure rise across the fan, $\dot{V}_a$ is the volumetric flow rate of air, a = 0.3927 inH2O and b = 0.0021 inH2O/cfm are the coefficients of the fan curve. The manufacturer has tested the heat exchanger with atmospheric air at $T_{a,in}$ = 20ºC and water at $T_{w,in}$ = 75ºC, p_w = 65 psia flowing through the tubes. The test data are shown in Table P8.3-15. The tubes are plumbed in series (i.e., all of the water flows through each tube) and the tube thickness is th_t = 0.035 inch.

Table P8.3-15: Manufacturer's data for heat exchanger.

Water flow rate	**Water outlet temperature**
0.13 gpm	44.3ºC
0.25 gpm	51.1ºC
0.5 gpm	60.1ºC
1 gpm	66.9ºC
2 gpm	70.8ºC
4 gpm	72.9ºC

a.) Develop a model using the effectiveness-NTU technique that can predict the outlet temperature of the water for a water flow rate of the water $\dot{V}_w$.

b.) Plot the outlet temperature of the water as a function of the water flow rate and overlay the manufacturer's data onto your plot.

8.3-16 An oil cooler for a small engine is a counterflow, tube-in-tube type heat exchanger that transfers heat from hot oil leaving the engine to cold water. The water enters the annular space with temperature $T_{c,in}$ = 10°C and mass flow rate $\dot{m}_w$ = 0.02 kg/s. The oil enters the the center tube with temperature $T_{o,in}$ = 90°C and mass flow rate $\dot{m}_o$ = 0.004 kg/s. The outer diameter of the outer tube is $D_{ot,o}$ = 0.75 inch and the outer diameter of the inner tube is $D_{ot,i}$ = 0.375 inch. The length and wall thickness of both tubes are L = 48 inch and th = 0.035 inch, respectively. The tubes are 304 stainless steel.

a.) Determine the outlet temperature of the oil as well as the heat transferred in the oil cooler using Eq. (8-36) in the text.

b.) Determine the performance of the heat exchanger using the ε-NTU solution and show that the result is consistent with (a).

c.) Plot the rate of heat transfer as a function of the oil inlet temperature. If the engine rejects 200 W to the oil then what will the oil outlet temperature be?

8.3-17 In Problem 8.3-15 (8-12 in the text) a heat exchanger with a core geometry that corresponds to finned circular tube core 'fc_tubes_s80_38T' in the compact heat exchanger library was modeled from first principles and the result compared to the manufacturer's data. An alternative method of making an engineering model of the heat exchanger is to use one data point to determine the conductance of the heat exchanger at a nominal operating condition and then model the heat exchanger by assuming that the conductance is nominally constant (or varies in some systematic way).

The frontal area of the core has dimensions W = 7.75 inch and H = 7.75 inch. The length of the core is L = 1.5 inch. The core is integrated with a fan that has a head-flow curve given by: $\Delta p = a - b\dot{V}_a$ where Δp is the pressure rise across the fan, $\dot{V}_a$ is the volumetric flow rate of air, a = 0.3927 inH2O and b = 0.0021 inH2O/cfm are the coefficients of the fan curve. The manufacturer has tested the heat exchanger with atmospheric air at $T_{a,in}$ = 20ºC and water at $T_{w,in}$ = 75ºC, p_w = 65 psia flowing through the tubes. The test data are shown in Table P8.3-15.

a.) Determine the conductance associated with the data point at $\dot{V}_w$ = 1 gpm.

b.) Assume that the conductance does not change with water flow rate in order to develop a simple model of the heat exchanger. Plot the outlet temperature of the water predicted by the model as a function of water flow rate and compare with the data.

8.3-18 A steam coil is used to provide heating to a zone of a building. The coil has geometry consistent with finned tube fc_tubes_sCF-775-58T in the compact heat exchanger library. The coil is installed in a square air duct with width W = 18 inch and height H = 18 inch. The coil can be installed in modules that are placed in series. The length of each module in the flow direction is L_{mod} = 6 inch. The volumetric flow rate of air entering the coil is $\dot{V}_{in}$ = 900 cfm. The air enters the coil at $T_{a,in}$ = -5ºC. The condensing steam in the coil is two-phase throughout and at pressure p_s = 10 psi. The temperature of the zone is T_{zone} = 20ºC. Air is heated in the steam coil and enters the zone at a temperature greater than T_{zone}. The air leaves the zone at T_{zone} and therefore provides heating to the zone. The blower that moves the air through the coil has an efficiency of η_b = 0.55. You may neglect fouling and assume that the efficiency of the fins on the air-side is 100%. Further, you may neglect the convection resistance associated with the condensing steam and conduction through the tubes.

a.) If only one module is installed, determine the power required by the blower, the temperature of the air leaving the steam coil, and the heat provided to the zone.

b.) Plot the heat provided to the zone and the blower power as a function of the number of modules. Explain the shape of your plot. How many modules would you suggest using for the steam coil? Why?

8.3-19 A simple heat exchanger is made by soldering two tubes together, as shown in Figure P8.3-19(a).

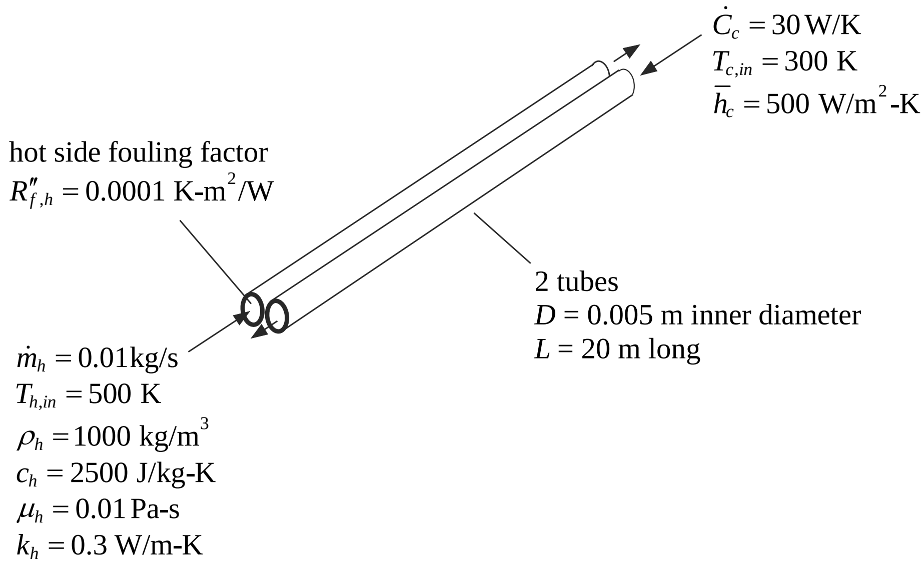


Figure P8.3-19(a): Heat exchanger made by soldering two tubes together.

The two tubes each have inner diameter D = 0.005 m and length L = 20 m. The resistance to conduction through the tube walls and the solder joint can be neglected. The hot fluid has mass flow rate $\dot{m}_h$ = 0.01 kg/s and enters with $T_{h,in}$ = 500 K. The properties of the hot flow are density ρ_h = 1000 kg/m^3, specific heat capacity c_h = 2500 J/kg-K, viscosity μ_h = 0.01 Pa-s, and conductivity k_h = 0.3 W/m-K. There is a fouling factor on the hot fluid side of $R''_{f,h}$ = 0.0001 K-m^2/W. There is no fouling on the cold side. The cold fluid enters with $T_{c,in}$ = 300 K and has capacitance rate $\dot{C}_c$ = 30 W/K. The average heat transfer coefficient on the cold fluid side is $\overline{h}_c$ = 500 W/m^2-K. Assume that the internal surface of both tubes is smooth.

Figure 8.3-19(b) illustrates the fully developed friction factor for a smooth, round tube as a function of Reynolds number.

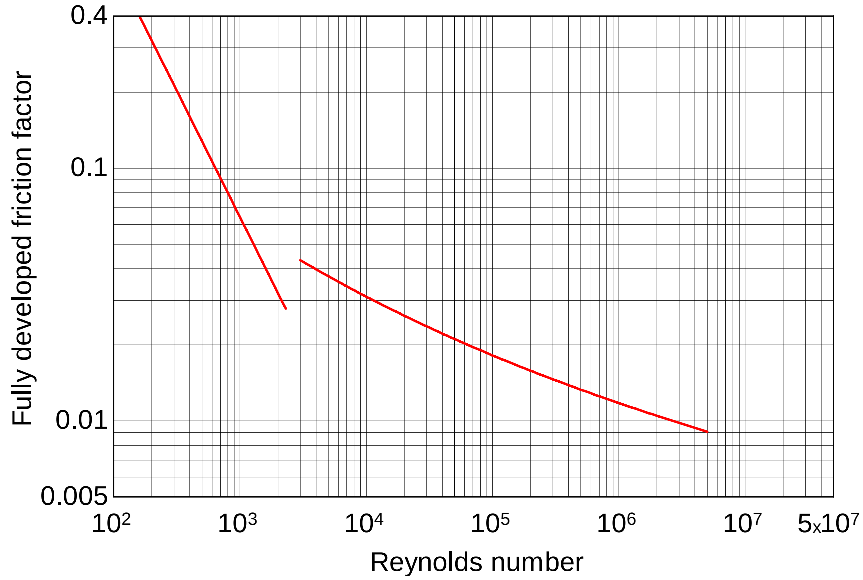


Figure 8.3-19(b): Fully developed friction factor for a smooth, round tube as a function of Reynolds number.

Figure 8.3-19(c) illustrates the fully developed Nusselt number with a constant wall temperature boundary condition for a smooth round tube as a function of Reynolds number and various values of the Prandtl number.

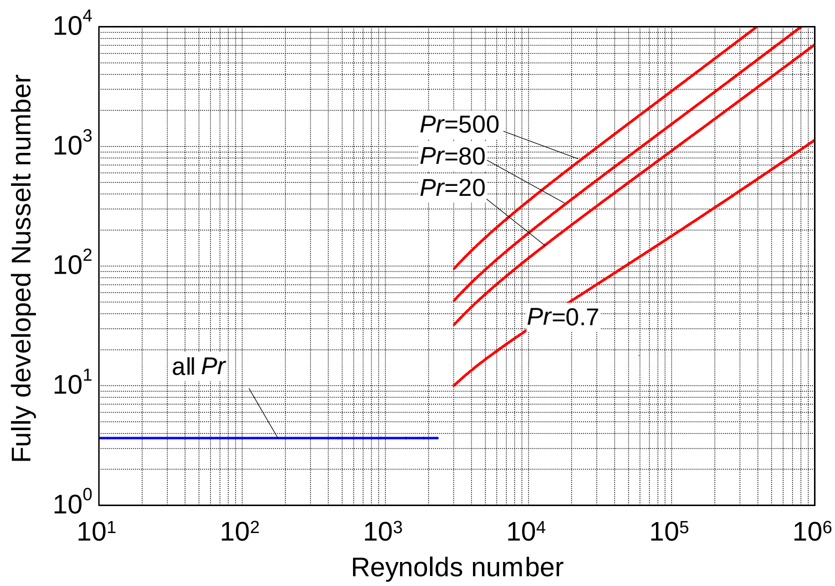


Figure 8.3-19(c): Fully developed Nusselt number at constant wall temperature for a smooth, round tube as a function of Reynolds number and various values of the Prandtl number.

You may assume that the tubes are long enough that the fully developed friction factor and Nusselt number represent the average friction factor and Nusselt number.

a.) Using Figure 8.3-19(b), estimate the pressure drop associated with the flow of the hot fluid through the heat exchanger.

b.) What is the minimum possible pumping power required to move the hot fluid through the heat exchanger?

c.) Using Figure 8.3-19(c), estimate the average heat transfer coefficient on the hot side of the heat exchanger.

d.) Explain briefly why the Nusselt number at low Reynolds number in Figure 8.3-19(c) is independent of both Reynolds number and Prandtl number.

e.) Determine the hot outlet temperature. Assume that your answer from part (c) was $\bar{h}_h = 150 \ \frac{\text{W}}{\text{m}^2\text{-K}}$ (this may or may not be the correct answer). Use Figure 8.3-19(d) to determine the effectiveness of a counter-flow heat exchanger.

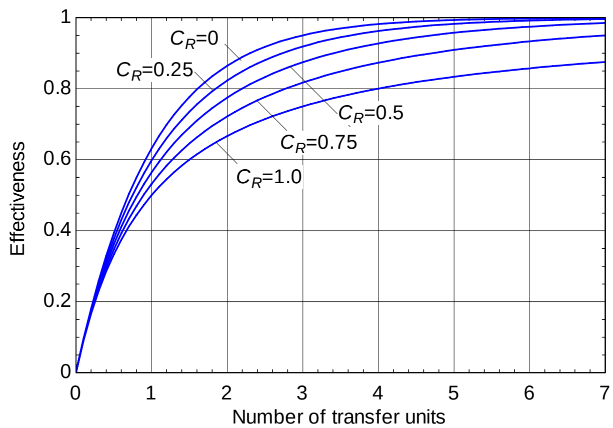


Figure 8.3-19(d): Effectiveness of a counterflow heat exchanger as a function of *NTU* for various values of C_R.

f.) Your boss has suggested that the fouling on the hot side controls the performance of the heat exchanger and therefore extensive testing should be carried out in order to understand this phenomenon. Do you agree? Why or why not?

g.) If the heat exchanger were re-plumbed so that it was in a parallel flow rather than a counterflow configuration do you expect the hot outlet temperature to change much? Will it go up or down? Justify your answers.

h.) If the tube on the hot side were replaced by one that was not perfectly smooth then what would the effect be on the hot outlet temperature and the pressure drop? Justify your answers.

Section 8.4: Pinch Point Considerations

8.4-1 A refrigeration cycles employs carbon dioxide as the refrigerant that enters the high temperature cooler at 80 bar and 70 C at steady conditions. The carbon dioxide flowrate is 0.063 kg/s and it must be cooled to 30°C before being expanded. A suggestion has been made to use this 'waste heat' to provide domestic hot water. The water enters at 5°C from the mains supply at atmospheric pressure.

a.) What is the maximum temperature that the water can be heated to and what is the corresponding flow rate of the water?

b.) Repeat part (a) if a pinch point temperature difference of 5 K is required to overcome heat transfer resistances.

8.4-2 A Kalina cycle is a power cycle that uses a mixture of ammonia and water as the working fluid, rather than a pure fluid such as steam. A mixture of x_{am} = 50% ammonia by mass is introduced to the evaporator heat exchanger at $T_{am\text{-}w,in}$ = 310 K and $p_{am\text{-}w}$ =10 bar. The mixture exits the evaporator at $T_{am\text{-}w,out}$ = 450 K and $p_{am\text{-}w}$ = 10 bar. The thermal energy needed to evaporate the mixture is provided by an air stream that enters the evaporator heat exchanger at $T_{a,in}$ = 460 K and p_a = 1 bar. Determine the minimum mass flow rate of the air needed for this purpose (corresponding to infinite heat exchanger conductance) per unit mass flow rate of the ammonia-water mixture. Also determine the outlet air temperature. Ammonia water properties are provided in EES with the NH3H2O external procedure. Directions for using these properties can be found in the Function Info dialog (Options menu) by selecting External routines and NH3H2O options and then clicking the Function Info button.

Section 8.5: Heat Exchangers with Phase Change

8.5-1 The power block for a solar central receiver power tower system is currently being designed. The fluid circulated through the receiver of solar power tower is a molten salt having an average specific heat of c_s = 1.52 kJ/kg-K. At design conditions, the salt will enter the counterflow superheater at $\dot{m}_s$ = 225 kg/s and $T_{s,in}$ = 480°C. The design calls for the salt to heat water at $\dot{m}_w$ = 25 kg/s from $T_{w,in}$ = 225°C to $T_{w,in}$ = 370°C at a pressure of p_w = 100 bar. You have been asked to determine the total heat exchanger conductance needed for this purpose and the exiting temperature of the salt.

8.5-2 A single pass shell and tube heat exchanger is used to condense steam exiting from the power turbine at p_s = 0.1 bar and $x_{s,in}$ = 95% quality at a mass flow rate of $\dot{m}_s$ = 3.8 kg/s. The steam flows in the shell and treated cooling water from a cooling tower at $T_{cw,in}$ = 24°C, p_{cw} = 1 bar and $\dot{m}_{cw}$ = 325 kg/s enters the tubes. There are N_{tube} = 384 tubes in the shell, each having an outer diameter of D_o = 38.1 mm, an inner diameter of D_i = 35.1 mm and a length of L = 8 m with a roughness of e = 1 μm. The tubes are made of steel with a thermal conductivity of k_t = 50 W/m-K. Fouling on the inside of the tubes is expected to occur for the treated water. Based on manufacturer's data, the heat transfer coefficient between the condensing steam and the outer tube wall is $\overline{h}_{s,sat}$ = 4625 W/m^2-K. Some of the tubes are submerged in the condensed steam and the heat transfer coefficient between the condensate and the tube wall for these tubes is estimated to be $\overline{h}_{s,sc}$ = 150 W/m^2-K. Heat losses from the shell to the surroundings are negligible. Determine the outlet temperature of the condensate and the well-mixed temperature of exiting cooling water.

Section 8.6: Numerical Modeling of Parallel-Flow and Counter-Flow Heat Exchangers

8.6-1 This problem considers the heat rejection heat exchanger in a refrigeration cycle that uses carbon dioxide as the refrigerant. Carbon dioxide is being considered as a replacement refrigerant for automobile cooling and heat pump systems because it is non-flammable and it is a natural refrigerant that does not contribute directly to ozone depletion or global warming concerns. The critical temperature of carbon dioxide is about 304 K (31°C), which is often exceeded in an air conditioning application. The critical pressure of carbon dioxide is about 7.4 MPa (74 atm). Therefore, the pressure of the carbon dioxide within the heat rejection heat exchanger may range from 80 to 120 atm; this very high pressure is one of the issues associated with using carbon dioxide as a refrigerant.

The heat rejection heat exchanger in a carbon dioxide refrigeration cycle is called a cooler rather than a condenser because the carbon dioxide is above its critical pressure and therefore it does not condense, even when cooled to temperatures below the critical temperature. The specific heat of carbon dioxide near its critical point is strongly temperature dependent and exhibits a distinct 'bump' that becomes larger and sharper as the pressure approaches the critical pressure. The carbon dioxide does not undergo a phase change and therefore the cooler cannot be modeled using the technique discussed in Section 8.5. However, the specific heat cannot be assumed to be constant in the cooler at a pressure of 80 or even 100 bar and therefore the ε-NTU solution for a cross-flow heat exchanger cannot be used.

In this problem, you will develop a numerical solution where the cooler is discretized into sub-heat exchangers that each operate over a small temperature span. If enough sub-heat exchangers are used, then the temperature span is sufficiently small that the specific heat capacity within each sub-heat exchanger can be assumed to be constant and therefore the performance of the sub-heat exchangers can be predicted using the ε-NTU solution. This approach is similar to the technique described in Section 8.6.3.

The finned, circular tube heat exchanger that is examined in EXAMPLE 8.1-1 and 8.1-2 is used as the cooler. Clean dry air is forced to flow through the heat exchanger perpendicular to the tubes (i.e., in cross-flow) with a volumetric flow rate $\dot{V}_C$ = 0.06 m^3/s. The inlet temperature of the air is $T_{C,in}$ = 20°C and the air is at atmospheric pressure. Carbon dioxide enters the tubes at $T_{H,in}$ = 60°C and $P_{H,in}$ = 80 bar with a mass flow rate $\dot{m}_H$ = 0.01 kg/s.

a.) Determine the rate of heat transfer and the outlet state of the carbon dioxide.

8.6-2 (8-13 in text) A Joule-Thomson refrigeration cycle is illustrated in Figure P8.6-2.

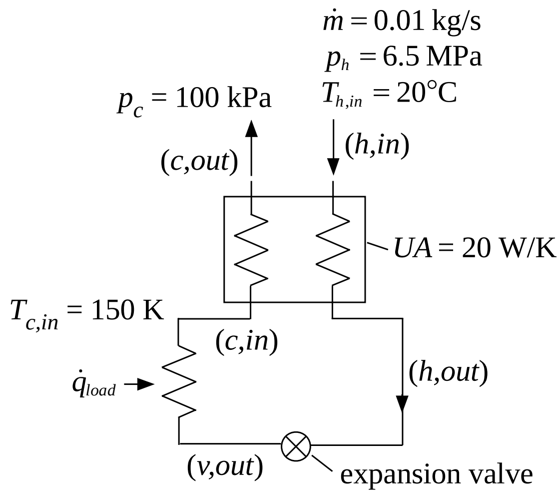


Figure P8.6-2: Joule-Thomson refrigeration cycle.

The system uses pure argon as the working fluid. High pressure argon enters a counterflow heat exchanger with mass flow rate $\dot{m}$ = 0.01 kg/s at $T_{h,in}$ = 20ºC and p_h = 6.5 MPa. The argon flows through the heat exchanger where it is pre-cooled by the low pressure argon returning from the cold end of the cycle. The high pressure argon leaving the heat exchanger enters an expansion valve where it is expanded to p_c = 100 kPa. The argon passes through a load heat exchanger where it accepts a refrigeration load, $\dot{q}_{load}$, and it heated to $T_{c,in}$ = 150 K. The conductance of the heat exchanger is *UA*= 20 W/K. Neglect pressure loss in the heat exchanger on both the hot and cold sides of the heat exchanger.

a.) Use the effectiveness-*NTU* method to estimate the effectiveness of the heat exchanger and the rate of heat transferred from the hot to the cold stream in the heat exchanger. Calculate the specific heat capacity of the high- and low-pressure argon using the average of the hot and cold inlet temperatures.

b.) Determine the refrigeration load provided by the cycle.

c.) Prepare a plot of refrigeration load as a function of cold inlet temperature for 85 K < $T_{c,in}$ < 290 K and various values of the conductance. A negative refrigeration load is not physically possible (without some external cooling); therefore, terminate your plots at $\dot{q}_{load}$ = 0 W.

d.) Instead of using the effectiveness-*NTU* method, divide the heat exchanger into sub-heat exchangers as discussed in Section 8.6.3. What is the heat transferred in the heat exchanger for the conditions listed in the problem statement?

e.) Determine the refrigeration load associated with your prediction from (d).

f.) Overlay on your plot from (c) the refrigeration load as a function of cold inlet temperature for the same values of the conductance.

8.6-3 (8-14 in text) A counter-flow heat exchanger has a total conductance of UA = 130 W/K. Air flows on the hot side. The air enters at $T_{h,in}$ = 500 K with pressure p_h = 1 atm and mass flow rate $\dot{m}_h$ = 0.08 kg/s. Carbon dioxide flows on the cold side. The CO_2 enters at $T_{c,in}$ = 300 K with pressure p_c = 80 atm and mass flow rate $\dot{m}_c$ = 0.02 kg/s.

a.) Plot the specific heat capacity of air at 1 atm and carbon dioxide at 80 atm and comment on whether the ε-*NTU* solution can be applied to this heat exchanger.

b.) Prepare a solution to this problem by numerically integrating the governing equations using the Euler technique, as discussed in Section 8.6.2.

c.) Using your solution from (b), plot the temperature of the carbon dioxide and air as a function of the dimensionless axial position (x/L).

d.) Plot the rate of heat transfer predicted by the model as a function of the number of integration steps.

e.) Prepare a solution to this problem by sub-dividing the heat exchanger into sub-heat exchangers, as discussed in Section 8.6.3.

f.) Overlay on your plot from (c) the temperature distribution predicted by your model from (e).

g.) Overlay on your plot from (d) the rate of heat transfer predicted by your model from (e) as a function of the number of sub-heat exchangers.

Section 8.7: Axial Conduction in Heat Exchangers

8.7-1 Nitrogen gas at $T_{n,in}$ = 300 K and atmospheric pressure is cooled by helium gas at $T_{h,in}$ = 240 K and atmospheric pressure in a concentric tube heat exchanger that utilizes N_{tube}= 30 tubes in parallel. The mass flow rate of the nitrogen is $\dot{m}_n$ = 3 g/s and the mass flow rate of helium is $\dot{m}_h$ = 1.5 g/s. The heat exchanger is made of copper with pipe wall thickness of δ = 1 mm and length L = 0.15 m.. The inner pipe has an inner diameter of D_{in} = 0.3 cm. The outer pipe (which is well-insulated on its outside surface) has an inner diameter of D_{out} = 1.0 cm. The roughness of the pipe surfaces are e = 1 μm.. The helium flows through the annulus and the nitrogen through the center tube in a counterflow configuration.

a) Estimate the outlet temperatures of the helium and nitrogen and the heat transfer rate assuming there is no axial conduction.

b) Calculate the axial conduction parameter and determine whether axial conduction is a concern for this heat exchanger. If so, use an approximate method to include axial conduction in the estimates of the heat transfer rate and outlet temperatures.

c) Use the numerical analysis provided in the EES AxialConductionHX library to determine the effectiveness including axial conduction. Comment on the accuracy of the approximate method you employed in part (b).

8.7-2 A parallel plate heat exchanger is made using th_m = 3 mm thick copper plates to form 100 channels. The vertical space between any two plates is H = 6 mm. The cold fluid is argon gas at atmospheric pressure that enters at $T_{C,in}$ = 90 K. The hot fluid is nitrogen at atmospheric pressure that enters at $T_{H,in}$ = 300 K. The flow rates of both fluids is $\dot{m}_H = \dot{m}_C$ = 25 g/s. Each plate in the heat exchanger extends L = 0.7 m in the flow direction and is W = 0.35 m wide. The heat exchanger is counterflow.

a) Estimate the convective heat transfer coefficients for the hot and cold fluids.
b) Determine the effectiveness of this heat exchanger assuming that there is no axial conduction.
c) Determine the value of the axial conduction parameter.
d) Correct your result in (b) using one of the approximate methods in Section 8.7.2.
e) Use the numerical analysis provided in the EES AxialConductionHX library to determine the effectiveness including axial conduction. Comment on the accuracy of the approximate method you employed in (d).

Section 8.10: Regenerators

8.10-1 (8-15 in text) A solar heating system is shown in Figure P8.10-1.

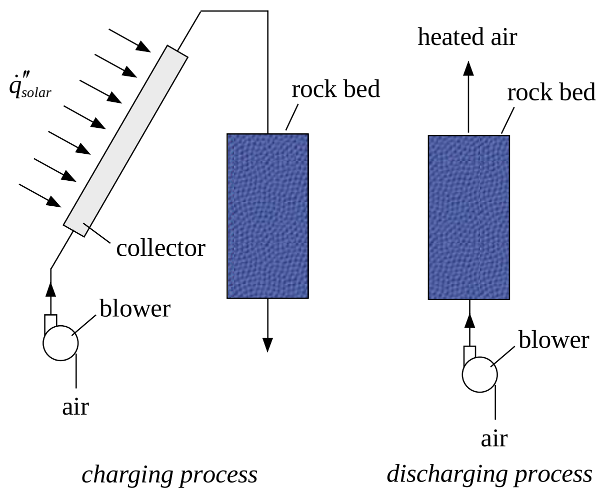


Figure P8.10-1: Solar heating system during charging and during discharging.

During the day, the solar heat is not required and therefore air is blown through a series of solar collectors where it is heated as shown in Figure P8.10-1. The thermal energy is stored in a large rock bed regenerator. The rock bed is L= 20 ft long in the flow direction and W = 10 ft x W = 10 ft in cross-sectional area. The bed is filled with D_p = 0.5 inch diameter rocks with density ρ_r = 100 lb_m/ft^3 and specific heat capacity c_r = 0.2 Btu/lb_m-°F. The charging process goes on for t_{charge} =12 hr. There are N_{col} = 40 solar collectors, each with area 8 ft x 4 ft. During the charging process, atmospheric air at T_{indoor} = 70°F enters the collectors where it is heated by the solar irradiation $\dot{q}''_{solar}$ = 750 W/m^2. The efficiency of the collector is given by: $\eta_{collector}$ = 0.75 - 0.0015 [K^{-1}] ($T_{r,in}$ - $T_{outdoor}$) where $T_{outdoor}$ = 10°F and $T_{r,in}$ is the temperature of the air leaving the collector and entering the regenerator. The collector efficiency is the ratio of the energy transferred to the air to the energy incident on the collector. During the night, the energy that was stored in the rock bed is used to provide heating, as shown in Figure P8.10-1. Air at T_{indoor} = 70°F enters the rock bed where it is heated. The hot air is provided to the building. The blower used during both the charging and discharging process has an efficiency of η_b = 0.6 and a pressure/flow curve that goes linearly from Δp_{dh} = 0.5 inch of water at zero flow to $\dot{V}_{open}$ = 1800 cfm at zero pressure rise. Neglect the pressure drop across the collectors and assume that the pressure drop that must be overcome by the blower is related to the flow through the rock bed. The porosity of the rock bed is ϕ = 0.35 and assume that the rock bed is well-insulated.

a.) What is the temperature of the air entering the rock bed during the charging process and the mass flow rate of air during the charging and discharging process?

b.) What is the amount of heat transfer from the rock bed to the air during the discharge process?

c.) There are 100 heating days per year in this location. What is the total amount of heating energy saved over a 10 year period?

d.) If the cost of natural gas is gc = 3.5$/therm then what is the total heating cost saved over a 10 year period? (Neglect the time value of money for this analysis.)

e.) The cost of the solar collectors is *cc* = 45$/ft^2 and the cost of the rock bed is *rc* = 40$/ton. The cost of the electrical energy required to run the blowers is *ec* = 0.12$/kW-hr. Determine the net savings associated with owning the equipment over a 10 year period.

f.) Plot the net savings as a function of the number of solar collectors. You should see that an optimal number of collectors exists. Provide an explanation for this observation.

g.) Plot the net savings as a function of the length of the rock bed (with N_{col} = 40). You should see that an optimal length of the rock bed. Explain this fact.

h.) Determine the optimal number of collectors and rock bed length.

8.10-2 (8-16 in text) A Stirling engine is shown in Figure P8.10-2.

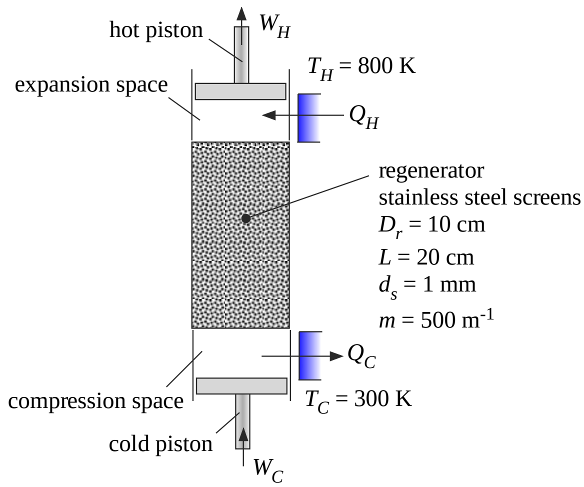


Figure P8.10-2: Stirling engine.

The mass of gas in the Stirling engine is M_{gas} = 0.01 kg. The gas is air and can be modeled as being an ideal gas with gas constant R_a = 287.1 J/kg-K and specific heat capacity ratio γ = 1.4. You may neglect the air entrained in the regenerator void volume and assume that all of the air is either in the compression or expansion space. The Stirling engine's performance will be estimated using a very simple model of the Stirling cycle. During the compression process, all of the gas is contained in the compression space and the cold piston is moved up until the pressure of the air goes from P_{low} = 1.0 MPa to P_{high} = 1.5 MPa. This process occurs isothermally at T_C = 300 K and will be modeled as being reversible. During the cold-to-hot blow process, the two pistons move together so that the gas moves from the compression space to the expansion space. To the extent that the regenerator is not 100% effective, the gas leaves the hot end of the regenerator at a temperature that is below T_H = 800 K and therefore a heat transfer occurs from the hot reservoir in order to heat this gas to T_H; this heat transfer is the manifestation of the regenerator loss. During the expansion process, all of the gas is contained in the expansion space and the hot piston is moved up until the pressure of the air goes from P_{high} to P_{low}. This process occurs isothermally at T_H and will also be modeled as being reversible. During the hot-to-cold blow process, the two pistons move together so that the gas moves from the expansion space to the compression space. The cycle occurs with a frequency of f = 10 Hz and each of the four processes take an equal amount of time.

a.) What is the efficiency of the cycle and the average power produced in the absence of any regenerator loss?

The regenerator is a cylinder filled with stainless steel screens. The regenerator diameter is D_r = 10 cm and the length is L = 20 cm. The screens have wire diameter d_s = 1 mm and mesh m = 500 m^{-1}.

b.) Estimate the regenerator loss per cycle and the efficiency including this regenerator loss.

c.) Plot the efficiency and average power as a function of the frequency of the Stirling engine. Explain the shape of your plot.