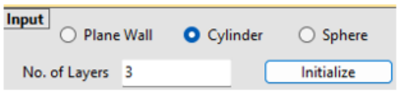
List of Heat Transfer Tools:

HT-01: Heat Conduction Composite Systems (Walls/Cylinders/Spheres)

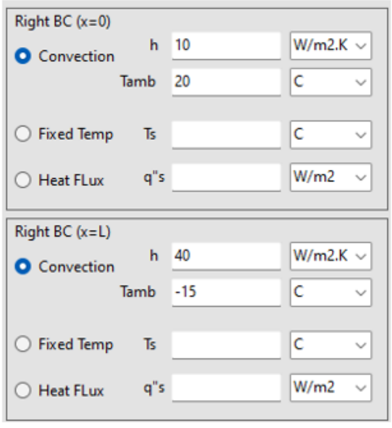
This tool provides an automated interactive panel to study one-dimensional heat transfer in layered composite structures (walls, cylinder and spheres).

Required Input:

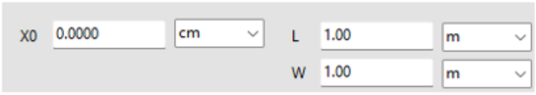
1. **Initialization**
   - Choose geometry type (plane wall, cylinder, or sphere).
   - Provide the number of layers to be modeled in the composite structure.
   - Click Initialize. This will result in modifications to input/output panels to accommodate the number of layers and geometry type



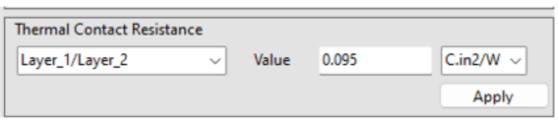
2. **Boundary Conditions**
   - Choose boundary condition type (convection, fixed temperature, or specified heat flux) for the "***Right***" (higher coordinate value) and "***Left***" (lower coordinate value).
   - For convection BC, enter value for heat transfer coefficient and ambient temperature.
   - For Fixed Temperature enter the value for the temperature.
   - For Specified Heat Flux, enter heat flux value.
   - Choose any unit for each from options provided in *Unit Combo Boxes*.



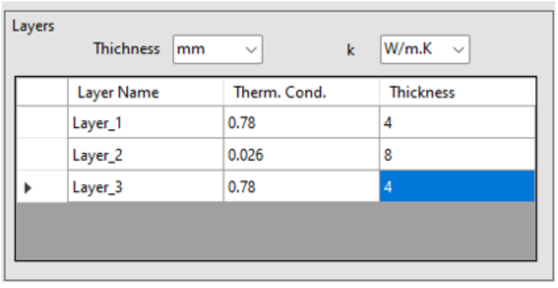
3. **Geometric Parameters**
   Wall: Minimum x value (Default is zero); Wall Size (Default 1m x 1m)
   Cylinder: Inner Radius; Cylinder Length
   Sphere: Inner Radius



4. **Thermal Contact Resistance**
   Interfacial contact resistance may be applied between any layers. Film resistance may also be applied at inner/outer surfaces.



5. **Layer Details**
   Provide thermal conductivity and thickness for each layer.
   - Choose unit for each can be set at the top of this panel.
   - Enter values in the spreadsheet.



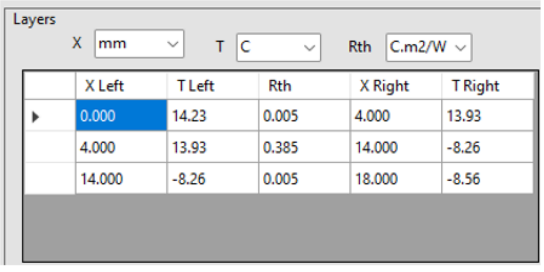
Results:

1. **Layers**

   In this section results for each layer is presented in a spreadsheet format. These include

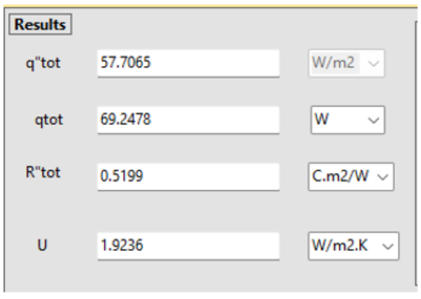
   - Temperature at boundaries and interfaces.
   - Thermal resistance in each layer.

   These results are presented using units above the spreadsheet.



2. **Overall Results**

   These results consist of heat flux, total rate of heat transfer, total thermal impedance, and overall heat transfer coefficient.



Example – Composite Wall:

A 0.8 m by 1.5 m glass window (thermal conductivity of 0.78 W/m.K) with a thickness of 8 mm. Assume the interior of the room is maintained at 20 °C with natural convection heat transfer coefficient of 10 W/m^2.K, and the external ambient at -10 °C with heat transfer coefficient of 40 W/m^2.K (which include thermal radiation effects). Determine the steady rate of heat transfer through this window and the temperature of its inner surface for the following two conditions:

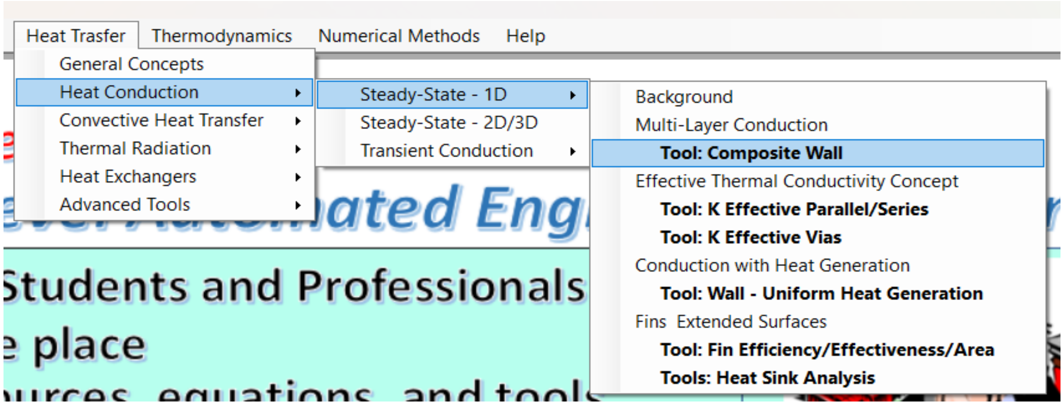
a) Single pane with a thickness of 8 mm.

b) Double-pane consisting of two 4-mm thick glass layers separated by 10-mm thick stagnant airspace.

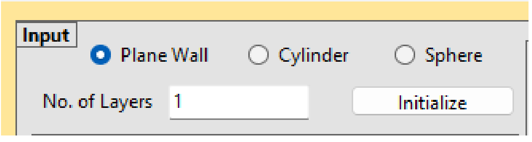
Solution:

a) Single pane

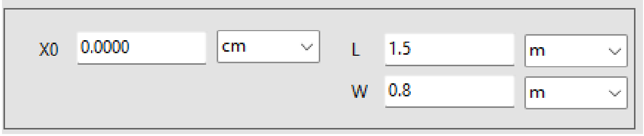
- Open "*Heat Conduction Composite Systems*" Panel:



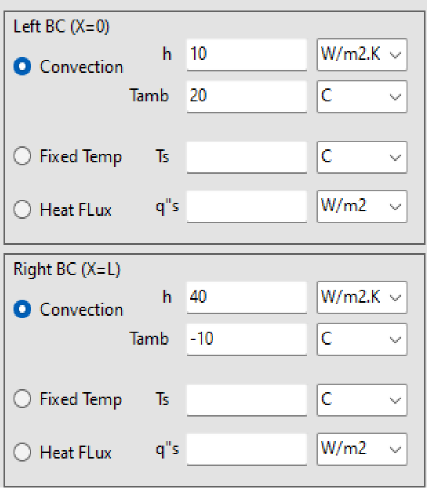
- Select *Plane Wall* (default) and set number of layers to "**1**" (for Case-a) and click *Initialize*.



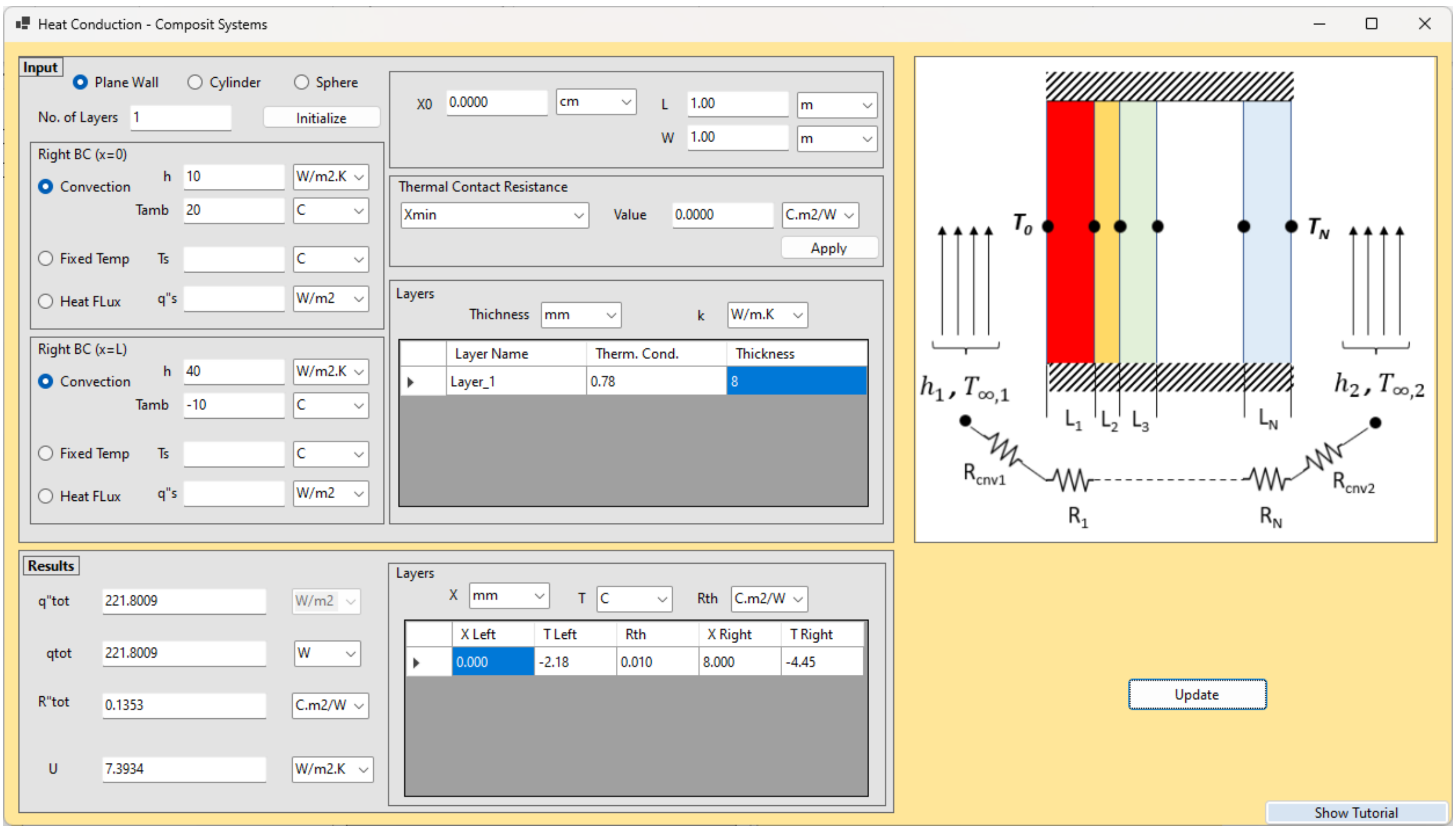
- Set Width and Length to 0.8 m and 1.5 m



- Set values for *Left BC (h=**10** W/m^2.K, T_{amb}=**20** °C)*, and *Right BC (h=**40** W/m^2.K, T_{amb}=**-10** °C)*



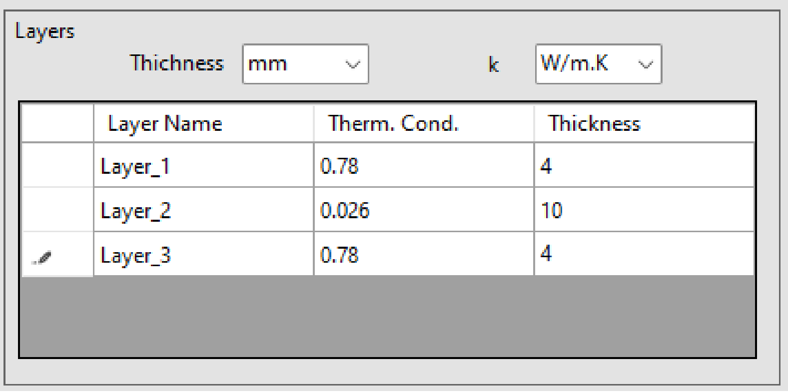
- Change *Layers thickness* unit to **mm**; and for *Layer_1* enter *Therm. Cond*=***0.78*** and *Thickness*=***8***
- Click *Update* to solve. The finished form with results is shown below:



The heat flux is 221.8 W/m^2. The total rate of heat transfer through the window is 266.2 W. The surface temperature at the inside wall is -2.19 °C, as shown under T_{Left} above.

b) Double-pane

- Change No. of Layers to **3** and click Initialize.
- Now, enter thermal conductivity and thickness for all layers, as shown below.

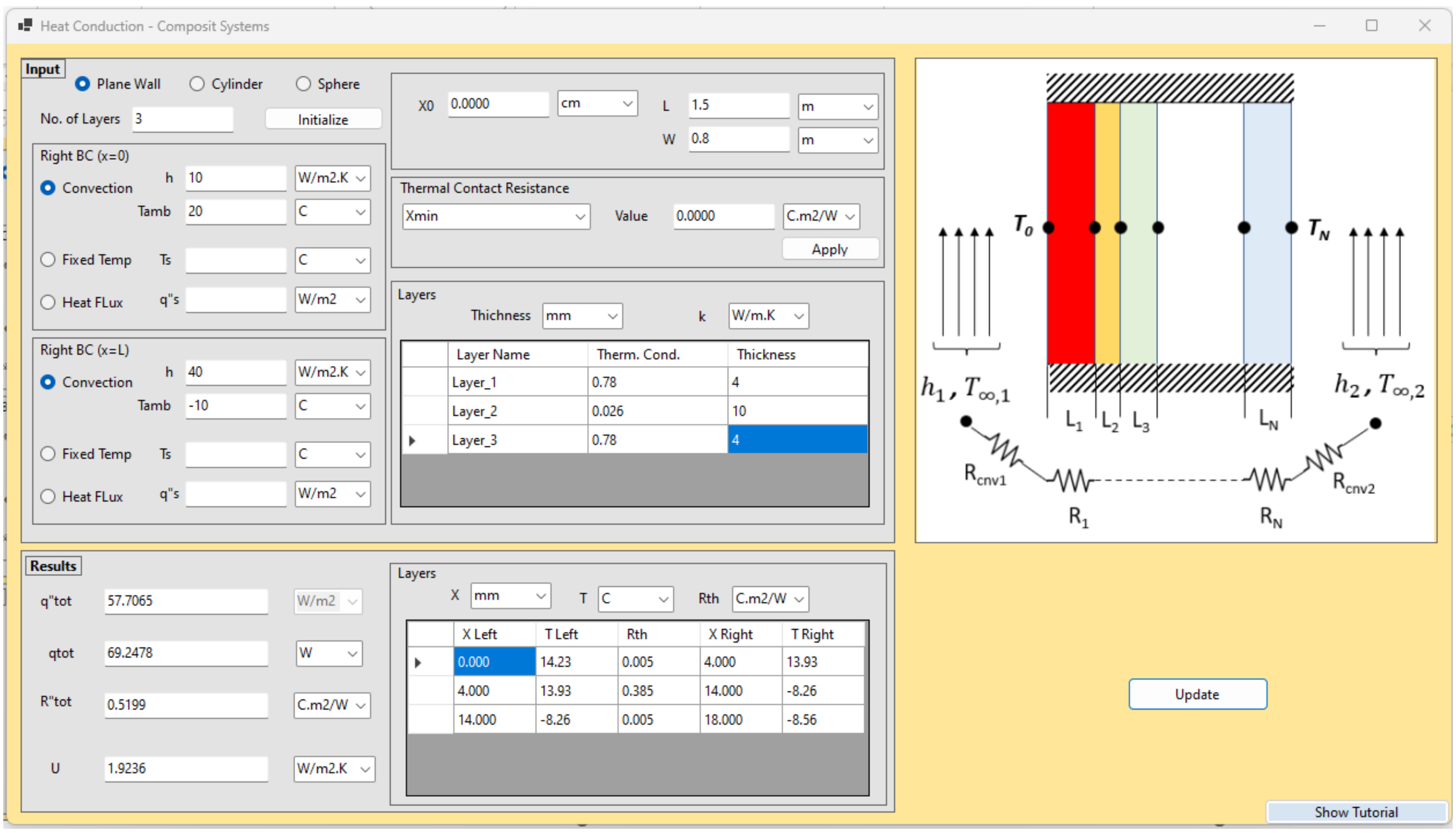


Layers

Thichness mm k W/m.K

Layer Name	Therm. Cond.	Thickness
Layer_1	0.78	4
Layer_2	0.026	10
Layer_3	0.78	4

- Click *Update* to solve. The finished form with results is shown below:



For this case, the heat flux is 57.7 W/m^2, and the rate of heat transfer through the window is 69.24 W. The surface temperature at the inside wall is 14.23 °C, as shown under T_{Left} in the first row above. Note, considerably better performance with the double-pane window.

<<End-of-Tutorial>>

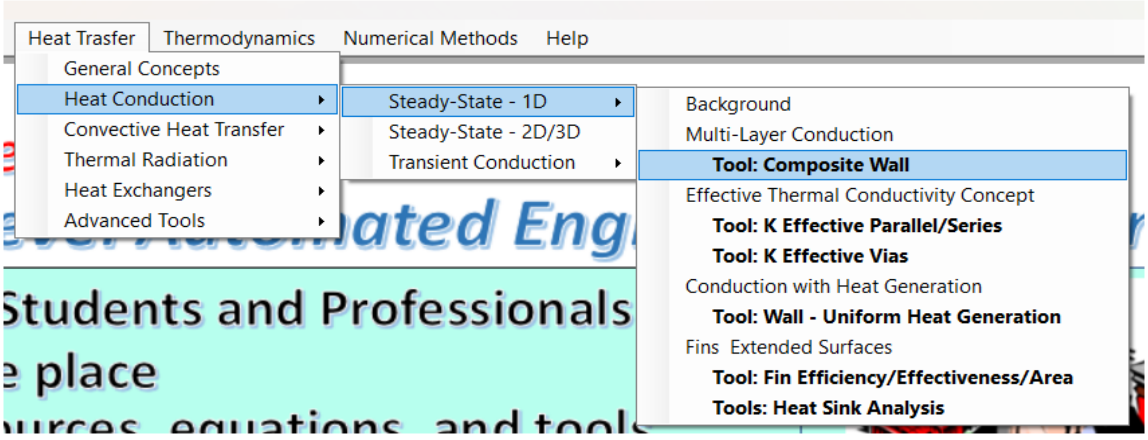
HT-01b:

Example – Composite Cylinder:

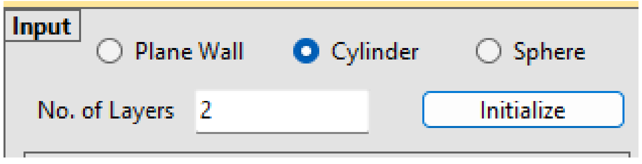
Steam at 320 °C flows in a 5-cm diameter cast iron pipe (k=80 W/m.K) with wall thickness of 0.25 cm. The pipe is covered with 3-cm-thick glass wool insulation (k=0.05 W/m.K). Assuming internal heat transfer coefficient of 60 W/m^2°C and external ambient at 5 °C with heat transfer coefficient of 18 W/m^2°C, determine the rate of heat loss from the pipe per unit length and temperature drops across the pipe shell and the insulation.

Solution:

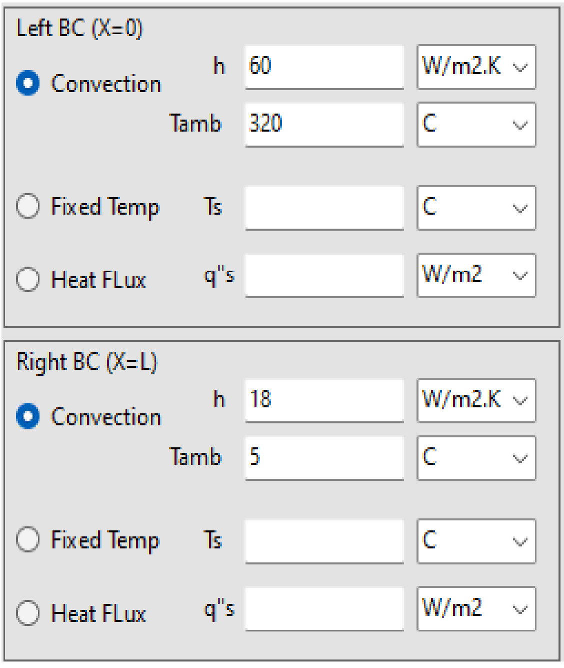
- Open "*Heat Conduction Composite Systems*" Panel:



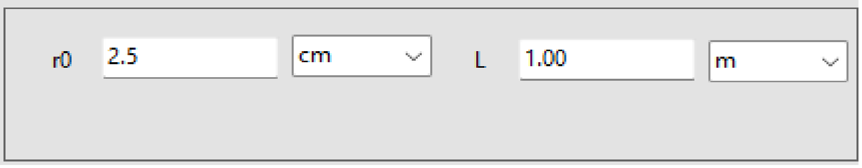
- Select *Cylinder* and set number of layers to "**2**" and click *Initialize*.



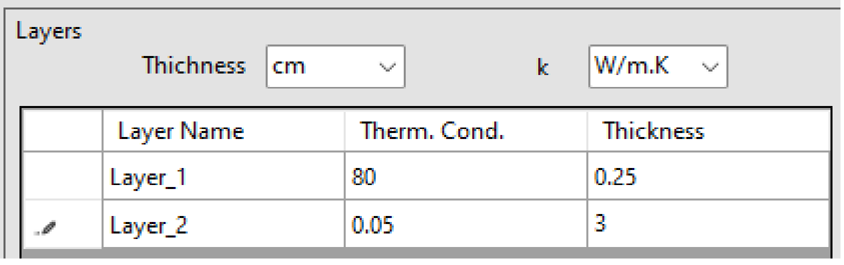
- Set values for *Left BC* (*h=**60** W/m^2.K, T_{amb}=**320** °C*), and *Right BC* (*h=**18** W/m^2.K, T_{amb}=**5** °C*)



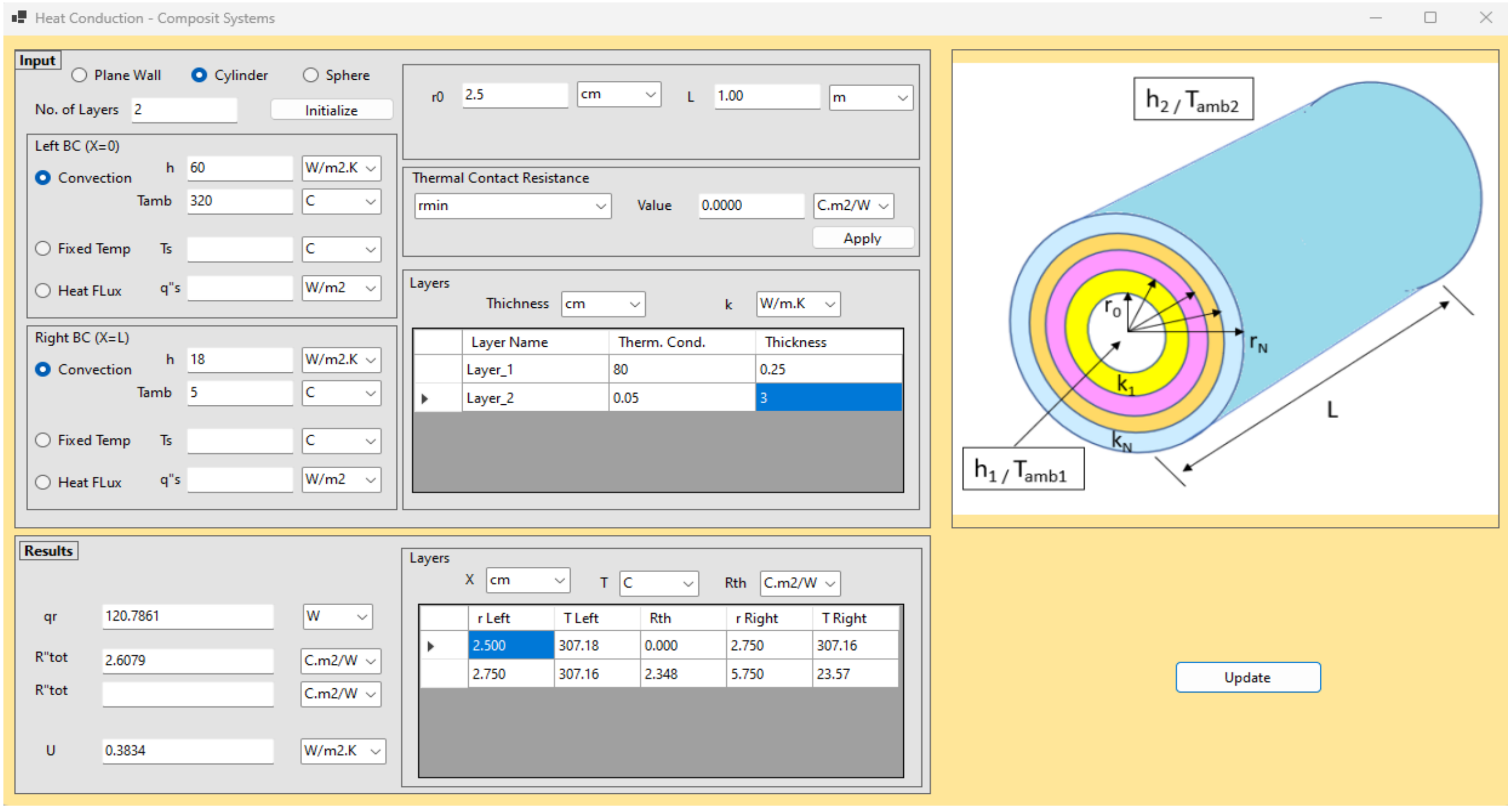
- Set r0=2.5 cm and length, L=1 m (default).



- Change *Layers thickness* unit to **mm**; and for *Layer_1* enter *Therm. Cond=**0.78*** and *Thickness=**8***
- Now, enter thermal conductivity and thickness for all layers, as shown below:



- Click *Update* to solve. The finished form with results is shown below



The rate of heat transfer from the pipe is 120.8 W. The temperature drop across the shell is $307.18 - 307.16 = .02$ °C, and the temperature drop across the insulation is $307.16 - 23.17 = 283.59\ °C$.

<<End-of-Tutorial>>

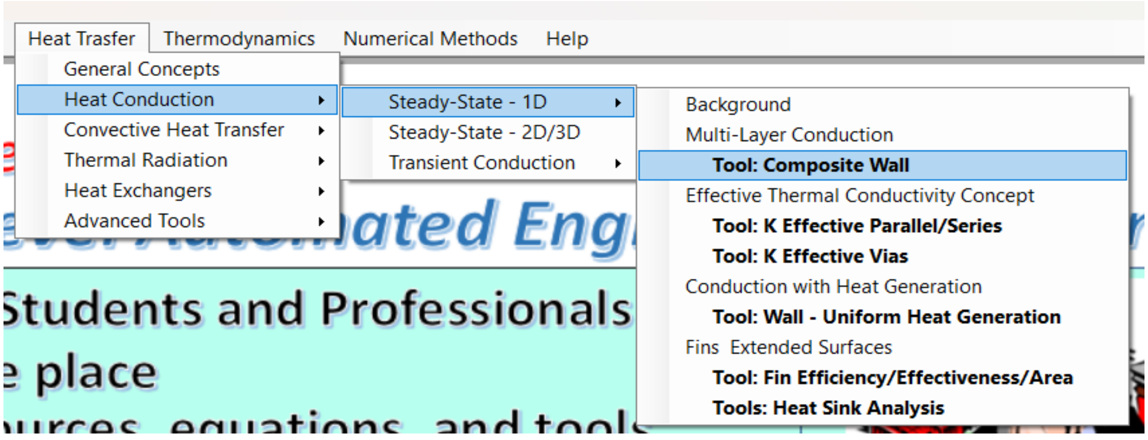
HT-01c:

Example – Composite Sphere:

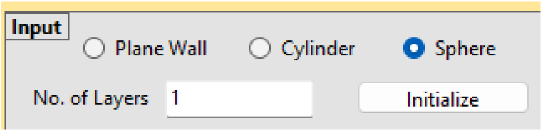
A stainless-steel spherical tank (k=15 W/m.K), with 3 m internal diameter of and 2 cm thickness, is used to store ice water at 0 °C. If the environment is at 22 °C, and assuming 80 W/m²°C and 16 W/m²°C for internal and external heat transfer coefficients, determine the rate of transfer into the container.

Solution:

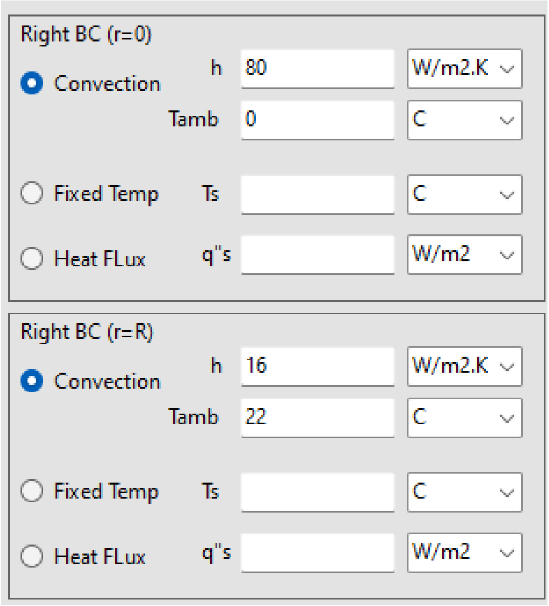
- Open "*Heat Conduction Composite Systems*" Panel:



- Select *Sphere* and set number of layers to "**1**" and click *Initialize*.

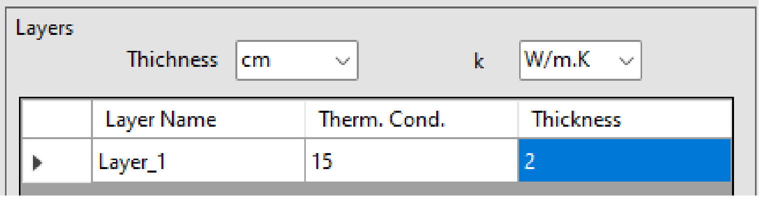


- Set values for *Left BC* (*h=**80** W/m².K, T_{amb}=**0** °C*), and *Right BC* (*h=**16** W/m².K, T_{amb}=**22** °C*)

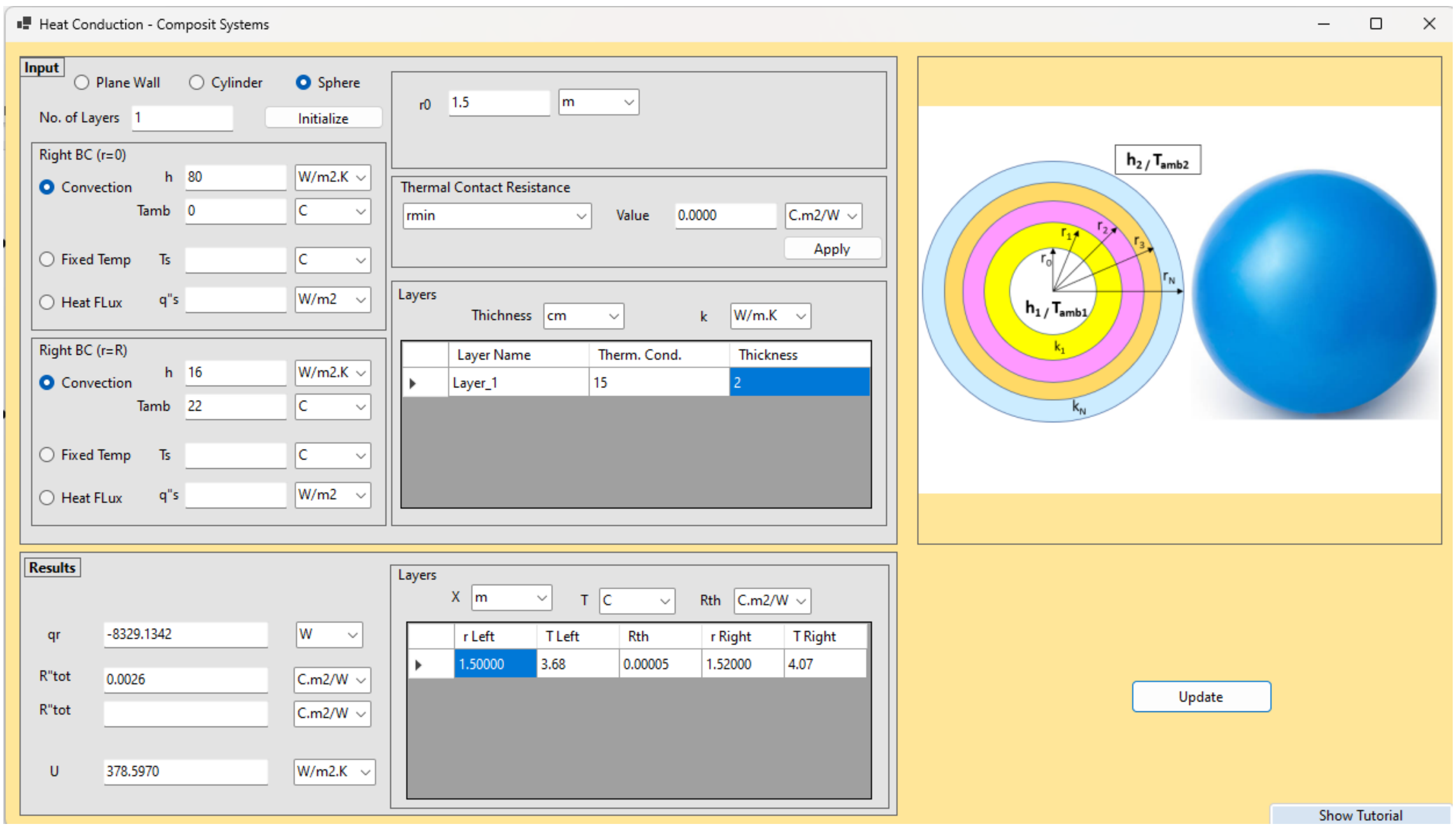


- Set r0=1.5 cm.

- Keep *Layers thickness* unit to **cm**; and for *Layer_1* enter *Therm. Cond*=***15*** and *Thickness*=***3***



- Click *Update* to solve. The finished form with results is shown below



The rate of heat transfer into the pipe is 8329.1W (negative sign indicated heat transfer in -r direction, or into the sphere).

<<End-of-Tutorial>>

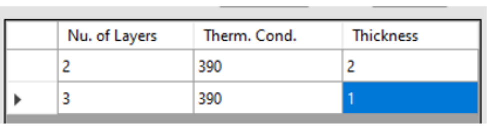
HT-02: K effective Parallel/Series

This tool provides can be used to determine "effective thermal conductivity" for composite layers (such as printed circuit boards), where a number of "thin" planes with relatively high thermal conductivity and embedded in a background material. Two values K_{eff} are provided for "in-plane" (parallel) and thru-plane (normal). These values are commonly used to simplify numerical modeling of complex geometries consisting of multiple layers.

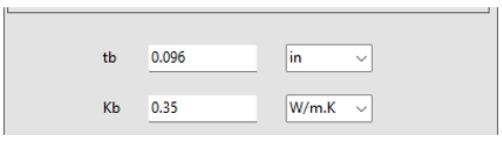
Required Input:

1. **Initialization**
   - Choose the number of "Groups". A group consist of a number of layers with uniform thickness. In PCBs these layers are commonly made of copper, with thickness provided in oz-cu (0.00137 in) or mils (0.001 in).
   - Click Initialize. This will result in creation of rows for each group to define layers.

2. **Definition of Layers**
   - Enter values for
     - Number of layers for each group.
     - Thermal conductivity for these layers (Choose unit from the Combo-Box above).
     - Thickness for each layer (Choose unit from the Combo-Box above).

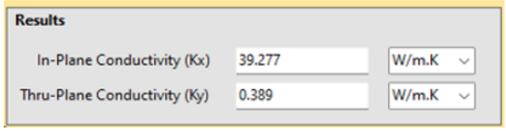


3. **Background (Dielectric) Definition**
   - Enter thermal conductivity and total thickness for the background (board).



Results:

Values for in-plane and normal equivalent thermal conductivities are presented in user-selected units.

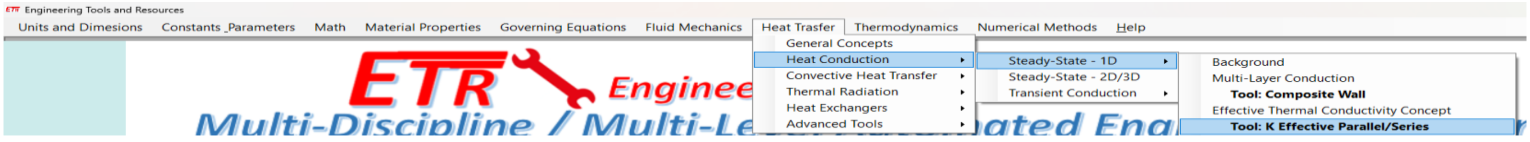


Example:

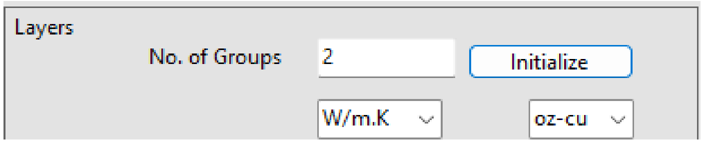
Consider a 0.096-inch-thick PCB, with two 2-oz-cu layers and three 1-oz-cu layers. Determine in-plane and thru-plane effective thermal conductivities for this board. Assume the board is made pf FR4 (k=0.35 W/m.K) and copper conductivity is 390 W/m.K.

Solution:

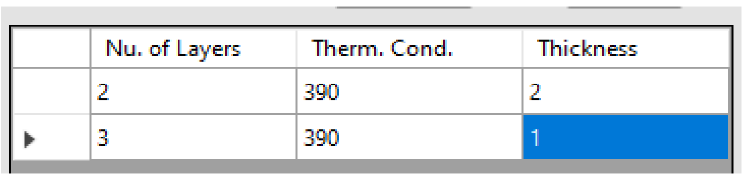
- Open "*Effective Thermal Conductivity*" Panel:



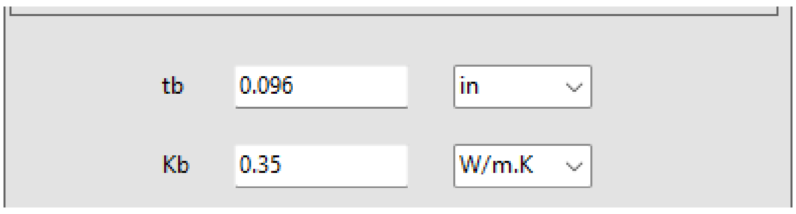
- Enter "**2**" for *No. of Groups* and click *Initialize*.



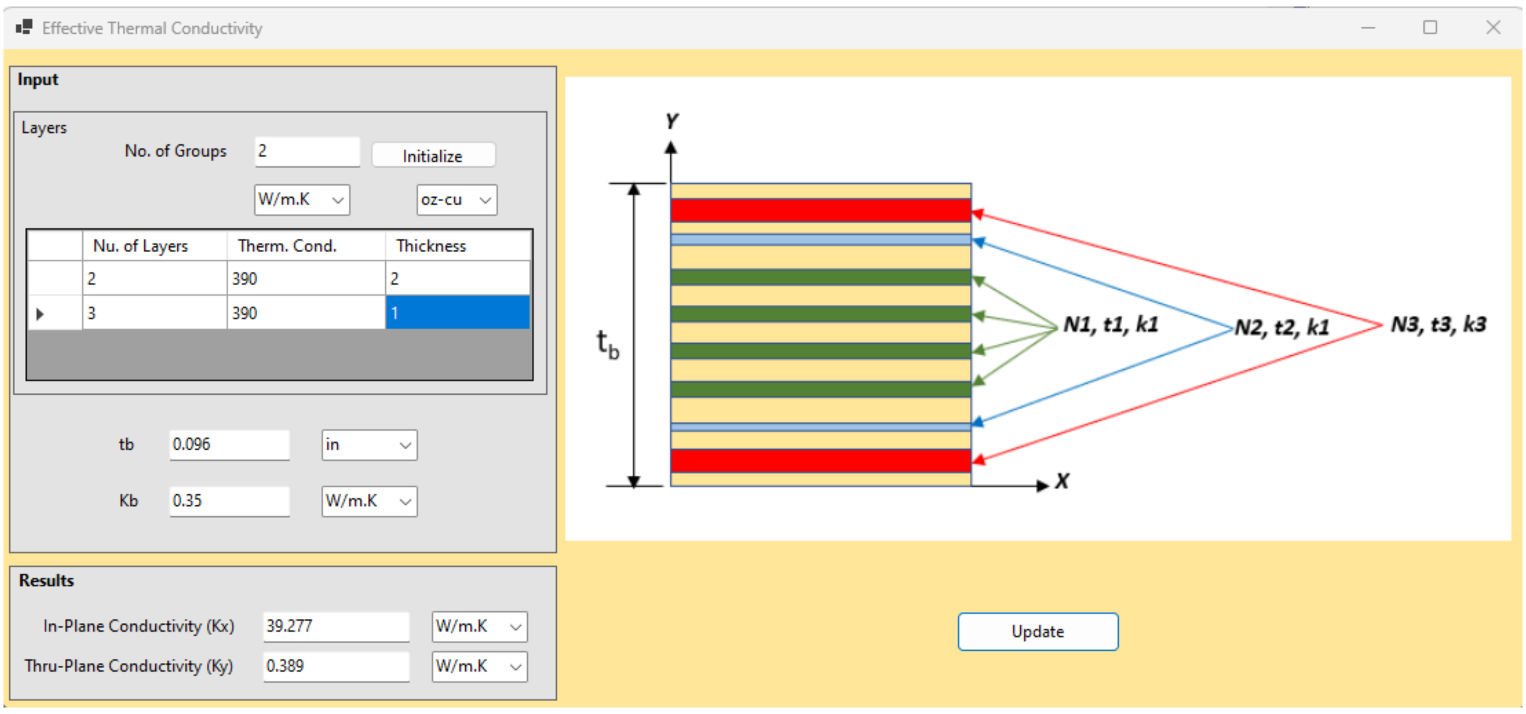
- Set values for each group *(number of layers, thermal conductivity and thickness)*



- Enter values for board thickness and conductivity.



- Click *Update* to solve. The finished form with results is shown below



The in-plane conductivity is 39.28 W/m.K and thru-plane conductivity is 0.39 W/m.K (dominated by board conductivity for series resistance).

<<End-of-Tutorial>>

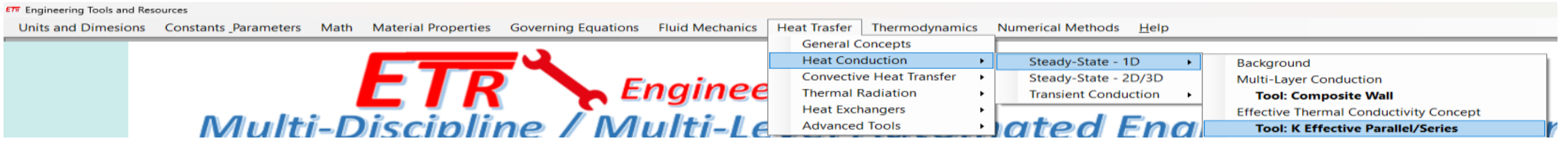
HT-03: K effective Vias

Example: Thermal vias are placed underneath an electronic component to enhance heat transfer through the PCB into a heat sink below. Determine the effective thermal conductivity through the PCB assuming

- Via region: 0.65 in x 0.4 in
- Via inside diameter: 0.3 mm
- Barrel wall thickness = 1 oz-copper
- Via spacing 1 mm.

Solution:

- Open "*Effective Conductivity - Vias*" Panel:



- Choose *Via Spacing* option and enter the following parameters:

Via Inner Diameter = **0.3 mm**

Via Wall Thickness = **1 oz-cu**

Via Conductivity (Kv) = **395 W/m.K** *(Default)*

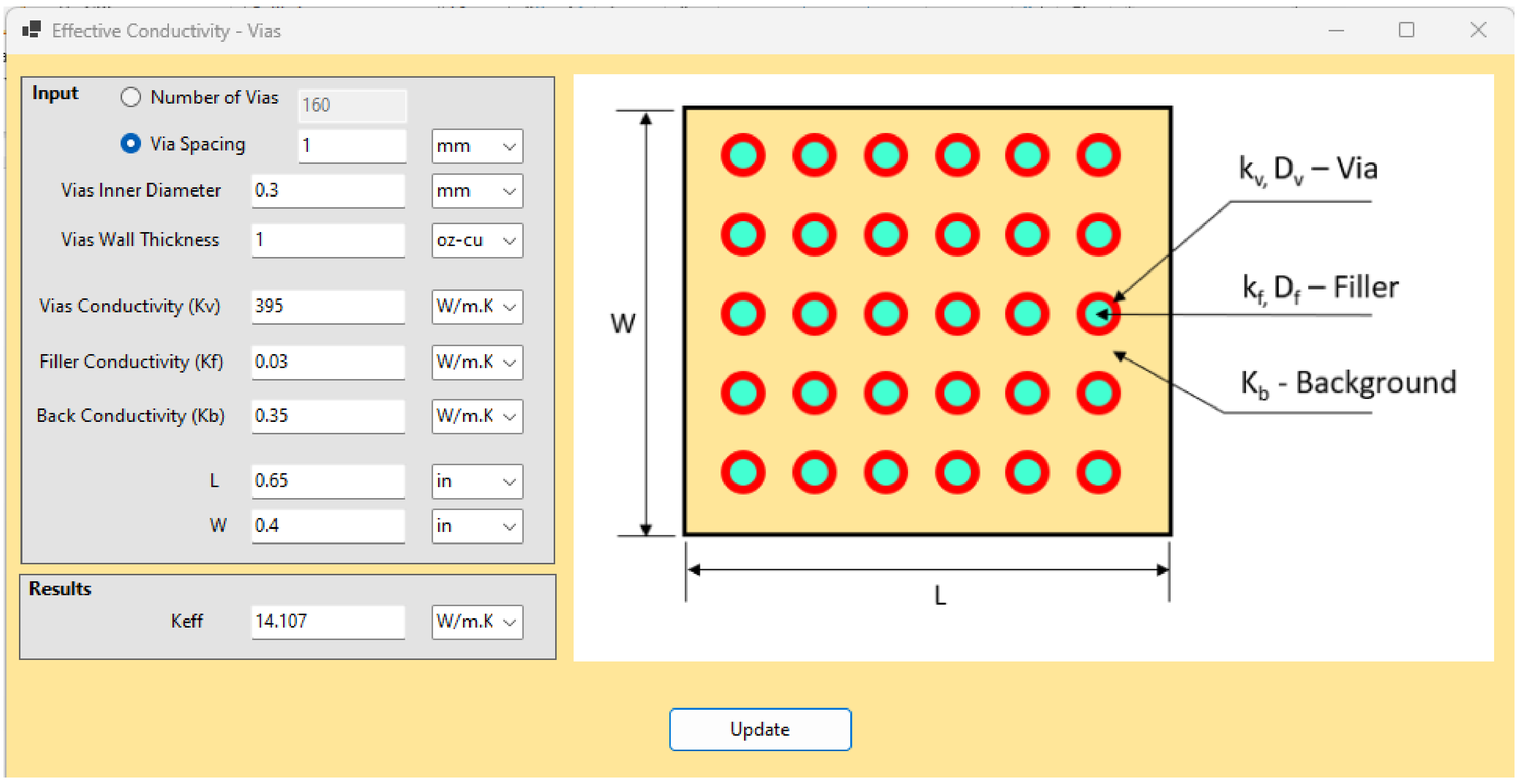
Filler Conductivity (Kf) = **0.03 W/m.K** *(Default)*

Back Conductivity (Kb) = **0.35 W/m.K** *(Default)*

L = **0.65 in**

W = **0.4 in**

Click *Update* to solve. The finished form with results is shown below



The calculated thru-board conductivity is 14.107 W/m.K. Note that this analysis assumes unfilled vias (air conductivity of 0.03 W/m.K is used by default).

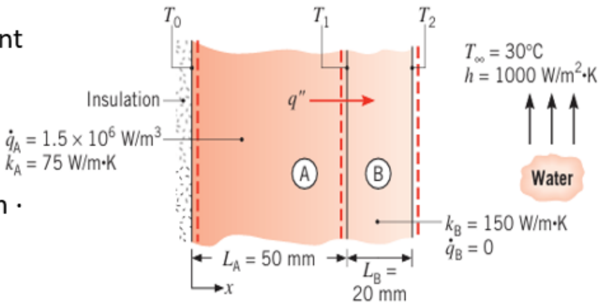
<<End-of-Tutorial>>

HT04: Conduction with Uniform Heat Generation (Walls/Cylinders/Spheres)

Example: Composite wall with Heat Generation: [Source: Bergman-Lavine Example 3.7]

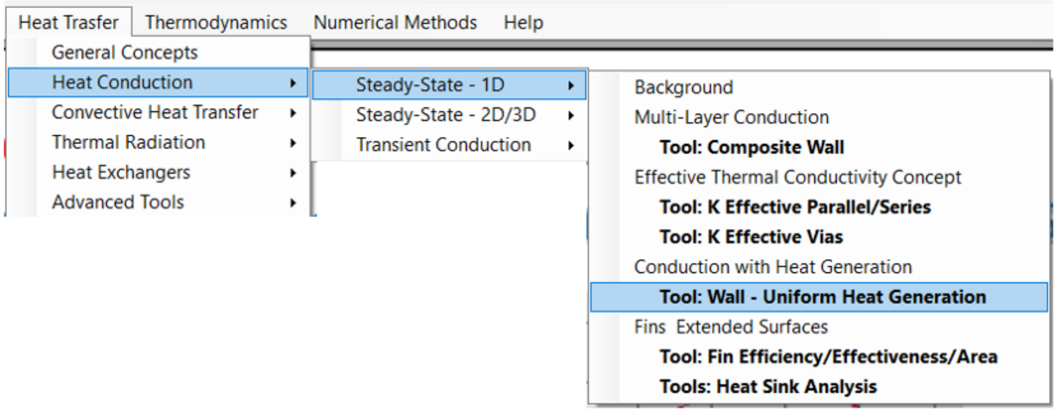
A plane wall of material A has uniform heat generation 1.50 MW/m^3, k_A = 75 W/m · K, and thickness L_A = 50 mm. The outer surface of the wall (larger x value) is cooled by a water stream with a temperature of 30 °C and heat transfer coefficient of 1000 W/m2.K. Determine the inner wall temperature for the following three scenarios.

a) The inner surface is exposed to air at 30 °C with heat transfer coefficient of 250 W/m2.K
b) The Inner surface is insulated.
c) The inner surface is insulated, and the outer surface is attached to another wall with material B that has no generation with k_B = 150 W/m · K and thickness L_B = 20 mm.



Solution:

a) Open "*Solids with Uniform Heat Generation*" Panel:



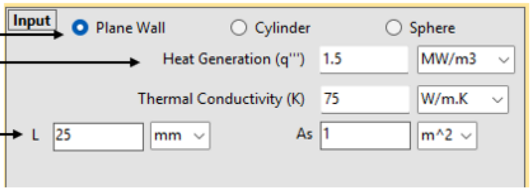
Enter input data

Geometric Parameters:

Geometry: Plane Wall

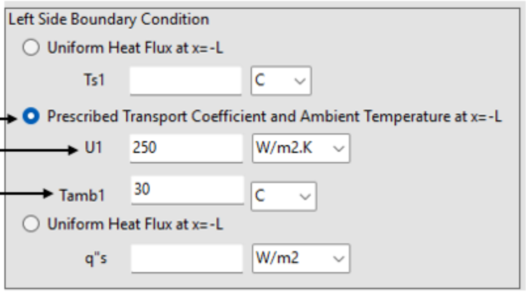
Heat Generation: Note can select 1.5 MW/m3, or 1.50E+06 W/m3

Wall Thickness and Area: Note half thickness must be entered.
Surface area 1.0 m2 is assumed since the value is not given.



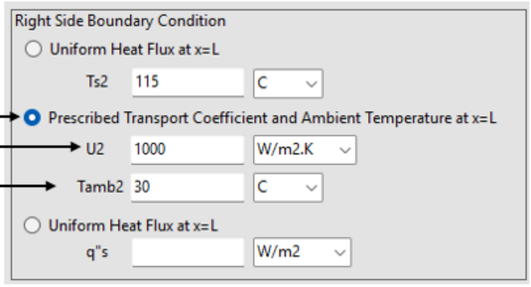
Left side (x=-L) Boundary Conditions:

Select Convective BC.
Enter the value for convection coefficient on the left side.
Enter the value for the ambient temperature on the left side.



Right side (x=L) Boundary Conditions:

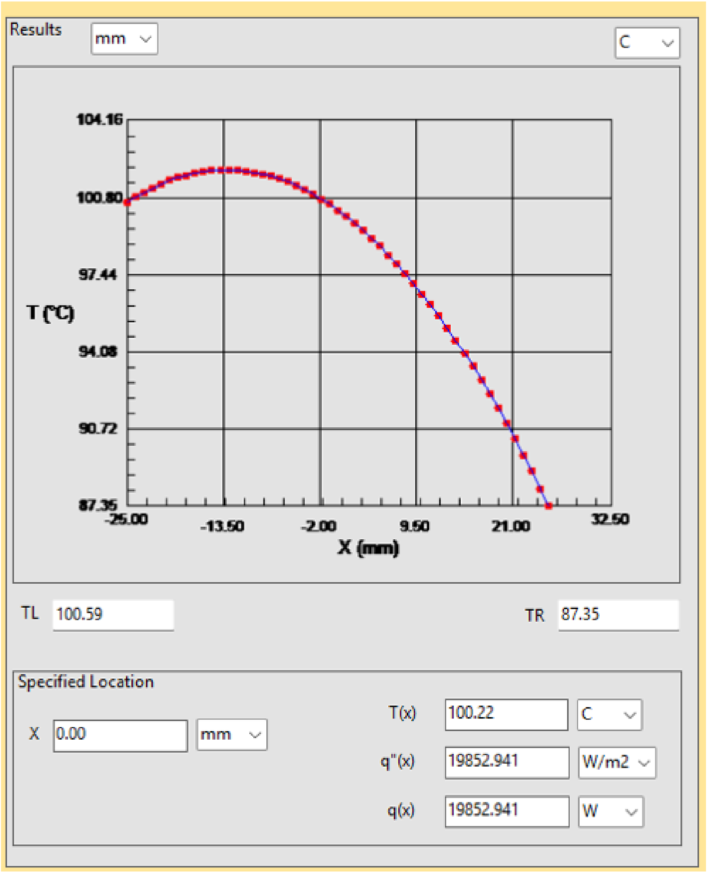
Select Convective BC.
Enter the value for convection coefficient on the right side.
Enter the value for the ambient temperature on the right side.



Results

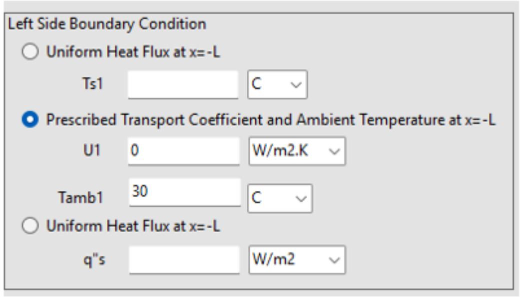
Click Update to solve.

1. *Plot of temperature distribution through the slab is provided.*
2. *Left-side (x=-L) and Right-Side (x=L) temperatures are shown.*
3. *The user can select the units for temperature and length for the plot are from unit boxes at the top.*
4. *Specified Location section allows the user to access local values.*
5. *Heat flux and heat rate values for this problem are identical because surface area is set to 1 m^2.*



b) The Inner surface is insulated

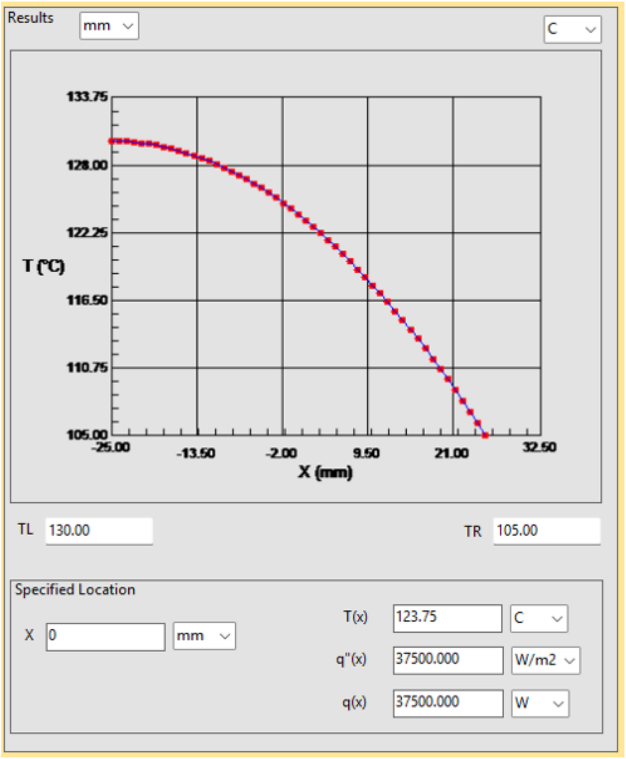
The Only change to be made is to make the left side insulated. This can be done by assigning a value of zero for the overall hea transfer coefficient U1.



Results

Click Update to solve.

Plot of temperature distribution through the slab indicates insulated BC on the left face and updated temperature and heat flux values.



c) The inner surface is insulated, and the outer surface is attached to another wall with material B that has no generation with k_B = 150 W/m · K and thickness L_B = 20 mm.

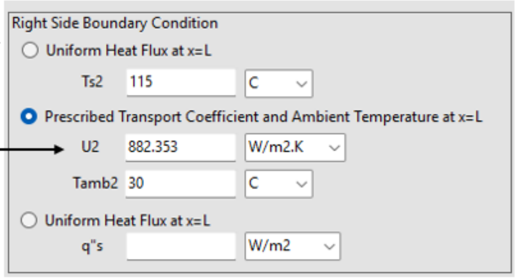
The Only change to be made is to the right-side Boundary by adjusting U2 to account for the additional conductive layer of material B.

This can be done by calculating the total thermal resistance on the right side of the heat generating section by using

- The composite wall tool (this method will be demonstrated on another example using a composite cylinder.
- Calculating the thermal resistance.

$$R"_{th,2} = \frac{L_B}{k_B} + \frac{1}{h_2} = \frac{0.02m}{150\ W/(m.K)} + \frac{1}{1000\ W/(m^2.K)} = 0.00113333\ K.m^2/W$$

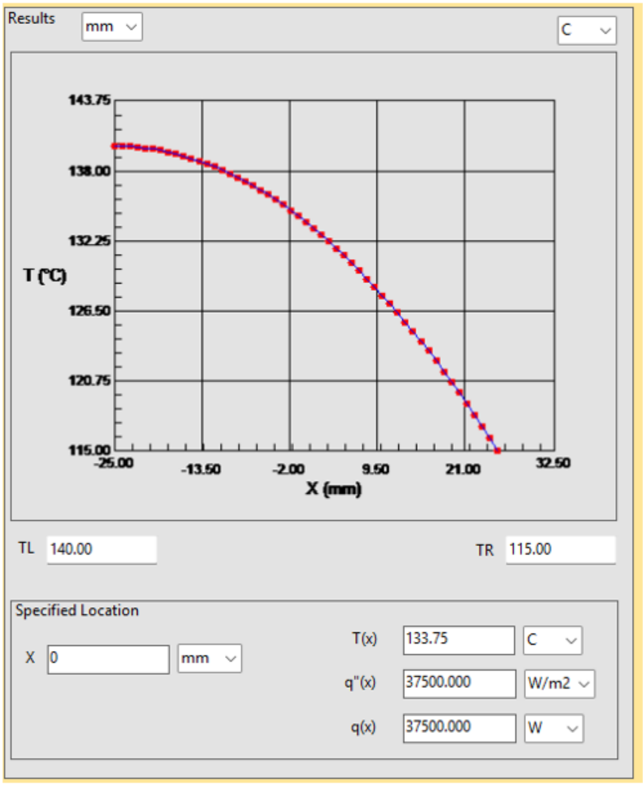
$$U_2 = \frac{1}{R"_{th,2}} = \frac{1}{0.00113333} = 882.353\ W/(m^2.K)$$



Results

Click Update to solve.

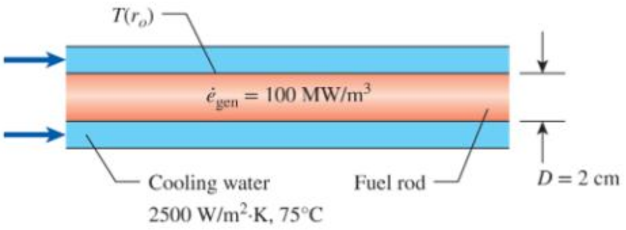
Plot of temperature distribution through the slab indicates insulated BC on the left face and updated temperature and heat flux values.



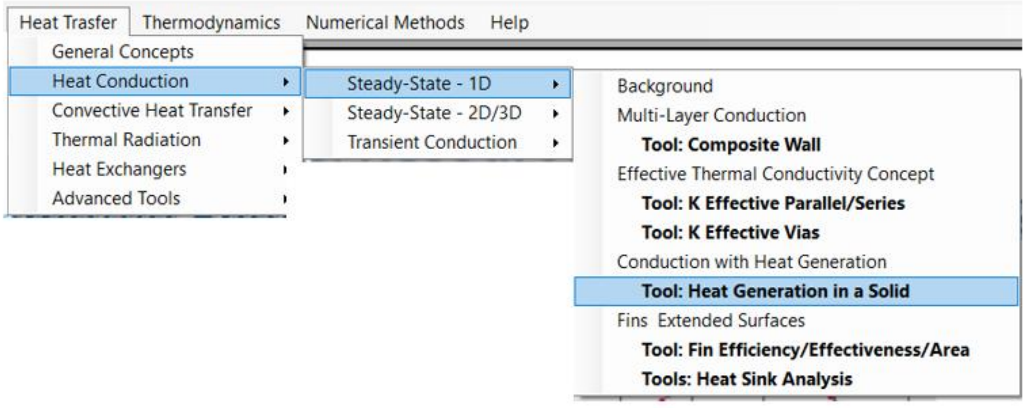
Example: Water-Cooled Fuel Rod [Source: Cengel-Ghagar Problem 2.104]

A cylindrical fuel rod (k = 30 W/m.K) 2 cm in diameter is encased in a concentric tube and cooled by water. The fuel rod generates heat uniformly at a rate of 100 MW/m3, and the average temperature of the cooling water is 75C with a convection heat transfer coefficient of 2500 W/m2.K. The operating pressure of the cooling water is such that the surface temperature of the fuel rod must be kept below 200°C to prevent the cooling water from reaching the critical heat flux (CHF). The critical heat flux is a thermal limit at which a boiling crisis can occur that causes overheating on the fuel rod surface and leads to damage. Determine the variation of temperature in the fuel rod and the temperature of the fuel rod surface. Is the surface of the fuel rod adequately cooled?

Solution:



Open "*Heat Generation in a Solid*" Panel

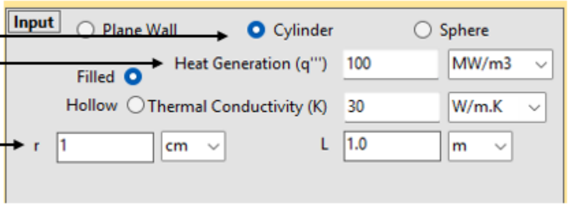


Enter input data

Geometric Parameters:

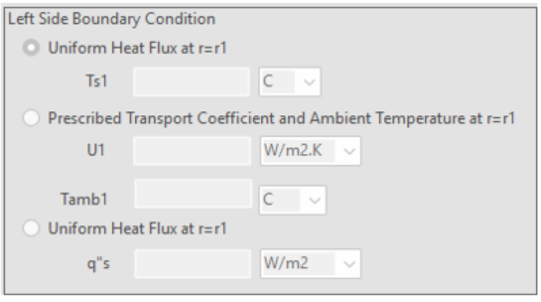
Geometry: Cylinder and select "Filled" option
Heat Generation: Enter 100 and change unit to MW/m3
Enter thermal conductivity 30 W/m.K

Radius: Enter 1 cm (diameter is given as 2 cm).
Length 1.0 m is assumed since the value is not given.



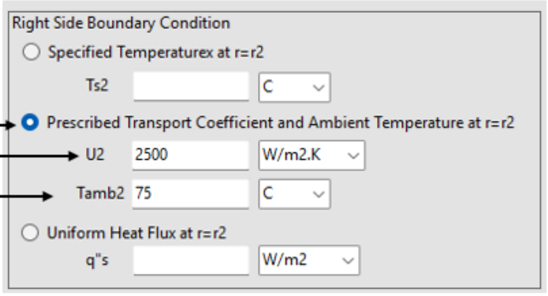
Left side (r=r1) Boundary Conditions:

This panel is disabled since we have a solid (filled) cylinder.



Right side (r=r2) Boundary Conditions:

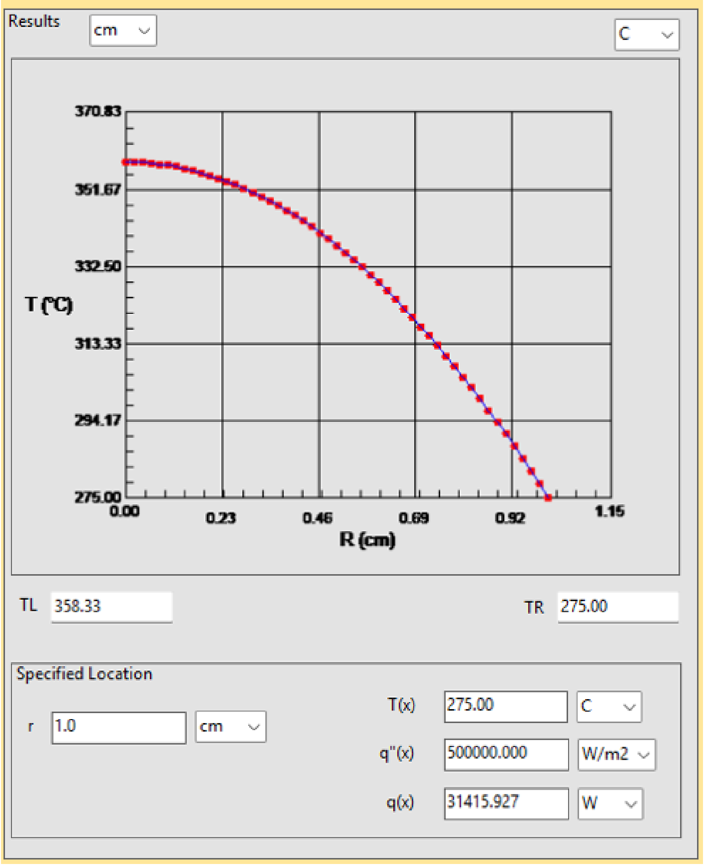
Select Convective BC.
Enter the value for convection coefficient on the right side.
Enter the value for the ambient temperature on the right side.



Results

Click Update to solve.

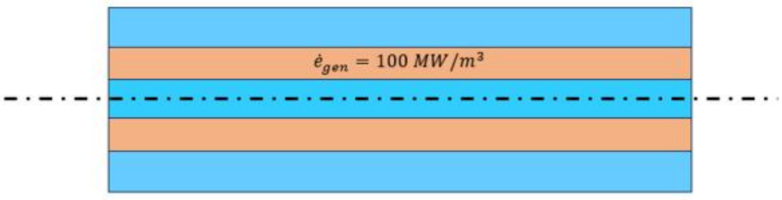
1. *Plot of temperature distribution through the slab is provided.*
2. *Left-side (centerline) and Right-Side (r=1 cm) temperatures are shown.*
3. *The user can select the units for temperature and length for the plot are from unit boxes at the top.*
4. *The surface temperature of 275 °C is 75°C higher than the temperature necessary to prevent the cooling water from reaching the CHF. So, the fuel rod is not adequately cooled.*
5. *The center of the rod is at 358.3 °C.*
6. *Specified Location section allows the user to access local values. Here r=1 cm points to the rod surface.*
7. *The surface heat flux is 500,000 W/m2. And the total heat is calculated to be 31,415.9 W for a 1-mete-long section of the rod.*



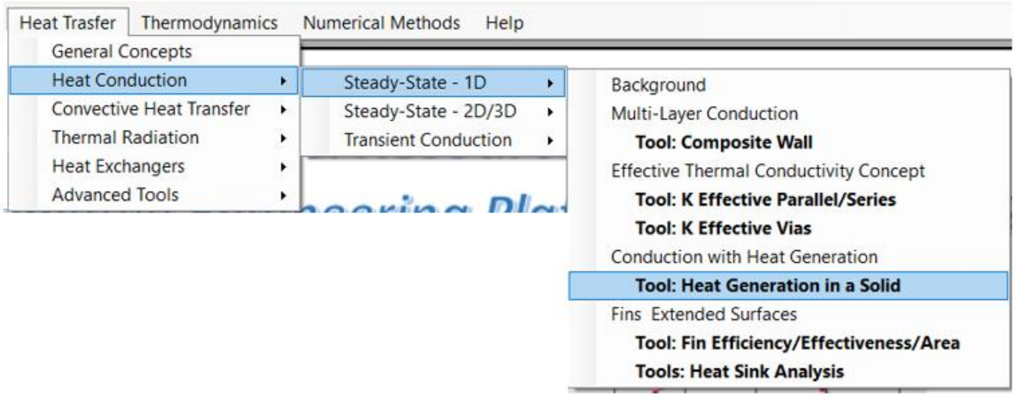
Example: Water-Cooled Fuel Rod [Source: Cengel-Ghagar Problem 2.104 - Modified]

Repeat the above example with added inner cooling channel of 2 cm, cooling water at 75 °C with a convection heat transfer coefficient of 2500 W/m^2.K. Assume the fuel rod has an inner radius of 1 cm and the outer radius determined to keep the total volume (and therefore, heat dissipation) unchanged. Assume the external surface of the rod is exposed to an identical convective environment.

Solution:



Open "*Heat Generation in a Solid*" Panel:



To determine the outer diameter of the hollow fuel rod, we set the volume equal to the volume of the solid rod.

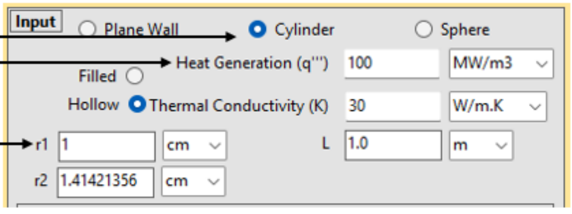
$$\pi r^2 L = \pi(r_2^2 - r_1^2)L \rightarrow r_2^2 = r^2 + r_1^2 \rightarrow r_2\sqrt{r^2 + r_1^2}$$

$$r_1 = 1\ cm; r = 1\ cm \quad \Rightarrow \quad r_2 = \sqrt{2} = 1.41421356 \text{ cm}$$

Enter input data

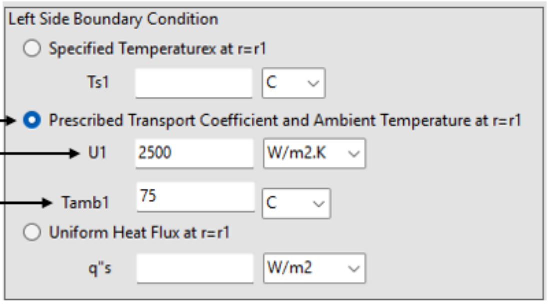
Geometric Parameters:

Geometry: Cylinder and select "Hollow" option
Heat Generation: Enter 100 and change unit to MW/m3
Enter thermal conductivity 30 W/m.K
Radius: Enter r1=1 cm; $r_2 = \sqrt{2} = 1.41421356$ cm



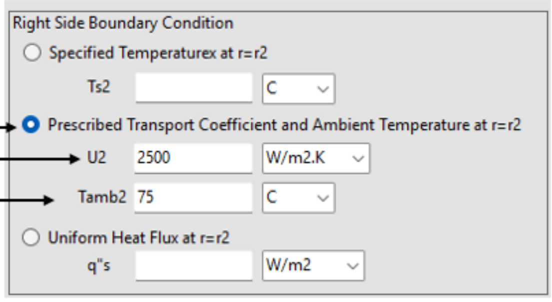
Left side (r=r1) Boundary Conditions:

Select Convective BC.
Enter the value for convection coefficient on the inner surface.
Enter the value for the ambient temperature on the inner surface.



Right side (r=r2) Boundary Conditions:

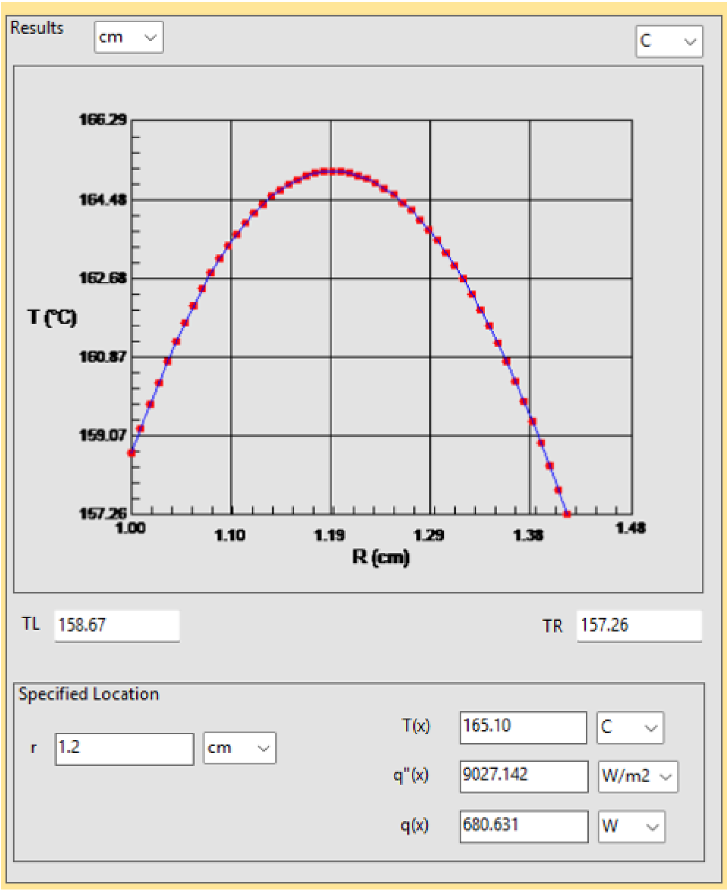
Select Convective BC.
Enter the value for convection coefficient on the outer surface.
Enter the value for the ambient temperature on the outer surface.



Results

Click Update to solve.

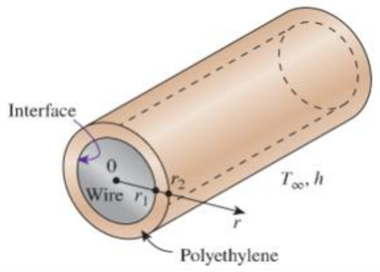
1. *Plot of temperature distribution through the slab is provided.*
2. *Left-side (centerline) and Right-Side (r=1 cm) temperatures are shown.*
3. *The user can select the units for temperature and length for the plot are from unit boxes at the top.*
4. *The surface temperature of 158.57 °C and 157.26 °C are well below 200 °C. So, the fuel rod is adequately cooled.*
5. *Specified Location section allows the user to access local values. Here r=1.2 cm points to the max temperature in the rod.*
6. *The temperature at this location is 165.10 °C, heat flux is 9,027.9 W/m2. And the total heat is calculated to be 680.6 W for a 1-mete-long section of the rod.*



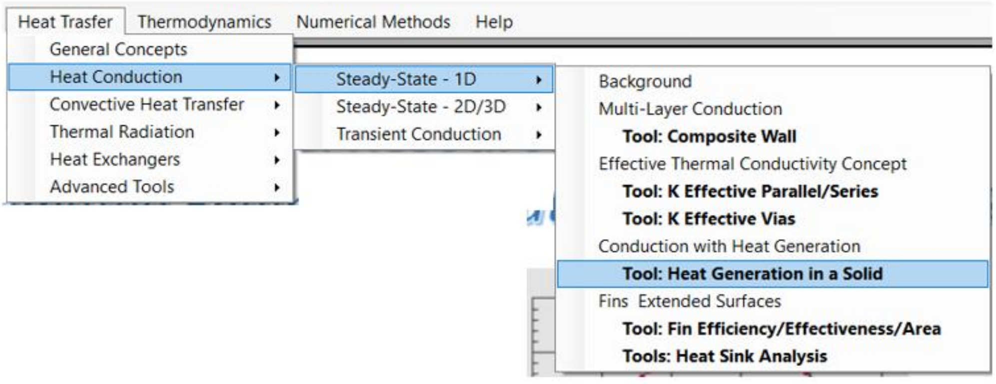
Example: Electrical Resistance wire with Insulation [Source: Cengel-Ghagar Problem 2.105 - Modified]

A long electrical resistance wire of radius r1 = 0.2 cm has a thermal conductivity k_w= 15 W/m.K. Heat is generated uniformly in the wire as a result of resistance heating at a constant rate of 1.2 W/cm^3. The wire is covered with polyethylene insulation with a thickness of 0.5 cm and thermal conductivity of k_{ins} = 0.4 W/m.K. The outer surface of the insulation is subjected to convection and radiation with the surroundings at 20 °C. The combined convection and radiation heat transfer coefficients is 7 W/m^2.K. Determine the temperature at the interface of the wire and the insulation and the temperature at the center of the wire. The ASTM D1351 standard specifies that thermoplastic polyethylene insulation is suitable for use on electrical wire with operation at temperatures up to 75C. Under these conditions, does the polyethylene insulation for the wire meet the ASTM D1351 standard?

Solution:



Open "*Heat Generation in a Solid*" Panel:

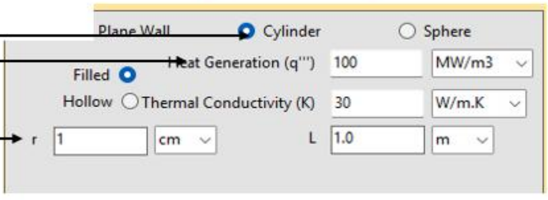


Enter input data

Geometric Parameters:

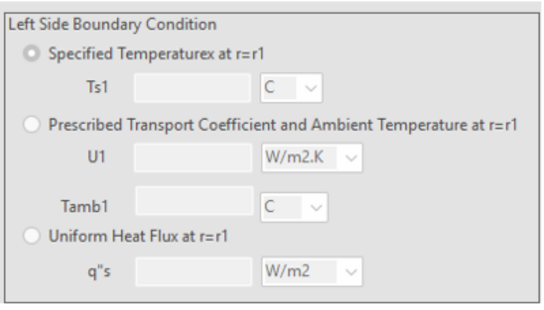
Geometry: Cylinder and select "Filled" option
Heat Generation: Enter 100 and change unit to MW/m3
Enter thermal conductivity 30 W/m.K

Radius: Enter 1 cm (diameter is given as 2 cm).
Length 1.0 m is assumed since the value is not given.



Left side (r=r1) Boundary Conditions:

This panel is disabled since we have a solid (filled) cylinder.



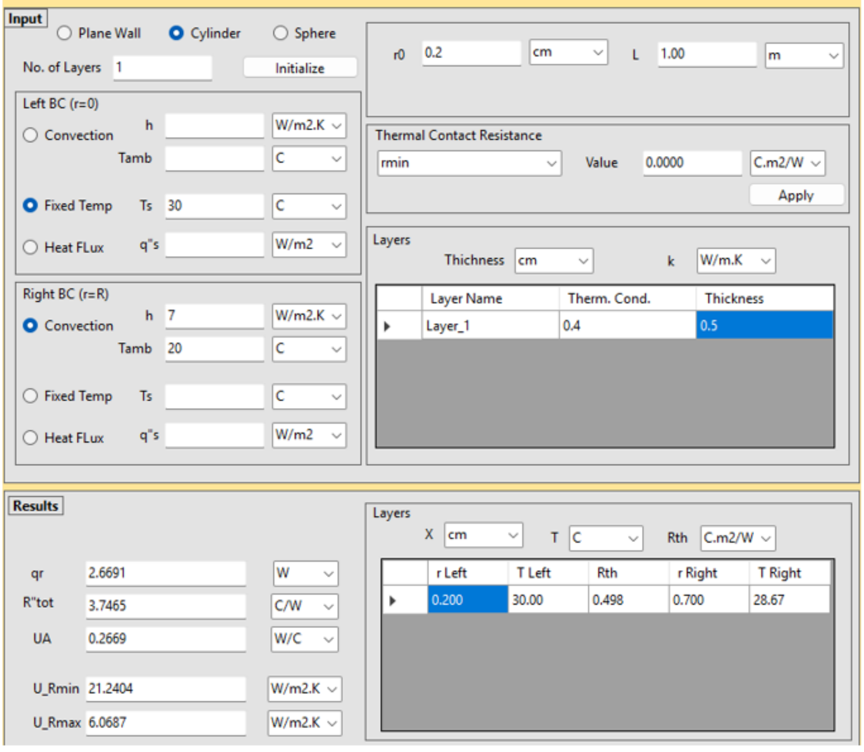
Right side (r=r2) Boundary Conditions:

The external surface of the wire is exposed to the ambient air through an intervening Polyethylene layer. Therefore, the transport coefficient U_2 must account for both convective and conductive effects. The overall heat transfer coefficient can be found by first calculating the total thermal resistance for the combined system and then determining the UA and U2:

$$R_{th,2} = \frac{ln\left(\frac{r_2}{r_1}\right)}{2\pi L k} + \frac{1}{(2\pi r_2 L)h_2} \Longrightarrow UA = \frac{1}{R_{th,2}} \Longrightarrow U_2 = \frac{UA}{(2\pi r_1 L)}$$

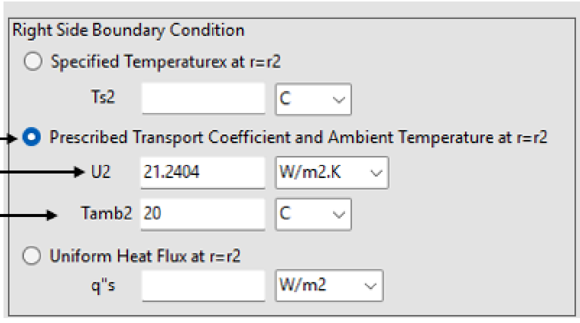
Alternatively, the Composite Solid tool may be used to evaluate U_2 automatically.

- Select Cylinder option.
- Set No. of Layer to "1"
- Click Initialize to set the form.
- Set r0 to 0.2 cm
- In the Layers table set the thermal conductivity to 0.4 W/m.K and thickness to 0.5 cm.
- In the left BC panel: set Ts to any temperature (30 °C). Note that the value of temperature does not matter since we are only interested in R_{th} calculations.
- In the right BC panel: set h to 7 W/m^2.K and T_{amb} to 30 °C.
- Click Update to solve.
- The UA value is calculated to be 0.2669 C/W
- The U_Rmin value of **21.24 W/m^2.K** is the value we need for the analysis; it is the U value using the radius of 0.2 cm.



Now, we get back to "*Heat Generation in a Solid" and enter the calculated vale of U2 in the Right-Side BC Panel.*

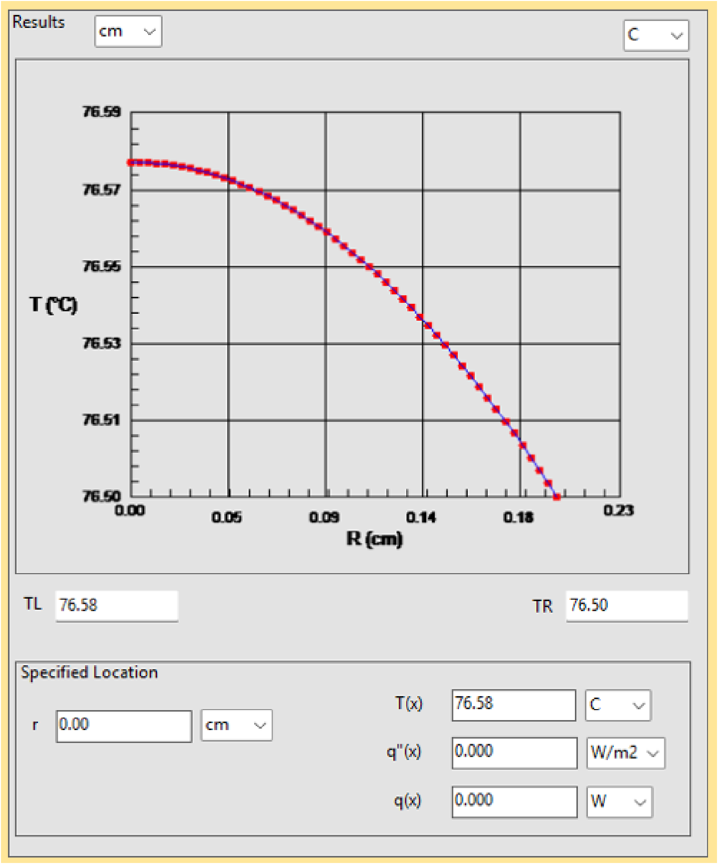
Select Convective BC.
Enter the value for convection coefficient on the outer surface.
Enter the value for the ambient temperature on the outer surface.



Results

Click Update to solve.

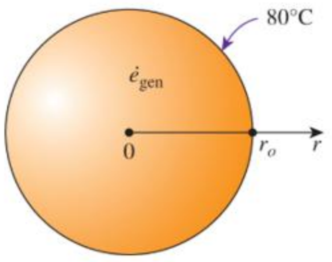
1. *Plot of temperature distribution through the slab is provided.*
2. *Left-side (centerline) and Right-Side (r=1 cm) temperatures are shown.*
3. *The user can select the units for temperature and length for the plot are from unit boxes at the top.*
4. *The surface temperature of 76.5 °C is 1.5 °C higher than the specification of the ASTM D1351 standard for polyethylene insulation.*
5. *There is very little temperature rise in the wire, as the center temperature is less than 0.1 °C higher than the interface temperature.*



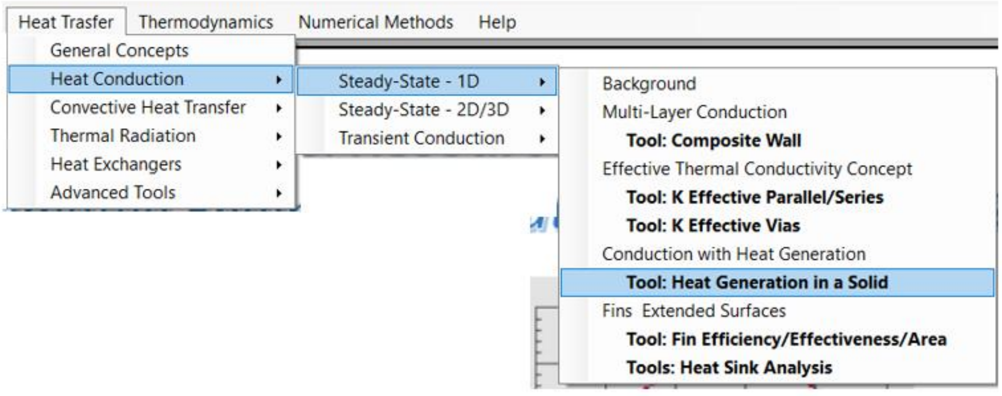
Example:

Consider a homogeneous spherical piece of radioactive material of radius r_o = 0.04 m that is generating heat at a constant rate of e_{gen} = 4 x 10^7 W/m^3. The heat generated is dissipated to the environment steadily. The outer surface of the sphere is maintained at a uniform temperature of 80C, and the thermal conductivity of the sphere is k = 15 W/m.K. Assuming steady one-dimensional heat transfer, determine the temperature at the center of the sphere.

Solution:



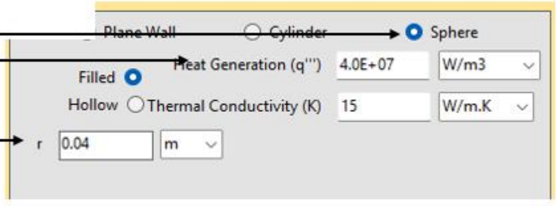
Open "*Heat Generation in a Solid*" Panel:



Enter input data

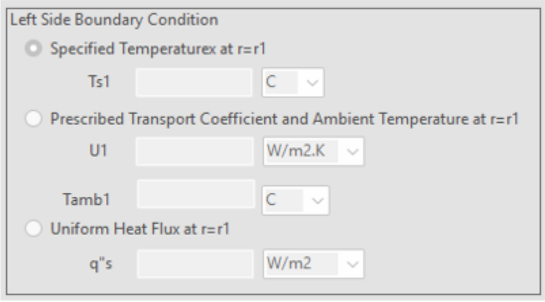
Geometric Parameters:

Geometry: Sphere and select "Filled" option
Heat Generation: Enter 4 × 10^7 W/m3
Enter thermal conductivity 15 W/m.K
Radius: Enter 0.04 m.
Length 1.0 m is assumed since the value is not given.



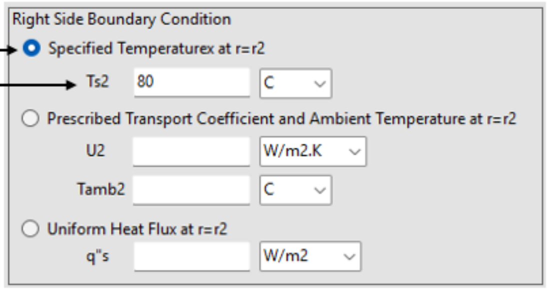
Left side (r=r1) Boundary Conditions:

This panel is disabled since we have a solid (filled) cylinder.



Right side (r=r2) Boundary Conditions:

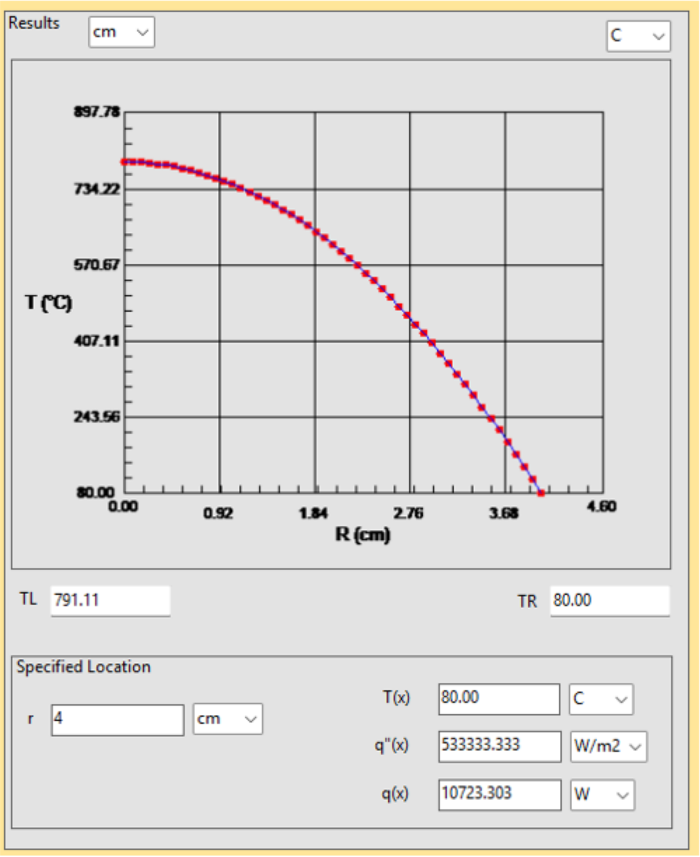
Select Specified Temperature.
Enter 80 °C.



Results

Click Update to solve.

1. *Plot of temperature distribution through the slab is provided.*
2. *Left-side (center) and Right-Side (r=4 cm) temperatures are shown.*
3. *The user can select the units for temperature and length for the plot are from unit boxes at the top.*
4. *The temperature at the center of the sphere is calculated to be 791.1 °C.*
5. *The surface heat flux is 533,333.3 W/m2 and the total heat is calculated to be 10,723.3 W.*

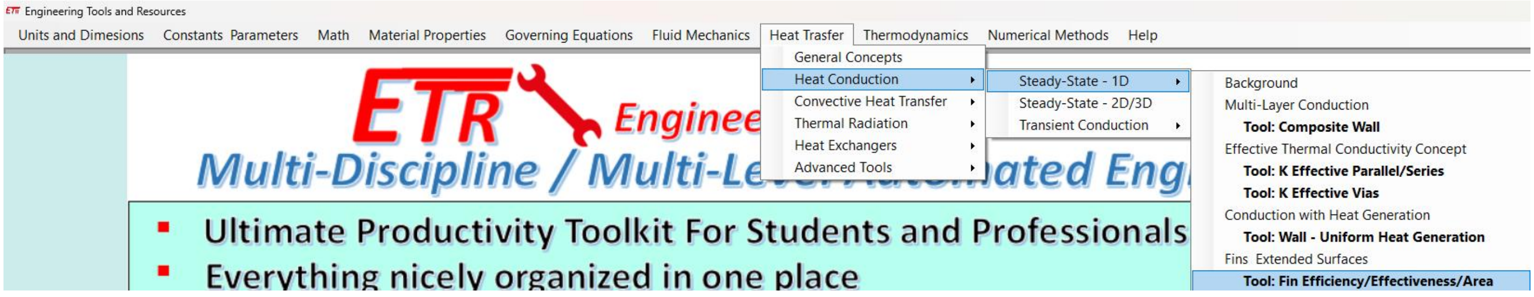


HT-05: Fin Efficiency/Effectiveness

Example: A heat sink utilizes aluminum 2024-T6 (k = 186 W/m · K) pin fins of parabolic profile with blunt tips. Each fin has a length of 20 mm and a base diameter of 5 mm. If the fins are in an environment with heat transfer coefficient of 20 W/m2 · K, determine the efficiency and effectiveness for each fin.

Solution:

- Open "*Fin Efficiency*" Panel:



- Choose ***f) Cone with blunt tip*** from the fin type combo list
- Enter the following parameters:

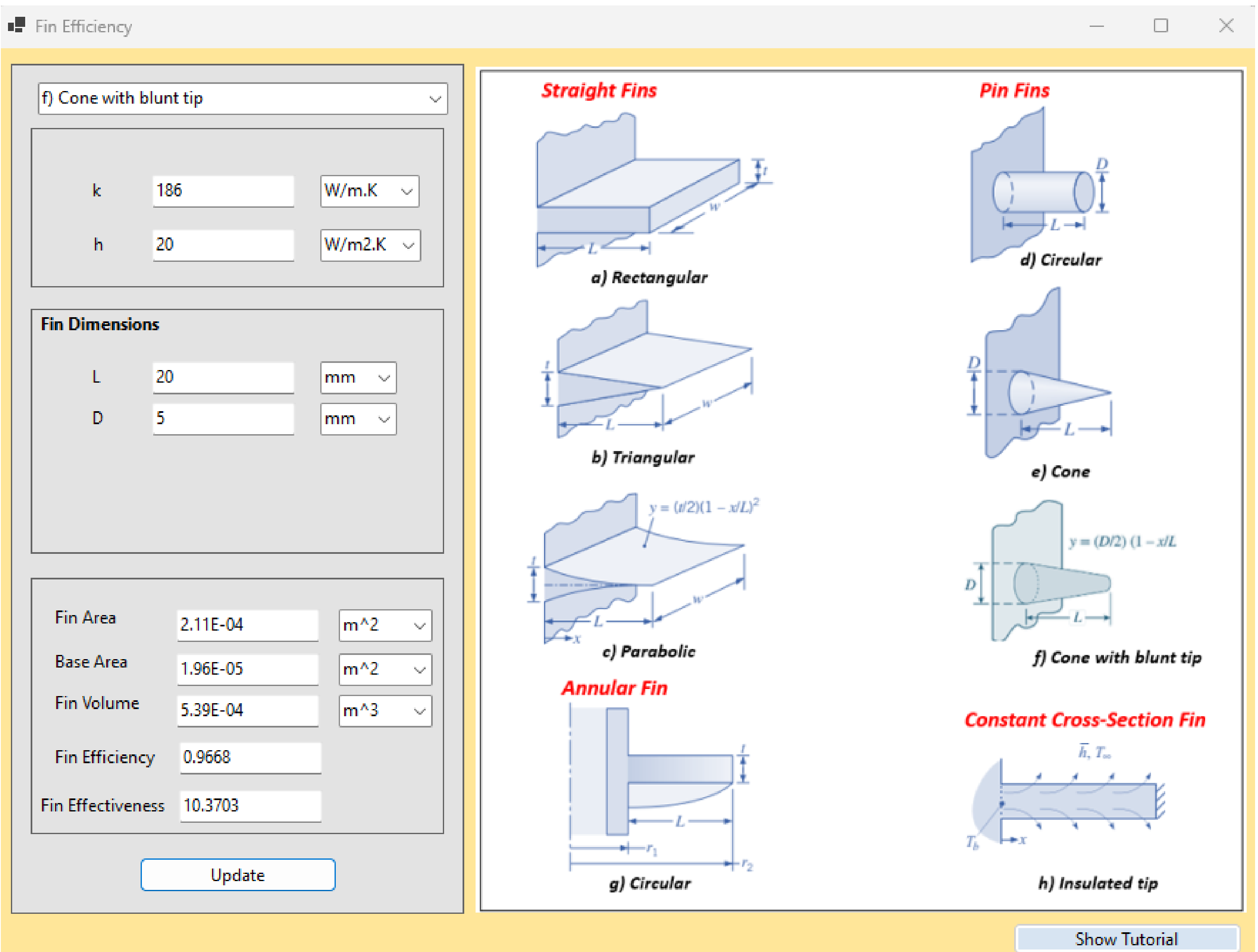
k = **186 W/m.K**

h = **20 W/m2 · K**

L = **20 mm**

D= **5 mm**

- Click *Update* to solve. The finished form with results is shown below



The fin efficiency is 96.67% and the effectiveness is calculated to be 10.37.

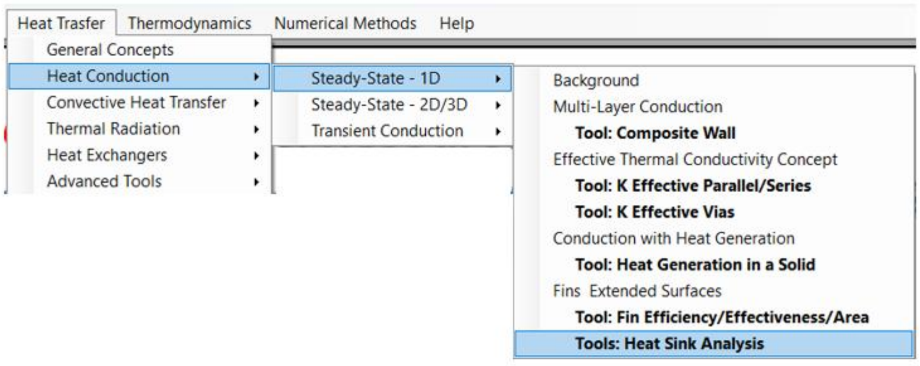
HT-06: Heat Sink Analysis

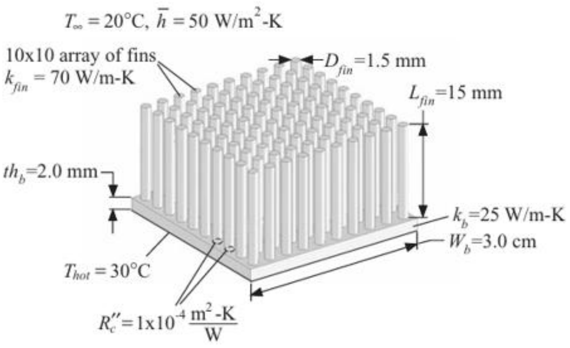
Example – THERMOELECTRIC HEAT SINK [Source: Nellis & Klein Example 1.6-2]

Heat rejection from a thermoelectric cooling device is accomplished using a 10 × 10 array of 1.5 mm diameter pin fins that are 15 mm long. The fins are attached to a square base plate that is 3 cm on each side and 2 mm thick, as shown below. The conductivity of the fin material is 70W/m-K and the thermal conductivity of the base material is 25W/m-K. There is a contact resistance of 1×10−4 m^2-K/W at the interface between the base of the fins and the base plate. The hot end of the thermoelectric cooler is at 30◦C and the surrounding air temperature is 20◦C. The average heat transfer coefficient between the air and the surface of the heat sink is h = 50 W/m2-K.

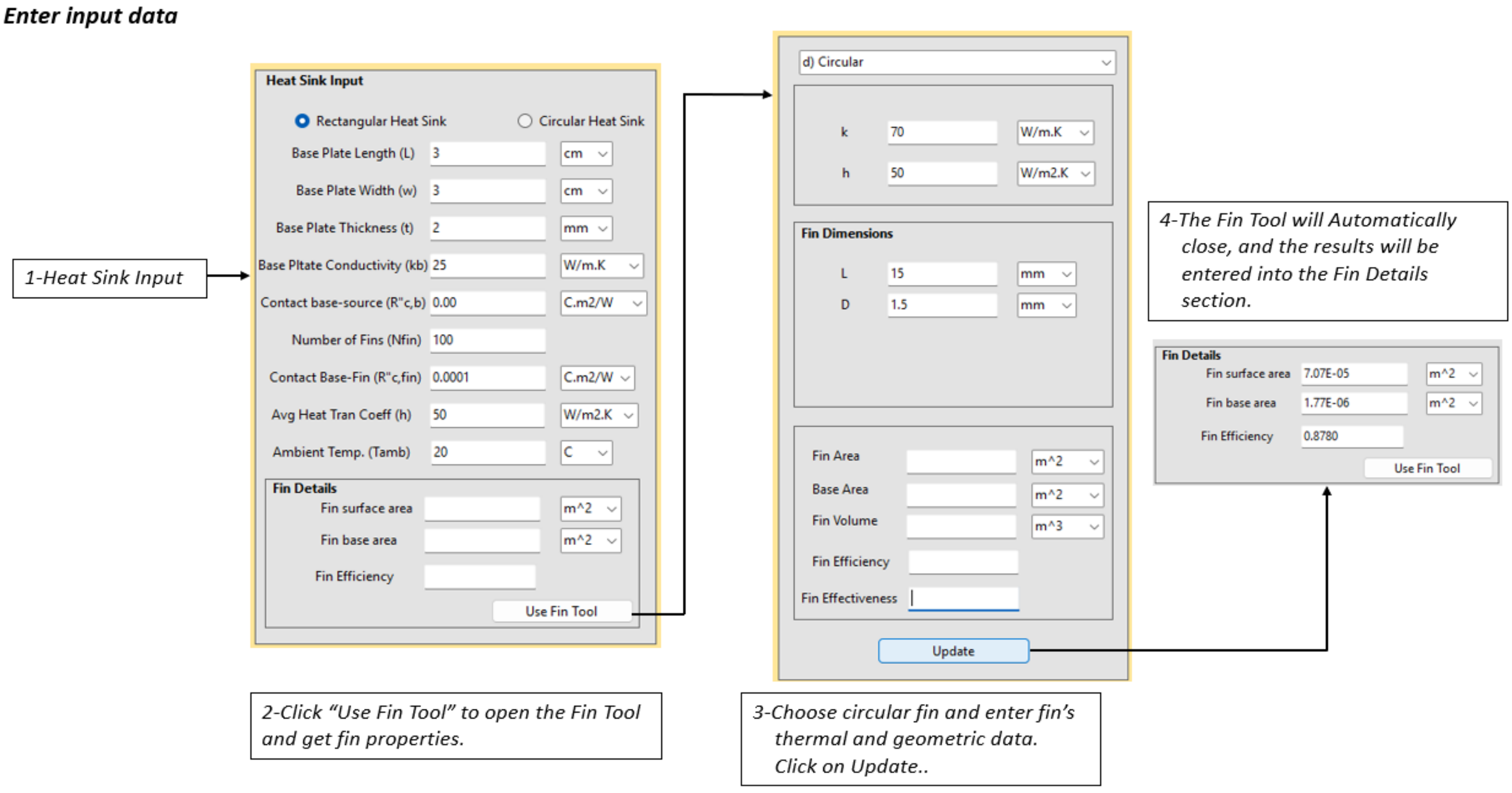
a) What is the total thermal resistance between the hot end of the thermoelectric cooler and the air?
b) What is the rate of heat rejection that can be accomplished under these conditions?

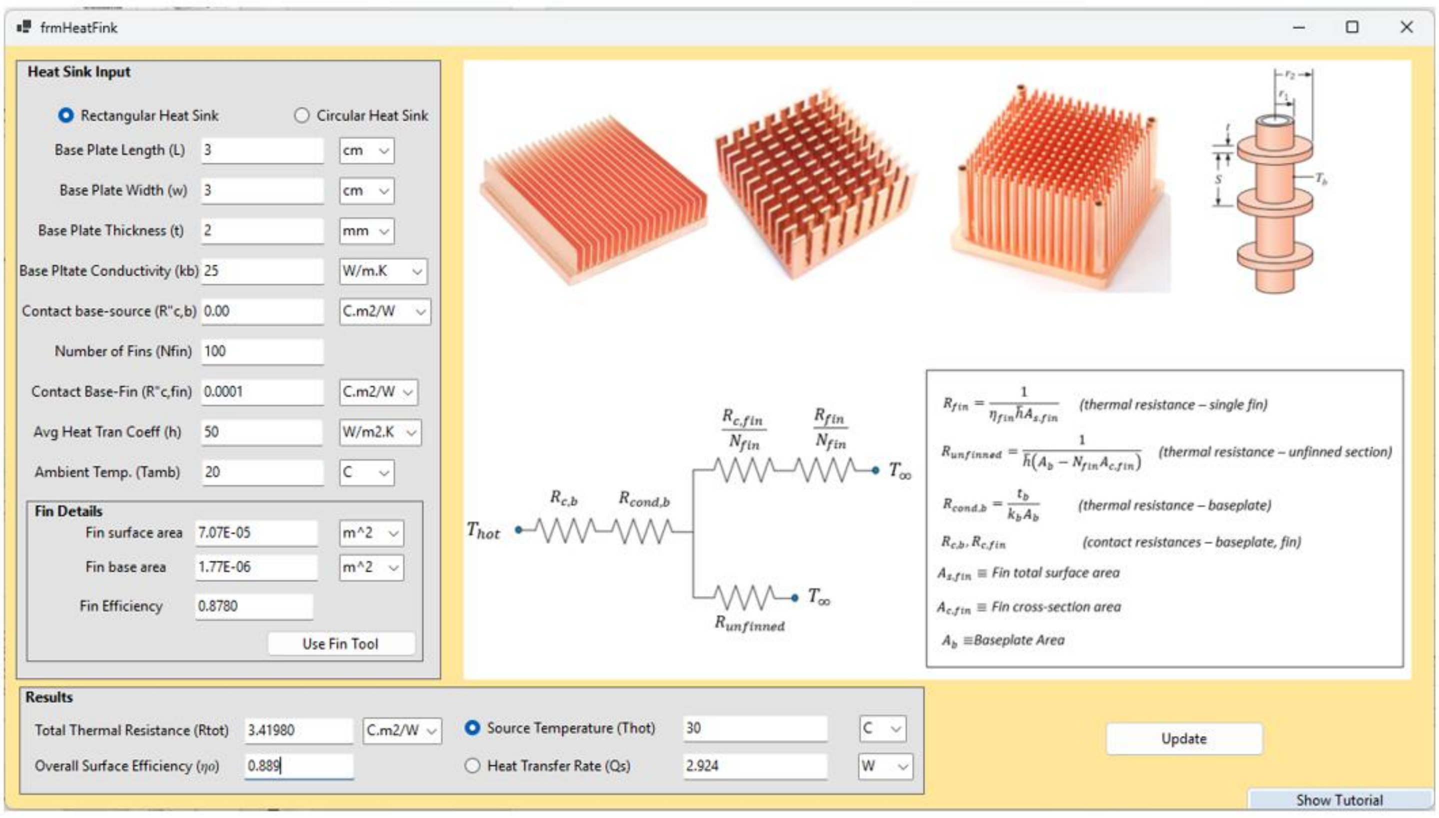
Solution:





Enter input data





5-Entrer 30 °C for Source Temperature and Click Update

6-The results are displayed as shown in completed form above.

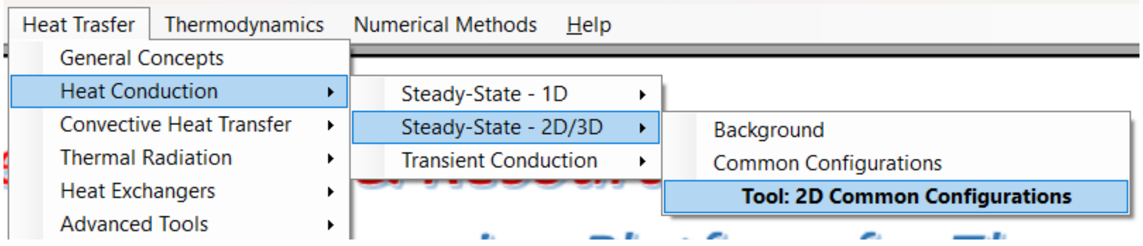
Note: The above results are identical to those in Nellis-Klein as obtained using EES program.

HT-07: Multidimensional Heat Transfer in Common Configurations

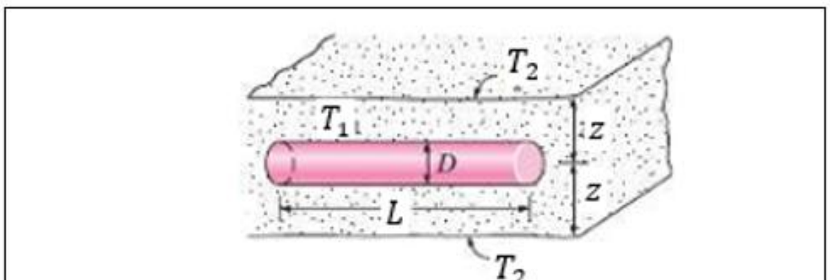
Example: Hot water at 53 °C flows through a 5-m long section of a thin-walled hot water pipe at an average velocity of 5 m/s. The pipe passes through the center of a 14-cm thick wall filled with fiberglass (k = 0.035 W/m · K) insulation. Assuming the surfaces of the wall are at 18 °C, determine a) the rate of heat transfer from the pipe to the air in the rooms and b) the temperature drop of the hot water as it flows through the pipe are to be determined.

Solution:

- Open "Common Configuration/*Shape Factor*" Panel:



- Choose configuration ***f)*** from the Shape Factors combo list



d) Parallel isothermal cylinder in mid-plane of wall

$T_1 = 53\ °C$

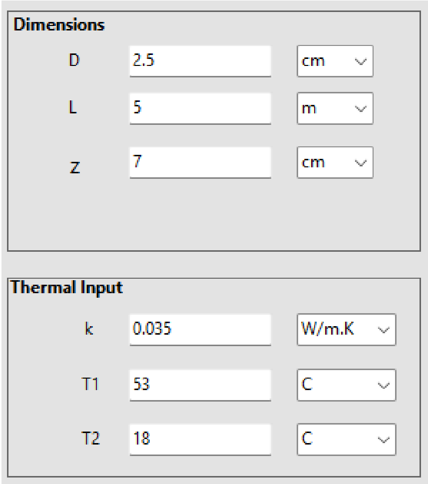
$T_2 = 18\ °C$

$D = 2.5\ cm$

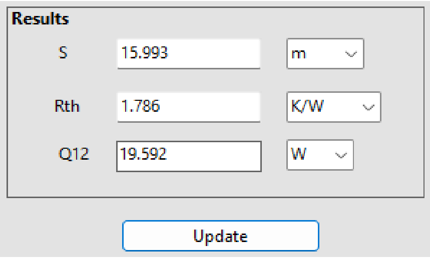
$L = 5\ m$

$z = 7\ cm$

Enter input in given units:



a) Click *Update* to the rate of heat transfer from the pipe to the air in the rooms.



The results (shown above) indicate the heat loss from the pipe is 19.6 W.

b) to calculate the temperature drop, we will perform an energy balance on the pipe:

$$\dot{Q} = \dot{m} c_p \Delta T$$

$$\Delta T = \frac{\dot{Q}}{\dot{m} c_p} = \frac{\dot{Q}}{\rho \dot{V} c_p} = \frac{\dot{Q}}{\rho V A_c c_p} = \frac{19.6\ \text{J/s}}{(1000\ \text{kg/m}^3)(0.4\ \text{m/s})\left[\dfrac{\pi(0.025\ \text{m})^2}{4}\right](4180\ \text{J/kg.°C})} = \mathbf{0.024°C}$$

<<End-of-Tutorial>>

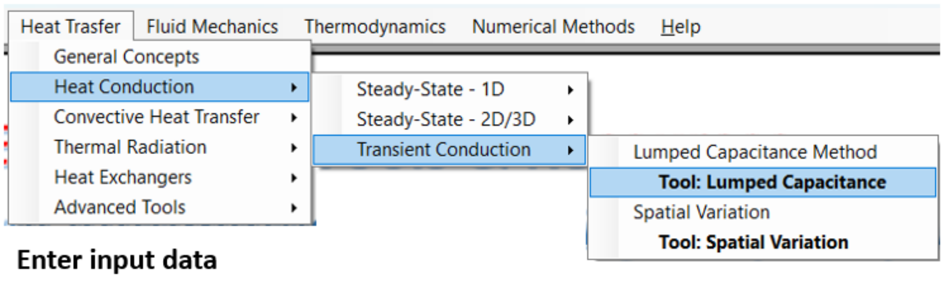
HT-08: Lumped Capacity (Constant Ambient Temperature/ Varying Ambient Temperature)

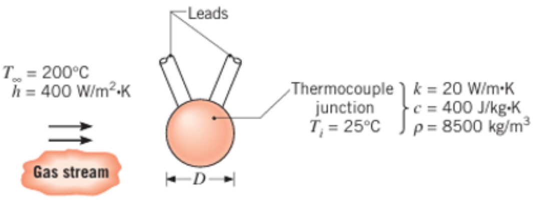
Example – Thermal Response of Thermocouple [Source: Bergman-Lavine Example 5.1]

A thermocouple junction, which may be approximated as a sphere, is to be used for temperature measurement in a gas stream. The convection coefficient between the junction surface and the gas is h = 400 W/m2.K, and the junction thermophysical properties are k = 20 W/m.K, c = 400 J/kg.K, and ρ = 8500 kg/m3. Assuming the junction diameter of the thermocouple to be 0.8 mm and initial temperature, T_i=25 °C.

a) temperature versus time for eight seconds.
b) Determine the junction temperature at time t=1 s.
c) Determine how long it will take for the junction to reach 198 °C.

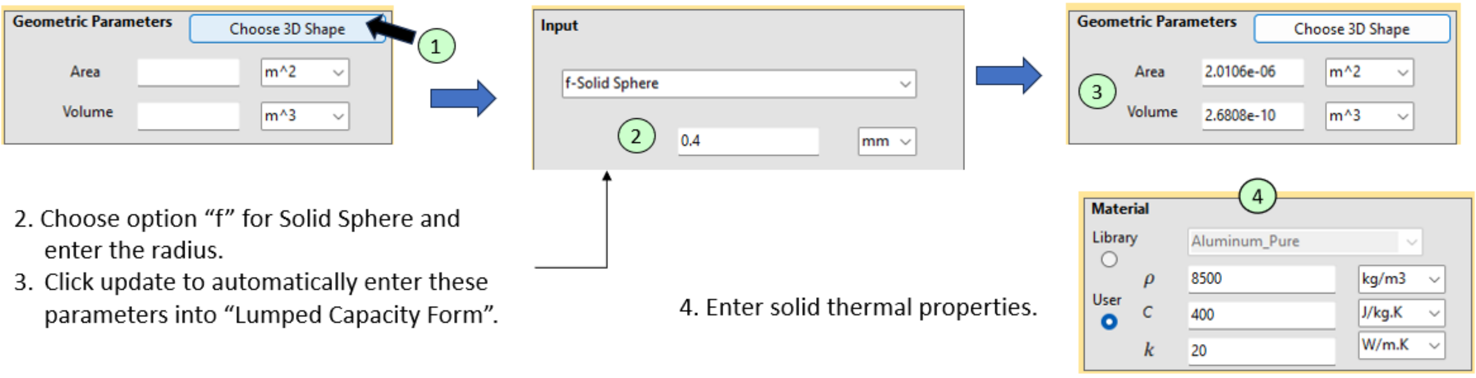
Solution:





Enter input data

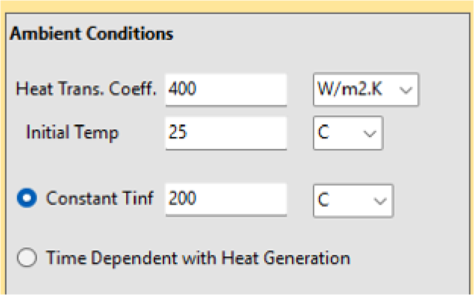
1. Click on "Choose 3D Shape" button to open the 3D shape panel area and volume calculation.



2. Choose option "f" for Solid Sphere and enter the radius.
3. Click update to automatically enter these parameters into "Lumped Capacity Form".
4. Enter solid thermal properties.

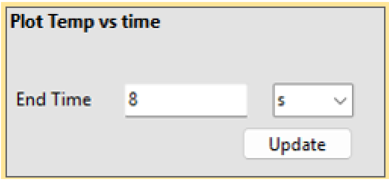
5) In “Ambient Conditions” section:

- Enter values for heat transfer coefficient and initial temperature.
- Keep Constant T_{inf} option “on” and enter the ambient temperature

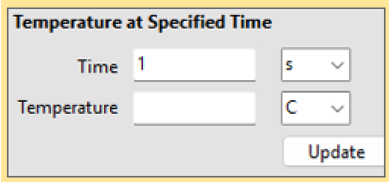


6) In “Plot Temp vs Time” section:

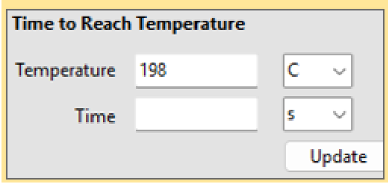
- Enter 8 second for “End Time” and click on Update.



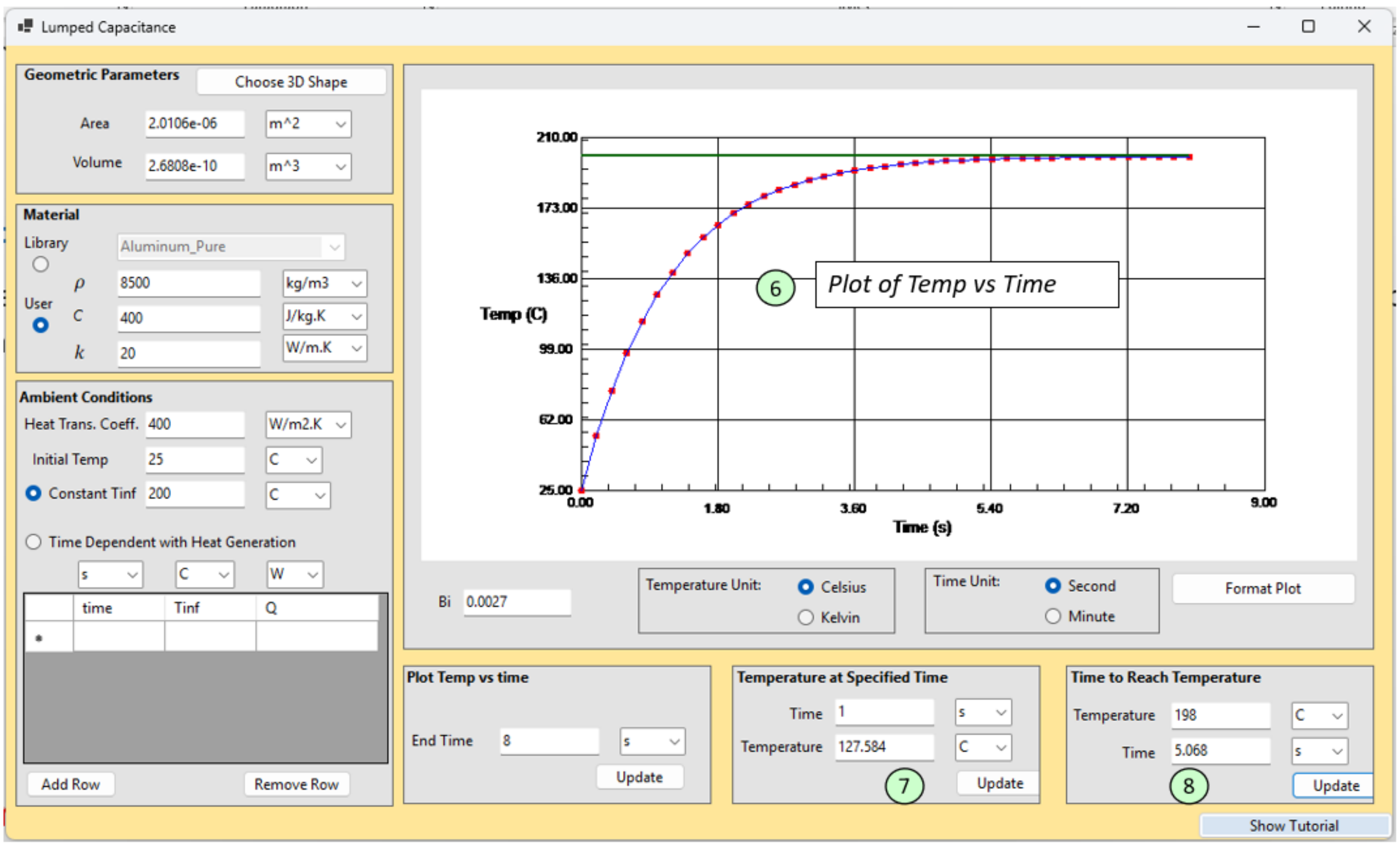
Part b):
7) To obtain ***temperature at 1 second*** enter 1 s for time and click update.



Part c):
8) To obtain ***Time to reach 198 °C*** enter 198 °C for temperature and click update.



Finished form is shown below.

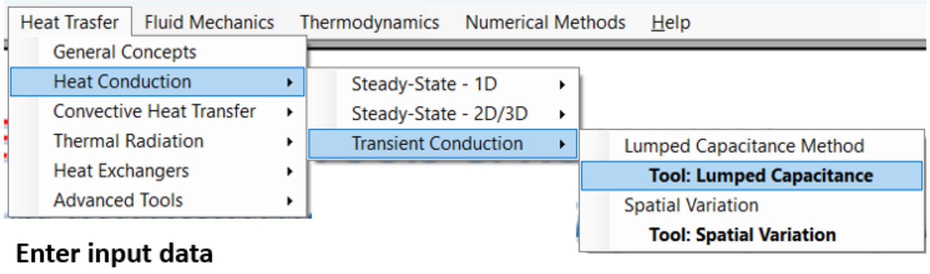


Note: the value of Bi = 0.0027 << 0.1, indicates lumped capacitance is assumption is valid.

Example – Thermal Response of Thermocouple in a Thermal Chamber

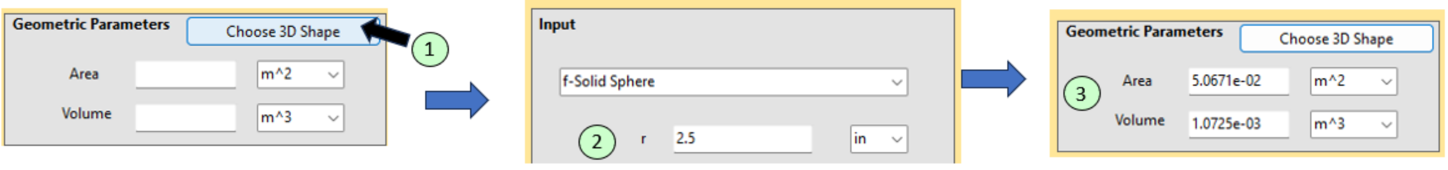
Consider a 5-inch solid metallic ball made of Aluminum_6061-T6 at initially at 85 °C in a thermal chamber with a heat transfer coefficient h=250 W/mK. Plot the sphere temperature versus time with varying chamber temperature profile shown below.

Time (min)	Temperature (°C)
0	85
10	-50
30	-50
40	85
60	85



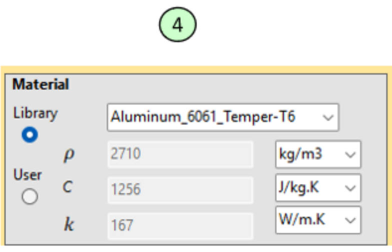
Enter input data

1. Click on "Choose 3D Shape" button to open the 3D shape panel area and volume calculation.



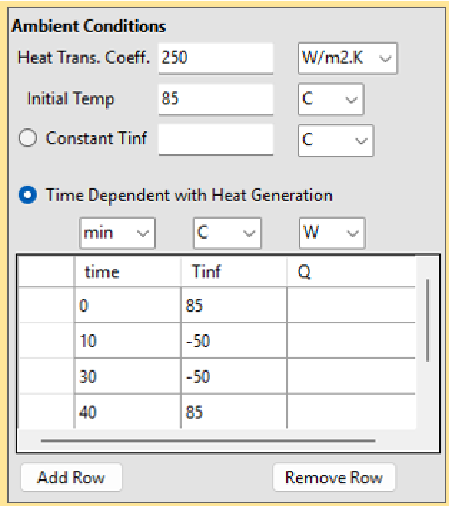
2. Choose option "f" for Solid Sphere and enter the radius.
3. Click update to automatically enter these parameters into "Lumped Capacity Form".

4. Enter solid thermal properties. Choose Al_6061-T6 from list of materials in the library.



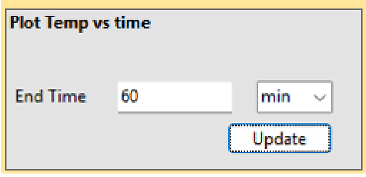
5) In "Ambient Conditions" section:

- Enter values for heat transfer coefficient and initial temperature.
- Change T_{inf} option to "Time Dependent" and enter values for time and chamber temperature from profile table above. Make sure to change the time unit to minutes.

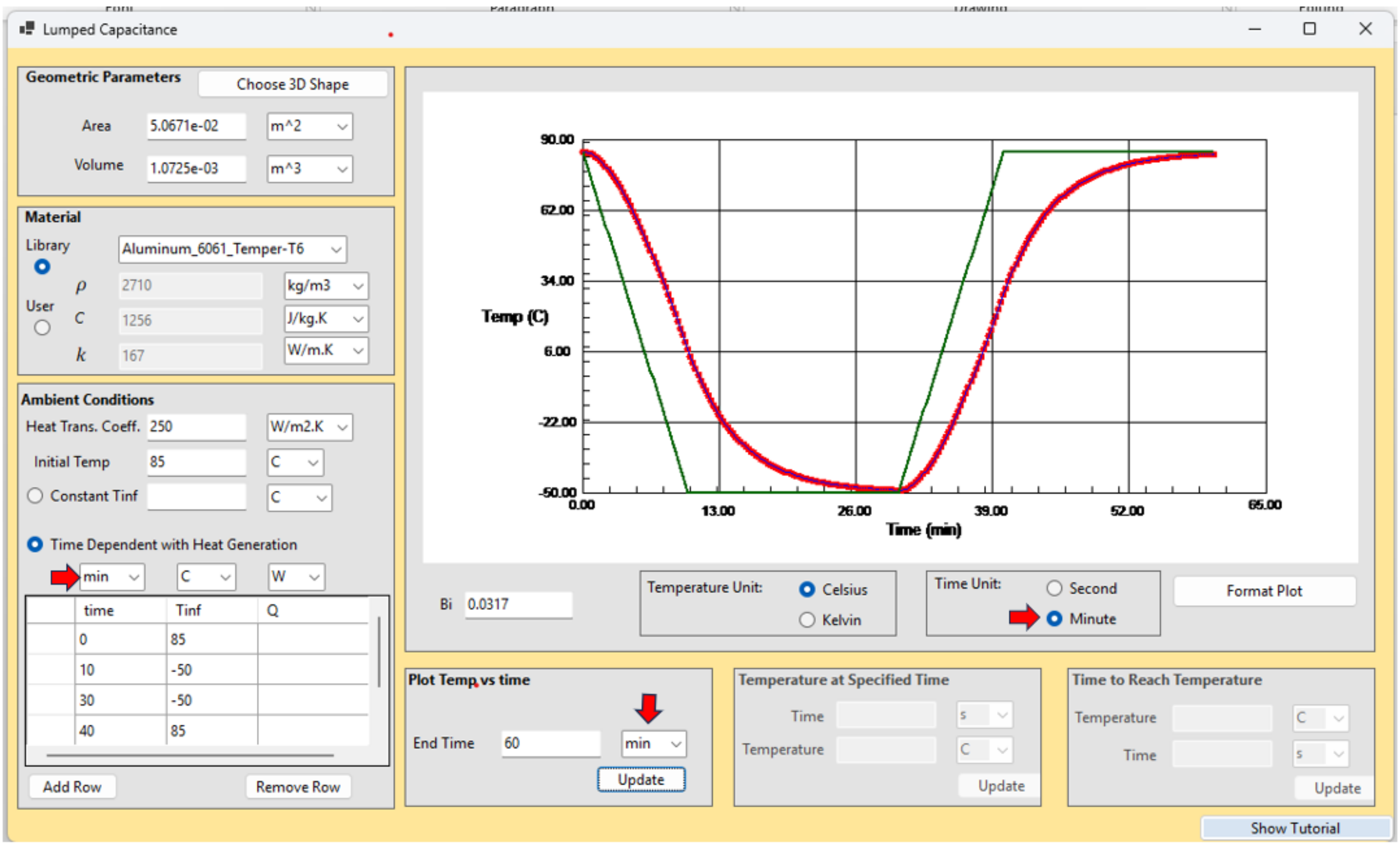


6) In "Plot Temp vs Time" section:

- Enter 60 minutes for "End Time"
- Change Time unit in the plot area to minutes
- click on Update.



Finished form is shown below.



Note: the value of Bi = 0.037 << 0.1, indicates lumped capacitance is assumption is valid.

HT-09: Transient Conduction (Walls, Cylinders, Spheres)

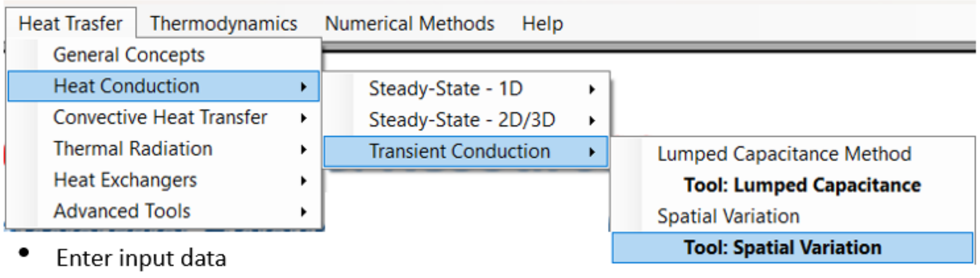
Example – Plate: Heating of a Steel Pipeline [Source: Bergman-Lavine Example 7.7]

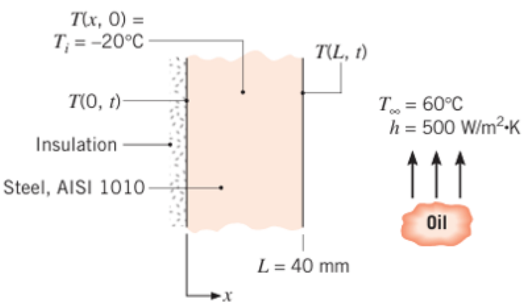
Consider a 1-m diameter steel pipeline (AISI 1010) with a wall thickness of 40 mm. The pipe is heavily insulated on the outside, and is initially at a uniform temperature of −20°C. With the initiation of flow, hot oil at 60°C is pumped through the pipe, creating a convective condition corresponding to h = 500 W/m2 · K at the inner surface of the pipe.

a. Determine the temperature of the exterior pipe surface covered by the insulation at t = 8 min?
b. How much energy per meter of pipe length has been transferred from the oil to the pipe at t = 8 min?

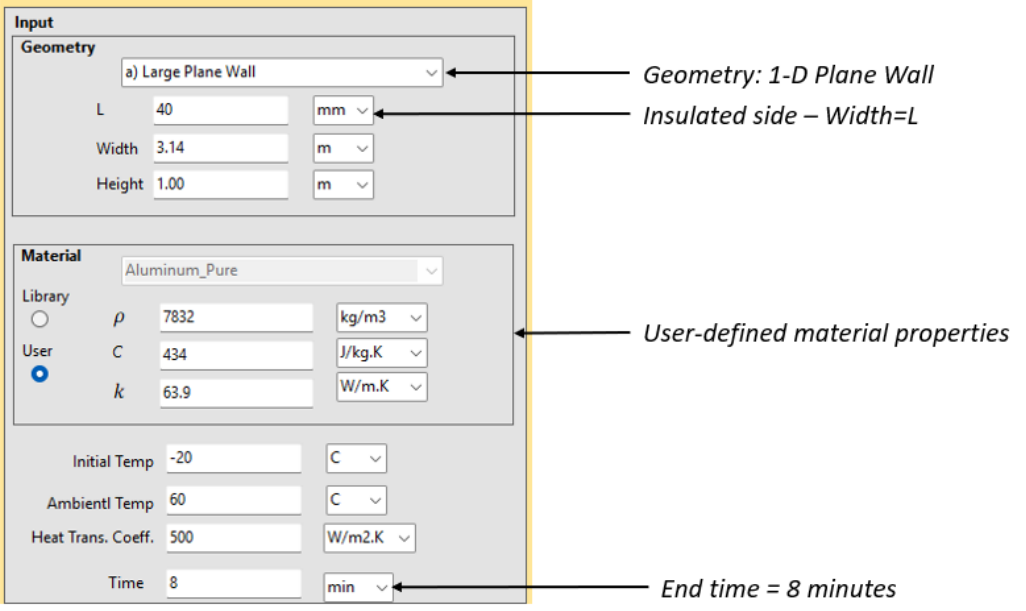
Solution:

- Open "*Transient Conduction*" Tool

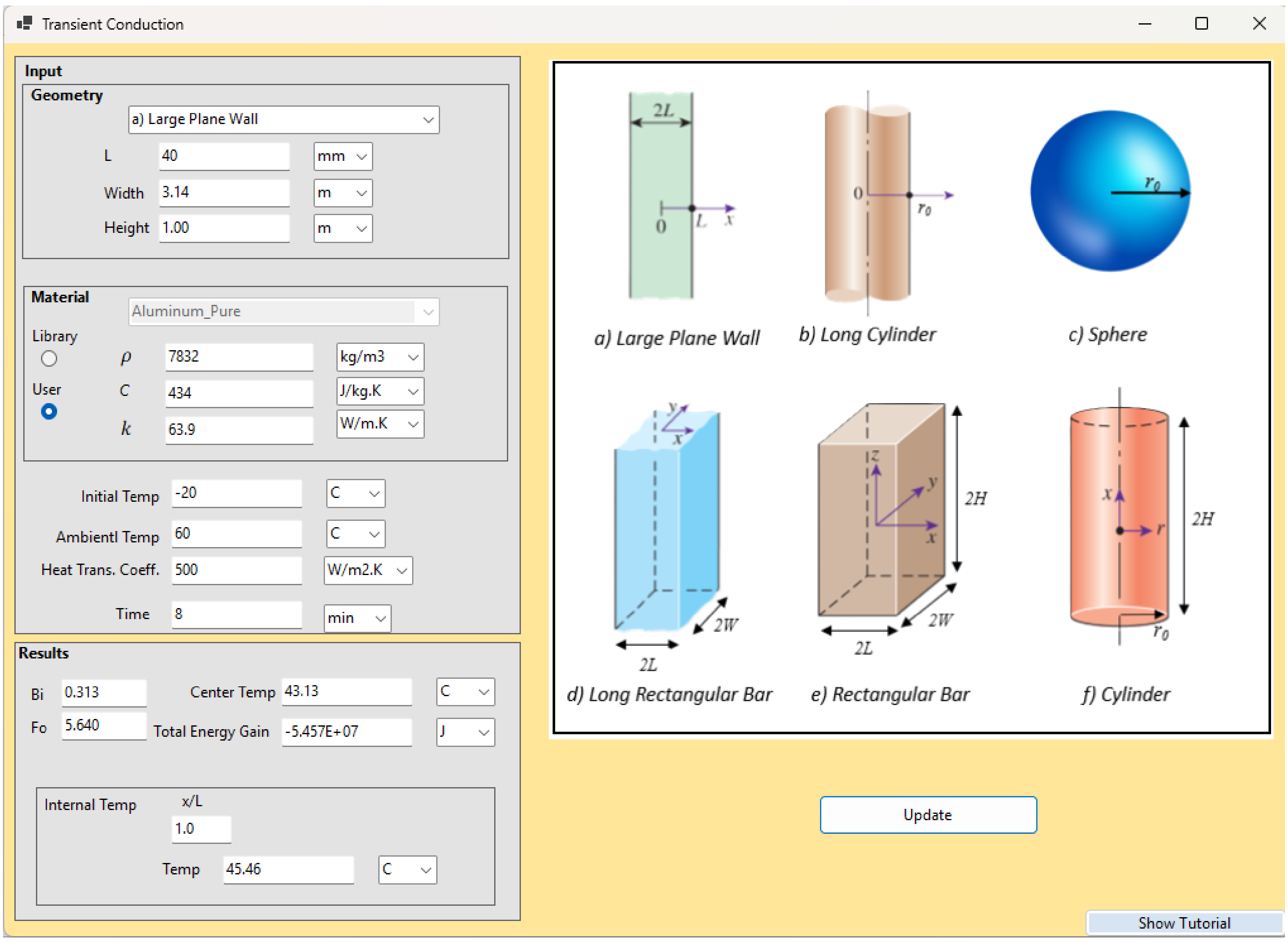




- Enter input data



- Click Update and Solve



Comments:

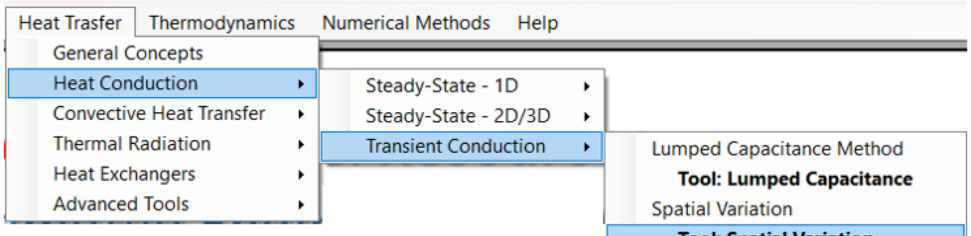
1. The Biot number, *Bi*=0.313 and the Fourier number Fo=5.64
2. The "*Center Temperature*" refers to the temperature at the center of a wall with thickness 2L. Here, it indicates the temperature at the insulated side of the wall.
3. The "Total Energy Gain" of -5.457E+07 J is the energy gained by a wall of thickness 2L during the period of 8 minutes. Here, since we the wall has an insulated side (L), the total energy gain will be half of this value; Q_{tot}=-2.73E+07 J per meter of the pipe. The negative sign implies heat transfer into the pipe from the oil
4. Temperature at x/L=1 (45.46 °C) represents the temperature on the surface that is exposed to the oil.

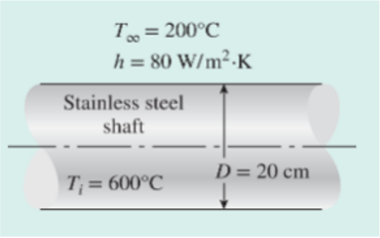
Example – Long Cylinder: Cooling of a Long Stainless-Steel Cylindrical Shaft [Source: Cengel-Ghagar Example 4.4]

A long 20-cm diameter cylindrical shaft made of stainless-steel 304 is out of an oven at a uniform temperature of 600 °C. The shaft is then allowed to cool in an environmental chamber at 200 °C and a heat transfer coefficient of 80 $W/m^2.K$. Determine a) the temperature at the center of the shaft after 45 minutes and b) total amount of heat transferred from the shaft to the air during the cooling process per unit length.

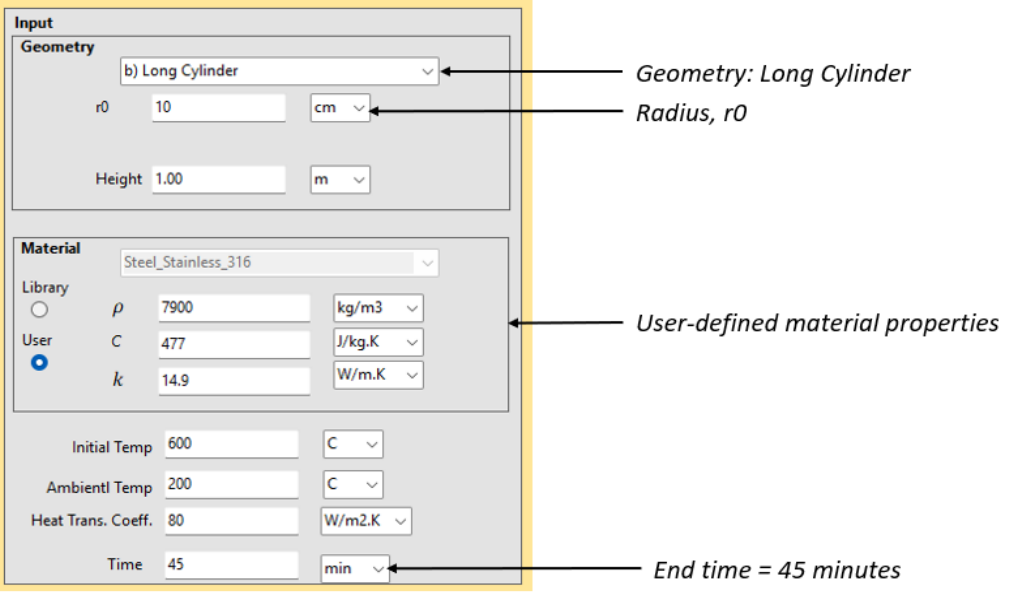
Solution:

- Open "*Transient Conduction*" Tool





- Enter input data



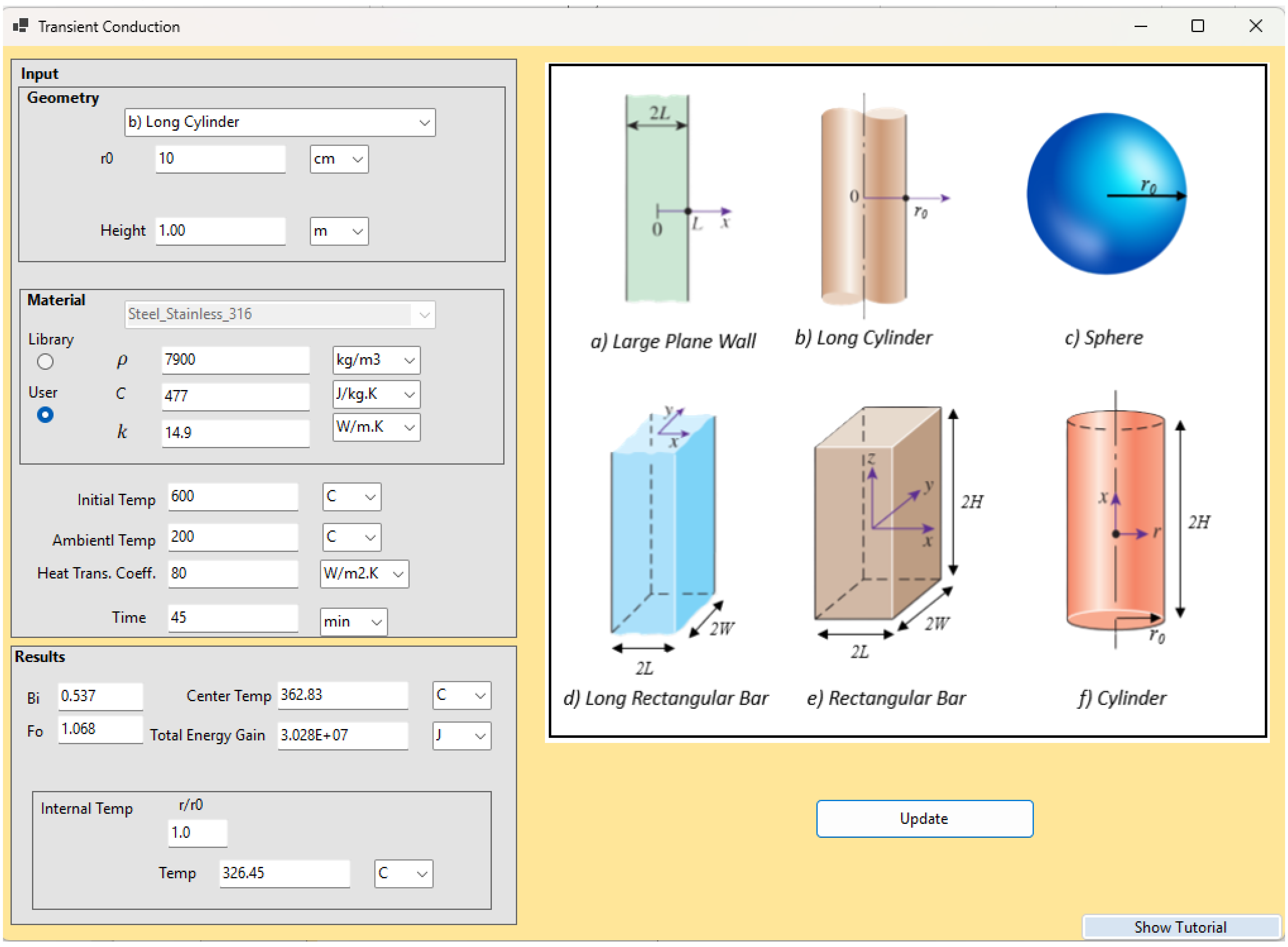
Properties of Stainless Steel as given:

$$\rho = 7900\ \frac{kg}{m^3}$$

$$c_p = 477\ \frac{J}{kg.k}$$

$$k = 14.9\ \frac{W}{m.k}$$

- Click Update and Solve



Comments:

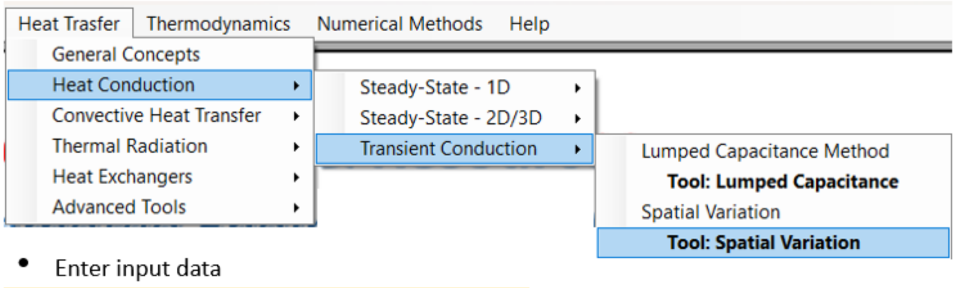
1. The Biot number, *Bi*=0.537and the Fourier number Fo=1.068
2. The "*Center Temperature*" refers to the temperature at the center of the cylinder is 362.83 °C.
3. The "Total Energy Gain"; Q_{tot}=3.028E+07 J per meter of the pipe.
4. The surface temperature (r/r0=1) is calculated to be 326.45.

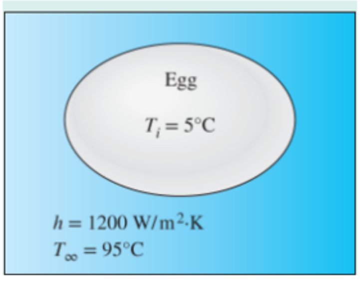
Example – Sphere: Boiling of Egg

An egg may be modeled as a spherical object of 5-cm diameter (*ρ=1000 kg/m3, cp=4150 J/kg.°C, k=0.627 W/m.K*). Determine the temperature at the center of an egg after 14 minutes in boiling water at 95 °C. Assume the egg is initially at 5 °C and the convection heat transfer coefficient of the boiling water to be 1200 W/m^2.K.

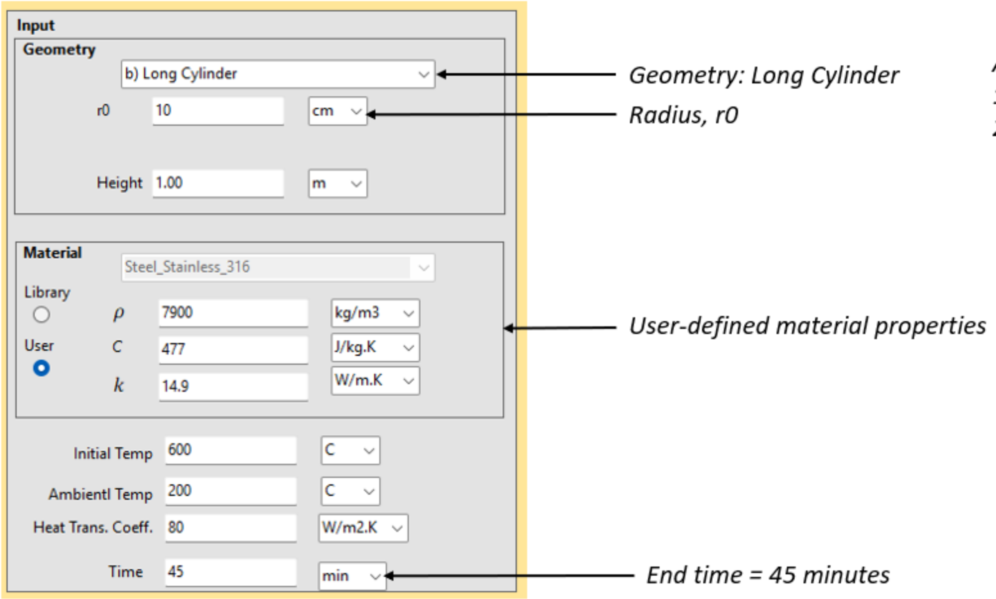
Solution:

- Open "*Transient Conduction*" Tool





- Enter input data



Assumptions:

1. *Egg modeled as a sphere with D=2.5cm*
2. *Egg is mostly water, so its properties are assumed as water the mean temperature.*

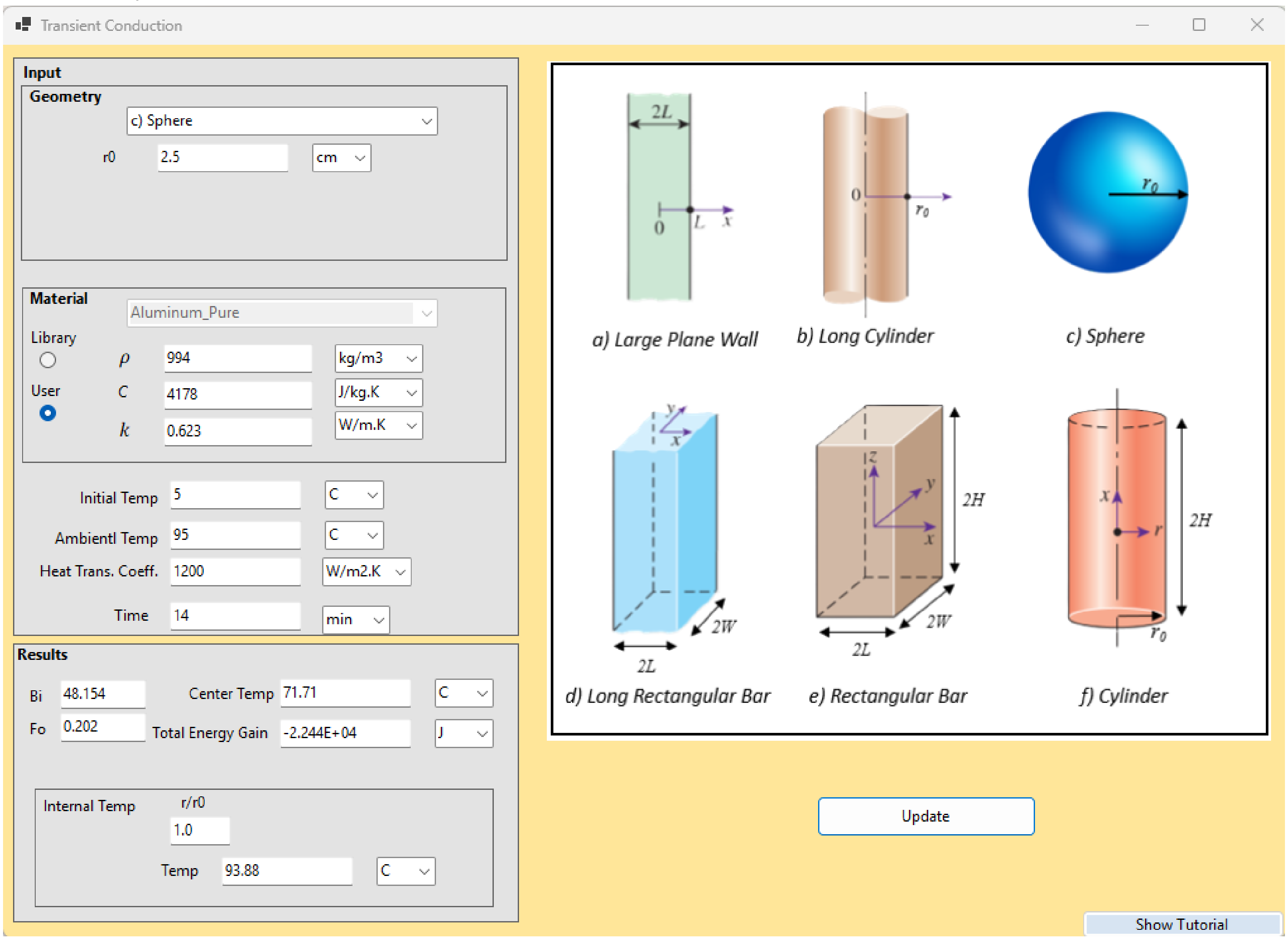
Thermal properties of egg:

$$\rho = 994\ \frac{kg}{m^3}$$

$$c_p = 4178\ \frac{J}{kg.k}$$

$$k = 0.623\ \frac{W}{m.k}$$

- Click Update and Solve



Comments:

1. The Biot number, *Bi*=48.15 and the Fourier number Fo=0.202
2. The "*Center Temperature*" is 71.7 °C.
3. The surface temperature of the egg is calculated to be 93.9 °C.
4. The "Total Energy Gain"; Q_{tot}=2.244E+04 J into the egg.
5. The surface temperature (r/r0=1) is calculated to be 93.88.

Example – Short Cylinder: Cooling of a Brass Bar

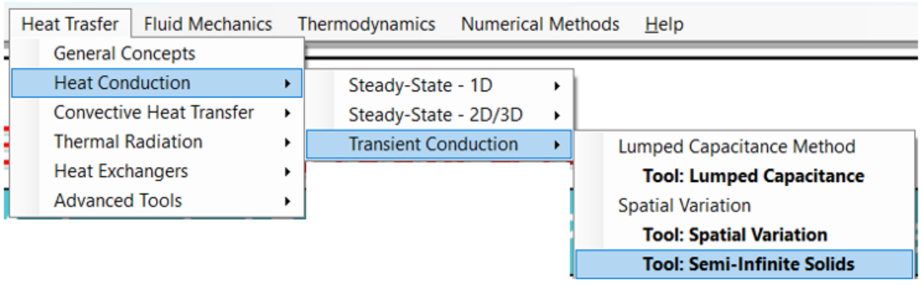
A short 10-cm diameter brass cylinder with the height H = 12 cm is initially at a uniform temperature T_i = 120C. The cylinder is now placed in atmospheric air at 25°C, where heat transfer takes place by Convection, with a heat transfer coefficient of h = 60 W/m2.K. Calculate the temperature at (a) the center of the cylinder and (b) the center of the top surface of the cylinder 15 min after the start of the cooling.

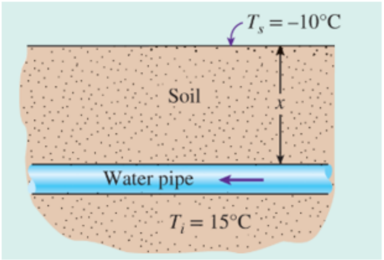
Solution:

HT-10: Semi-Infinite Solids

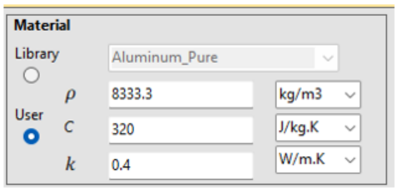
Example – Burial Depth of Pipes to Avoid Freezing

The ground at a particular location is covered with snowpack at -10°C for a continuous period of three months ($90 \times 24 = 2160\ hours$), and the average soil properties at that location are k = 0.4 W/m.K, ρ=8333.3 kg/ m^3 ,and cp =320 J/kg.K. Assuming an initial uniform temperature of 15 °C for the ground, determine a water pipe will freeze at the burial depth of 80 cm during the 3 month period. Repeat the simulation using the depth of 500 cm and notice.

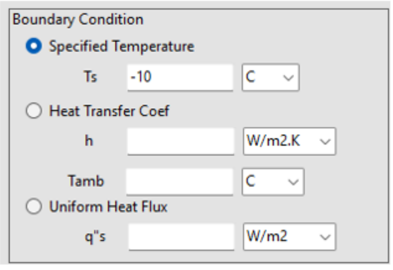




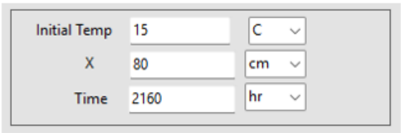
Enter input data



1. Select "User" material type.
2. Enter density, specific heat and thermal conductivity.

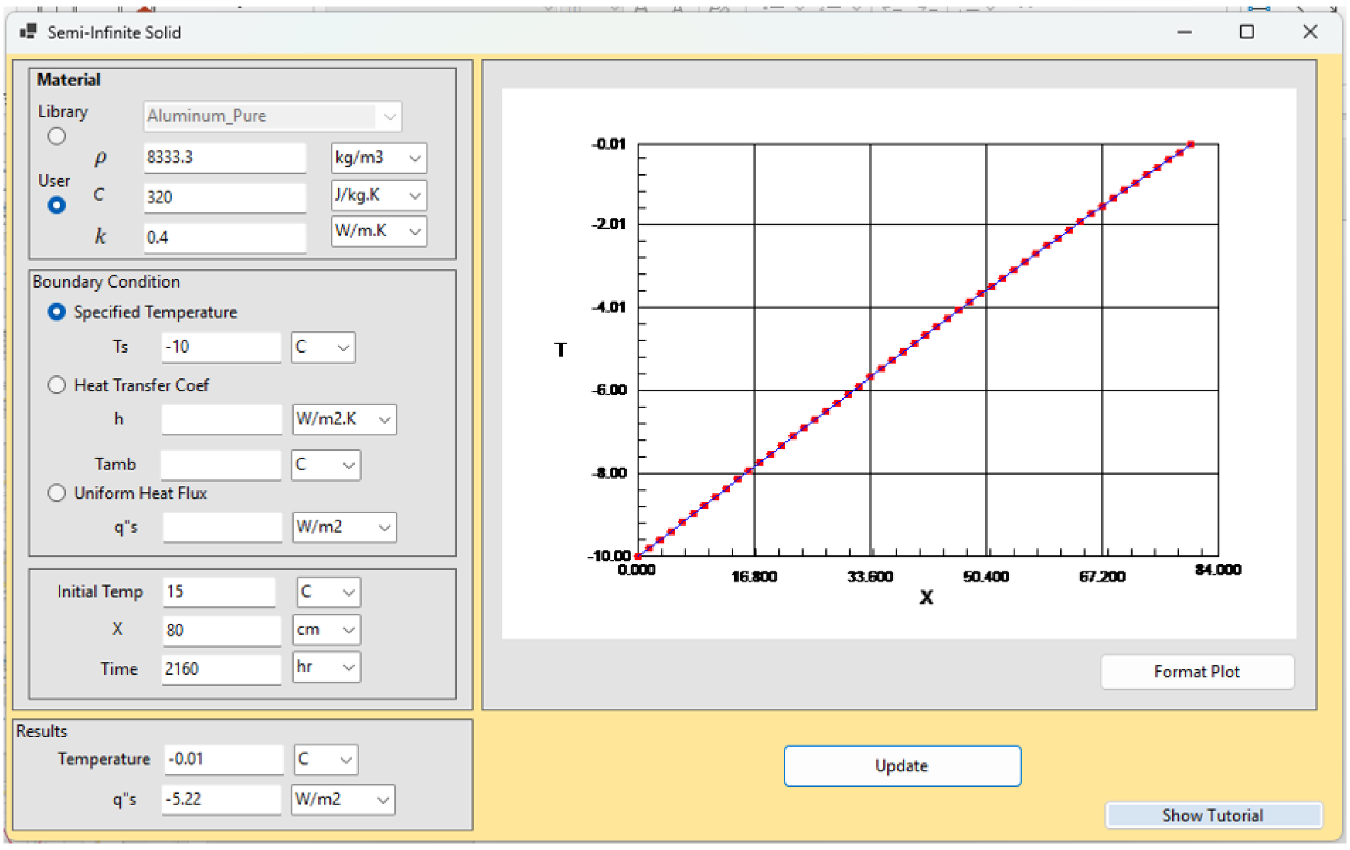


3. Choose "Specified Temperature" BC and enter the given value.

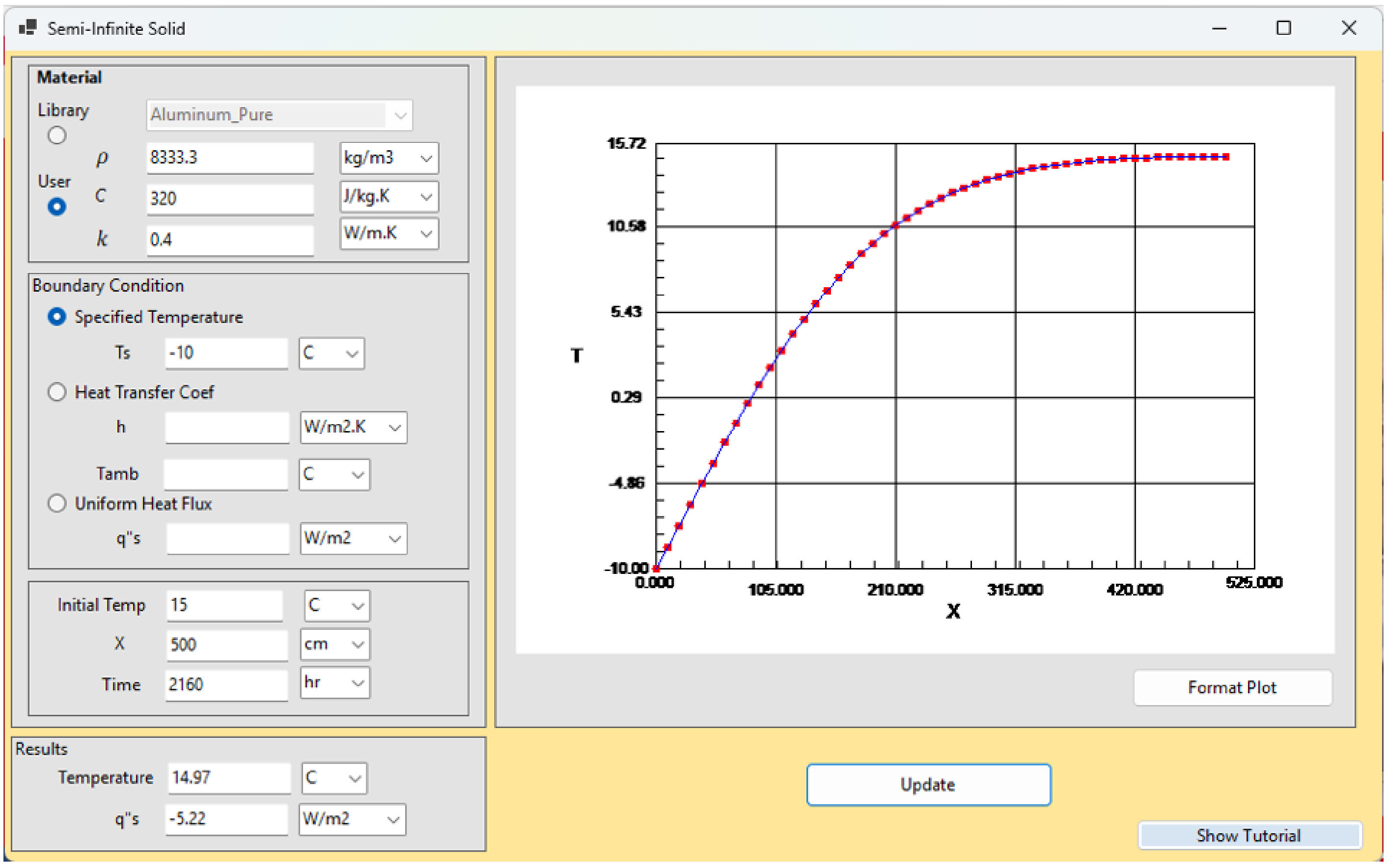


4. Enter values for Initial Temperature, Depth into the soil, and time.

Finished form is shown below.



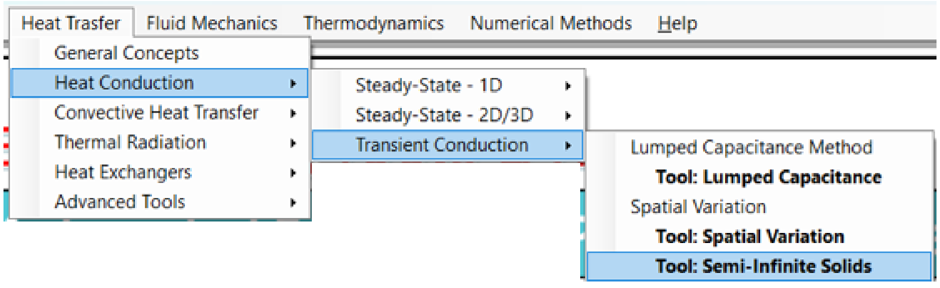
Note: 1. The value of 80 cm is the minimum burial depth of pipes, since the pipe temperature reaches 0 °C after 3 months.
2. As evident from the plot, the at this depth the variation is nearly linear. If you rerun with x=500 cm, we can see the decay.

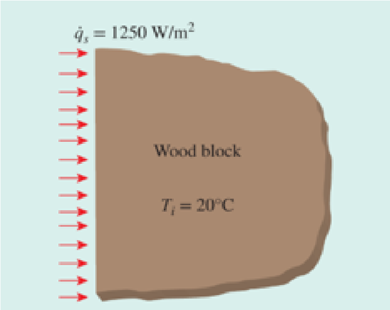


Example – Surface temperature rise of Heated Block

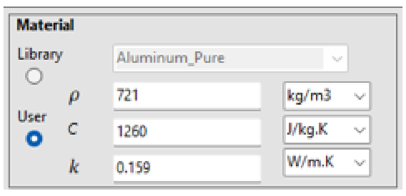
A thick, black-painted wood block ($k = 0.159\,\frac{W}{m.K}$, $\rho = 721\,\frac{m^3}{kg}$, $c_p = 1260\ \frac{J}{kg.K}$) at 20 °C is subjected to constant solar heat flux of 1250 W/m2.

1) Determine the exposed surface temperature of the block after 20 minutes. Plot results to 20 cm into the block.
2) Repeat for block made of pure aluminum, and Plot results to 100 cm into the block.

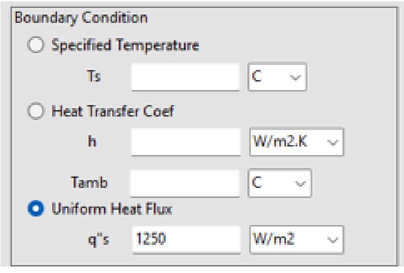




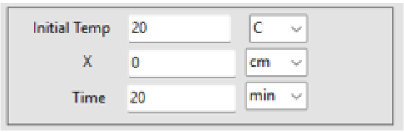
Enter input data



1. Select "User" material type.
2. Enter density, specific heat and thermal conductivity.

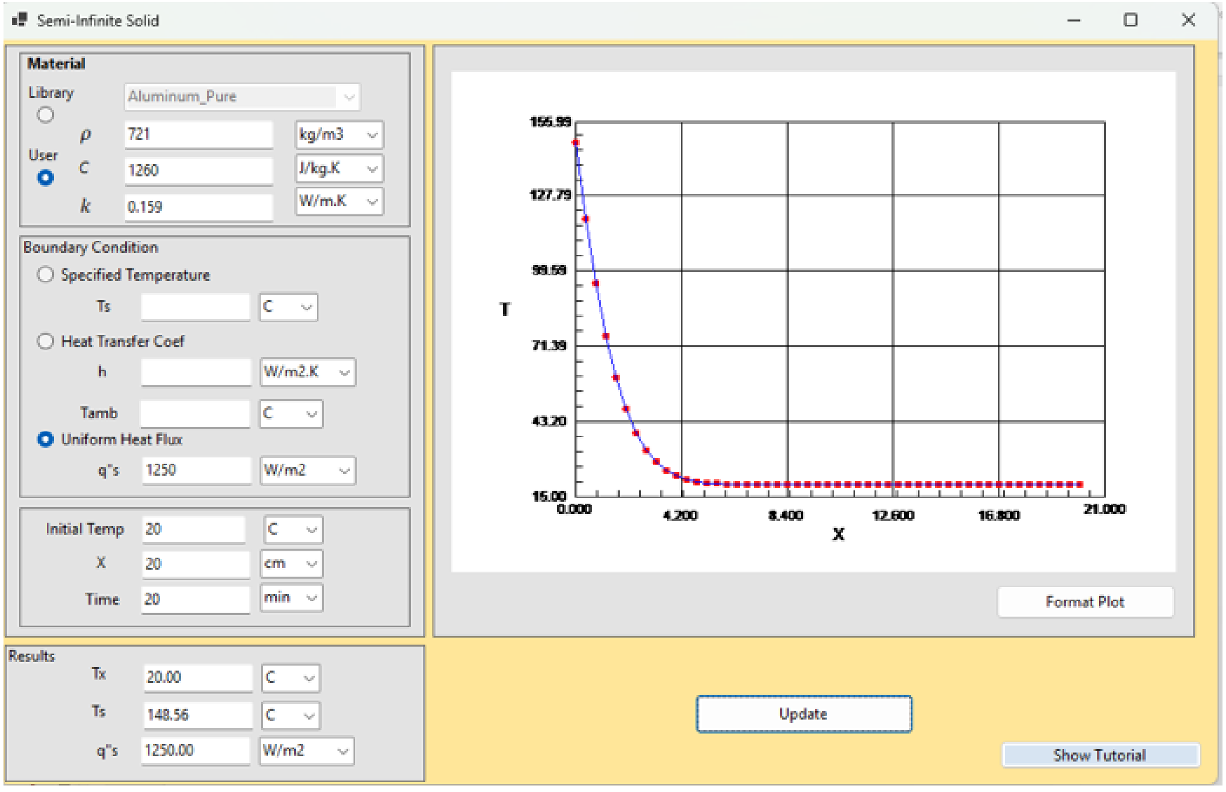


3. Choose "Uniform Heat Flux" BC and enter the given value.

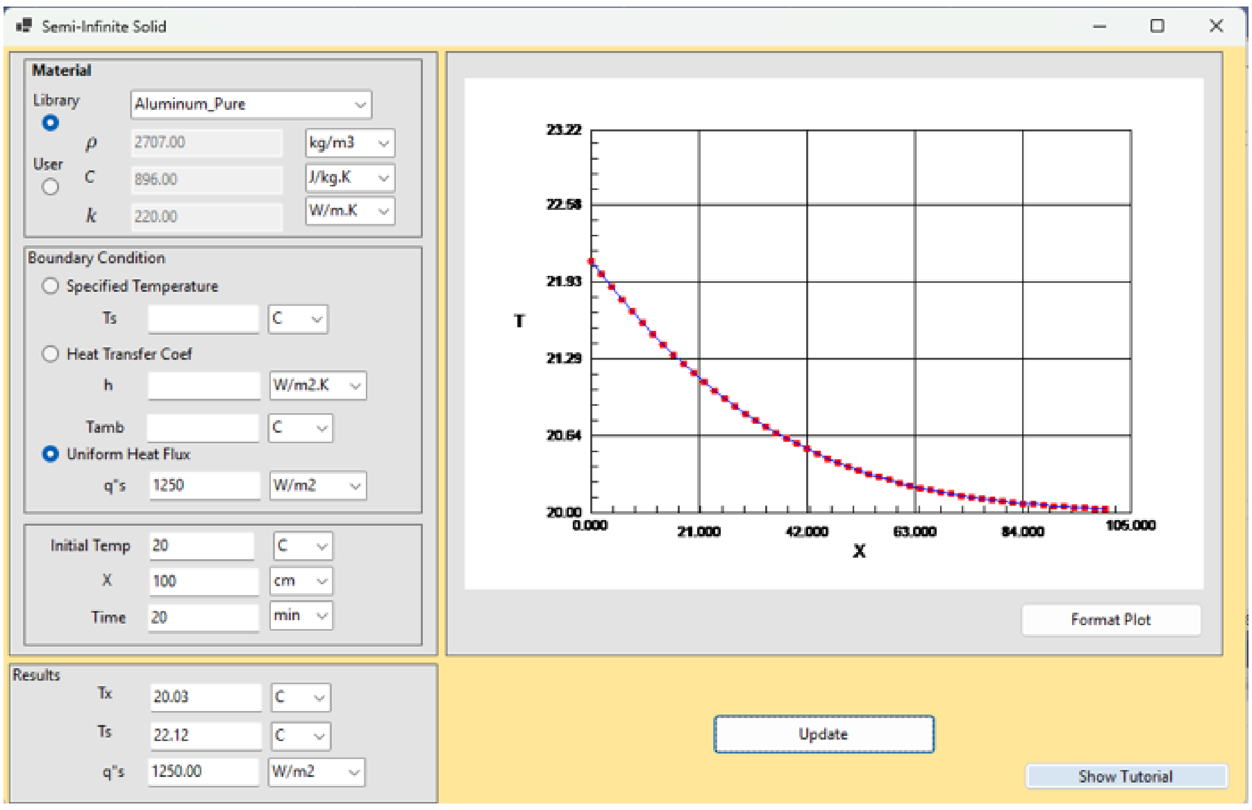


4. Enter values for Initial Temperature, Depth into the soil (0 for surface), and time.

Finished form is shown below.



Wood Block: 1. The surface temperature is 148.6 °C.
2. The plot shows thermal penetration within the wooden block to the depth of 4.2 cm.



Aluminum: 1. The surface temperature is 22.12 °C.
2. The plot shows thermal penetration within the wooden block to the depth of 100 cm.

Example – Compliance of ASME Codes for Bolts Exposed to Cryogenic Fluid

A series of long stainless-steel bolts (ASTM A437 B4B) are fastened into a thick metal plate. The metal plate has a thermal conductivity of 16.3 W/m.K, a specific heat of 500 J/kg .K, and a density of 8 g/cm3. The upper surface of the plate is occasionally exposed to cryogenic fluid at -70°C with a convection heat transfer coefficient of 300 W/m2.K. The bolts are fastened into the metal plate from the bottom surface, and the distance measured from the plate's upper surface to the bolt tips is L = 1 cm. The ASME Code for Process Piping limits the minimum suitable temperature for ASTM A437 B4B stainless steel bolt to -30 °C. If the initial temperature of the plate is 10°C and the plate's upper surface is exposed to the cryogenic fluid for 30 minutes, would the bolts fastened in the plate still comply with the ASME code?

HT-11: Contact of Two Semi-Infinite Solids

HT-12: Flat Plate in Parallel Flow

Example – Flow of Hot Oil Over a Flat Plate

Engine oil at 60°C flows over the upper surface of a 5-m-long flat plate whose temperature is 20°C with a velocity of 2 m/s. Determine the total drag force and the rate of heat transfer per unit width of the entire plate.

HT-13: Flow Over 3D Bodies

Example – Heating of Horizontal Cylinder in Cross Flow

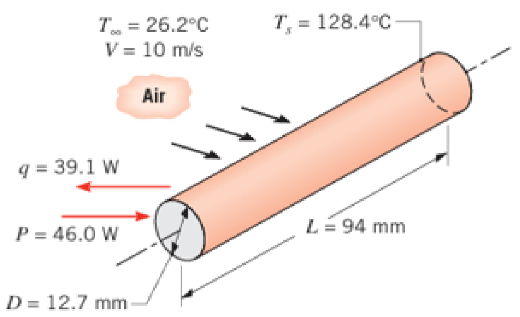
Experiments have been conducted using a metallic cylinder 12.7 mm in diameter and 94 mm long. The cylinder is heated internally by an electrical heater and is subjected to a cross flow of air in a low-speed wind tunnel. Under a specific set of operating conditions for which the upstream air velocity and temperature were maintained at V = 10 m/s and 26.2°C, respectively, the heater power dissipation was measured to be P = 46 W, while the average cylinder surface temperature was determined to be Ts = 128.4°C. Assuming that 15% of the power dissipation is lost through the cumulative effects of surface radiation and conduction through the endpieces:

a) Determine the convection heat transfer coefficient from the experimental observations.
b) Compare the experimental result with the convection coefficient computed from the correlation.

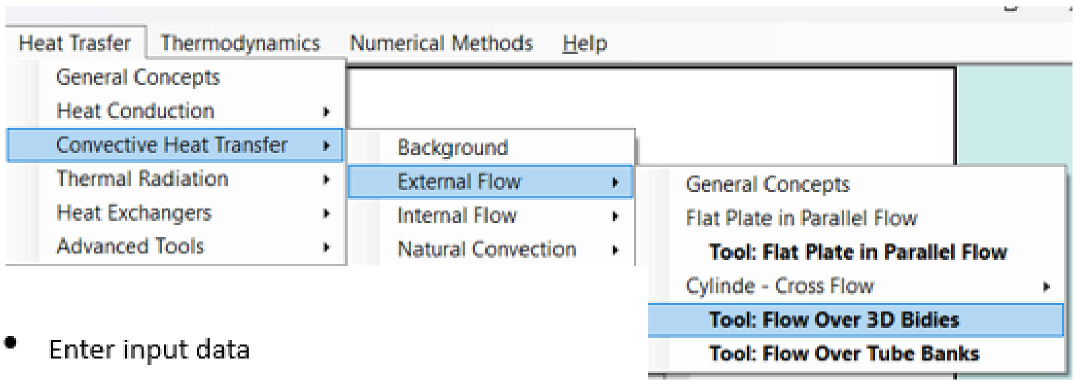
a) The convection coefficient can be evaluated from the experimental data, assuming 85% of the total heat dissipation dissipated from the surface of the rod:

$$\bar{h} = \frac{q}{A(T_s - T_\infty)}$$

$$\bar{h} = \frac{0.85 \times 46}{\pi(0.0127)(128.4 - 26.2)} = 102\,\frac{W}{m^2.K}$$



b) To obtain the heat transfer coefficient using the correlation, open "*Flow Over 3D Bodies Tool*"



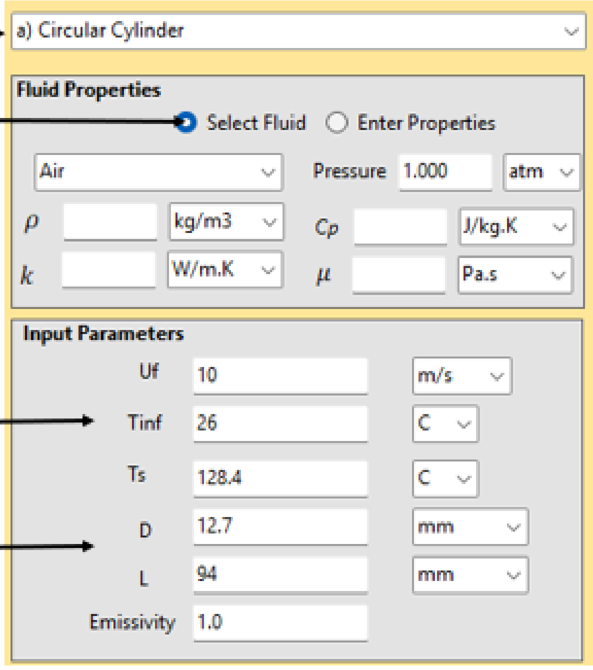
- Enter input data

Geometry: Circular Cylinder

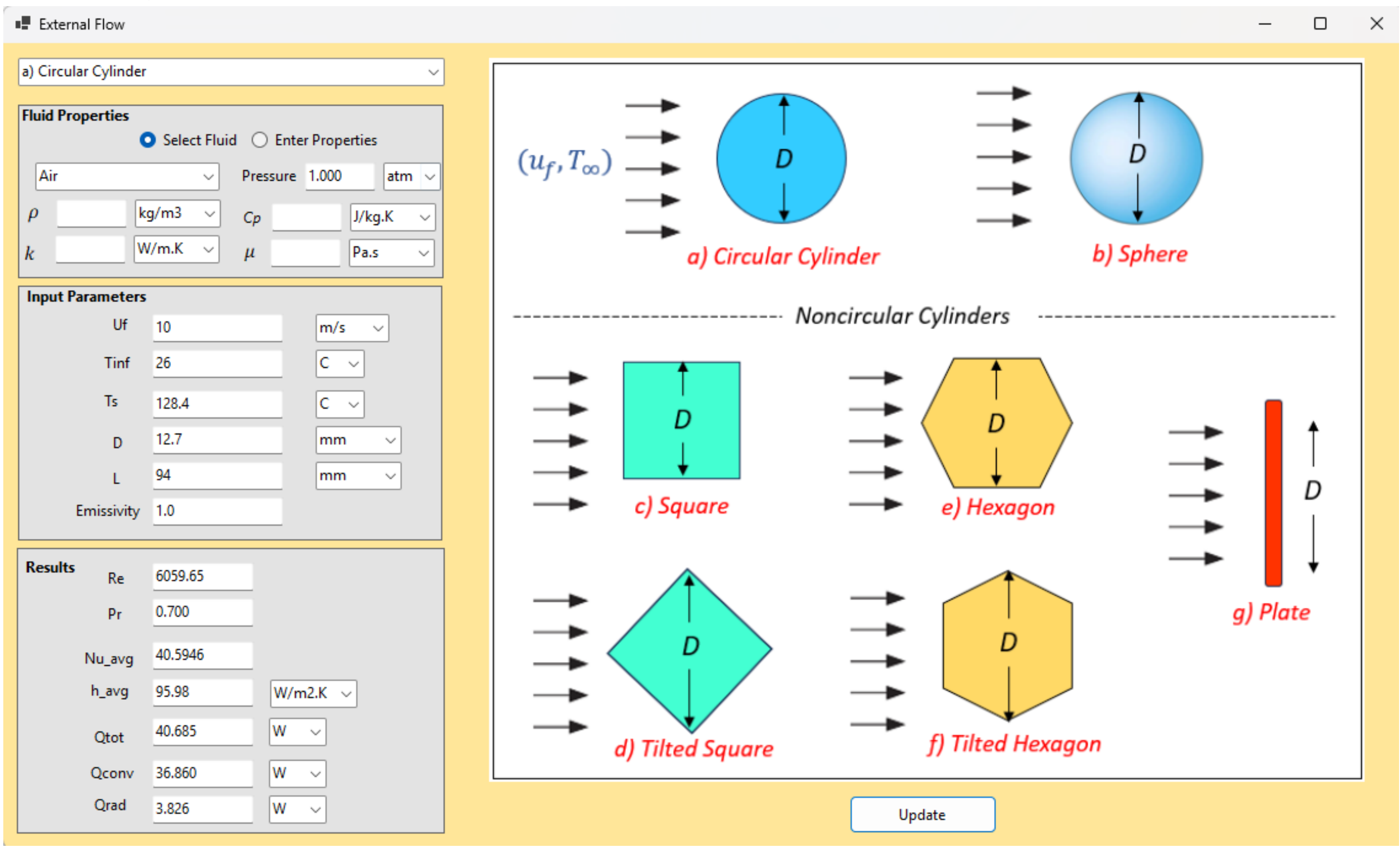
Select Air from materials library

Enter Flow and Thermal Information

Enter Geometric Information



- Click Update and Solve



Comments:

1. The Reynolds number, Re_D=6059.
2. The Nusselt number, *Nu*=40.6.
3. The average heat transfer coefficient is evaluated to be 95.6 W/m^2.K. This is very close (about 6% difference) to the experimentally determined value of 102 W/m^2.K.
4. The theoretically calculated rate of heat transfer from the rod is 40.69 W, which is very close to the experimentally measured value of 39.1 W.

Solution:

HT-14: Flow Over Tube Banks

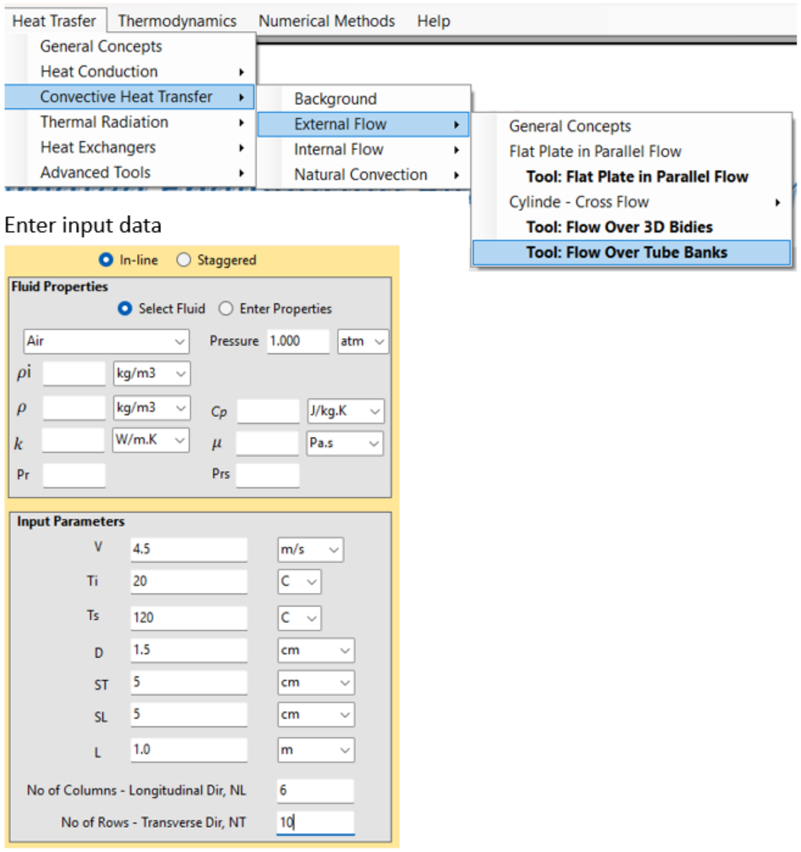
The following three examples illustrate how to solve heat transfer problems involving flow across tube b both in-line and staggered configurations.

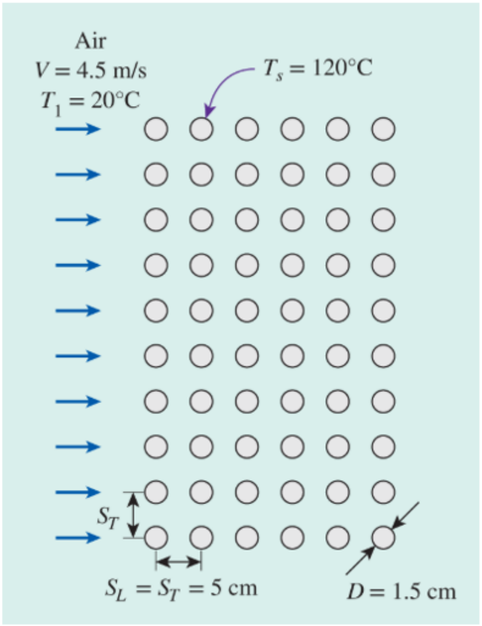
Example – Preheating Water in a Bank Tube [Source: Cengel-Ghagar Example 7.8]

In an industrial facility, air is to be preheated before entering a furnace by geothermal water at 120 °C flowing through the tubes of a tube bank located in a duct. Air enters the duct at 20 °C and 1 atm with a mean velocity of 4.5 m/s and flows over the tubes in normal direction. The outer diameter of the tubes is 1.5 cm, and the tubes are arranged in-line with longitudinal and transverse pitches of S_L= S_T = 5 cm. There are 6 rows in the flow direction with 10 tubes in each row. Determine the rate of heat transfer per unit length of the tubes and the pressure drop across the tube bank.

Solution:

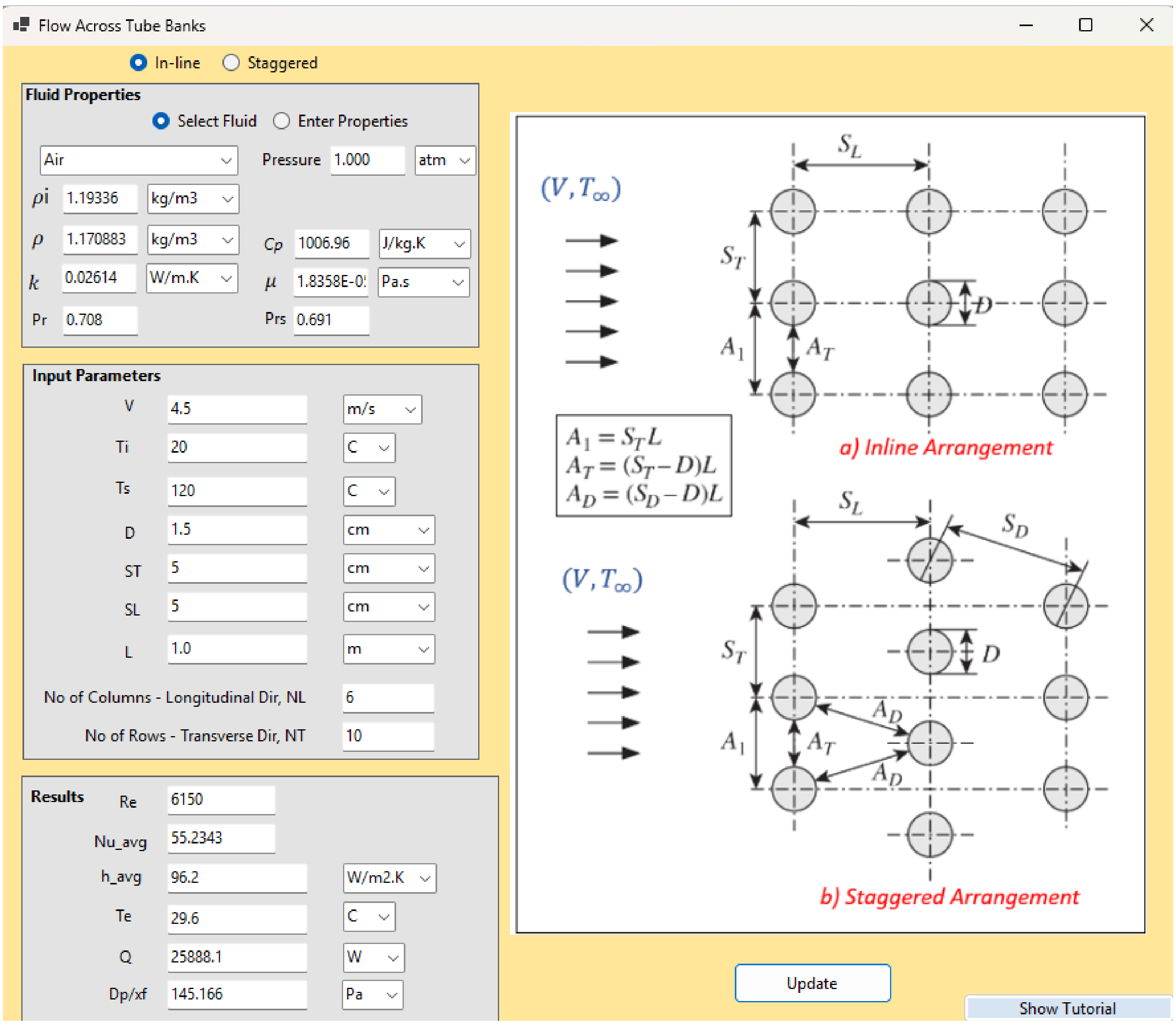
- Open "*Flow over Tube Banks*" Tool





$N_L = 6$
$N_T = 10$
$L = 1\ m$

- Click Update and Solve



Comments:

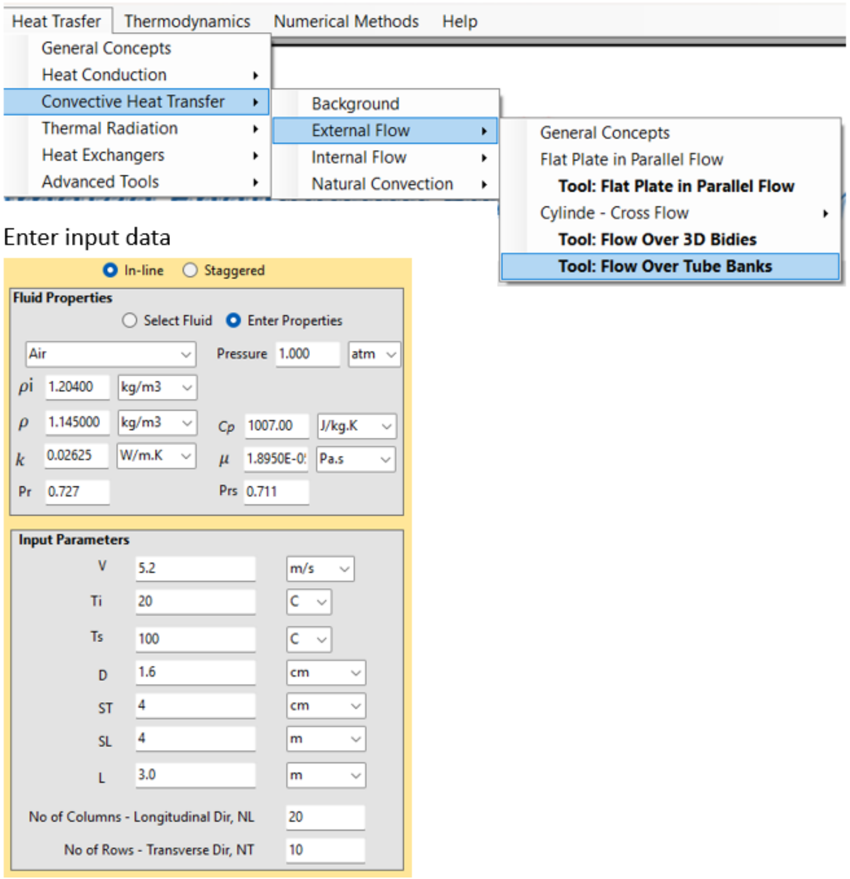
5. The Reynolds number, *Re*=6150 and it is based on V_{max}
6. Average heat transfer coefficient is 96.2 W/m.K
7. Air exit temperature is 29.6 °C
8. Heat transfer into air is 25888.1 W (25.8 kW)
9. Pressure drop across the bank can be calculated to be 23.2 Pa, by multiplying *Dp/xf* value of 145.66 Pa by the product of the friction factor and the correction factor (roughly 0.16 and 1.0 from the chart).
10. Note that, following the solution process, values fluid thermal properties are displayed corresponding to the latest updated film temperature. The user can use fixed properties by selecting "Enter Properties" option.
11. *pi* (1.193 shown in the first density box) represents the value of density at the inlet (evaluated using inlet temperature). This value is used to determine the mass flow rate through the bank.

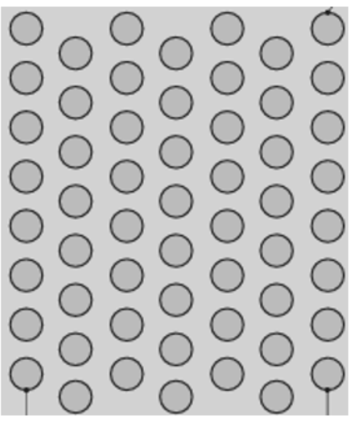
Example – Air Heating by Steam Tubes [Source: Cengel-Ghagar Problem 7.112]

Air is to be heated by passing it over a bank of 3-m-long tubes inside which steam is condensing at 100 °C. Air approaches the tube bank in the normal direction at 20 °C and 1 atm with a mean velocity of 5.2 m/s. The outer diameter of the tubes is 1.6 cm, and the tubes are staggered with longitudinal and transverse pitches of S = ST = 4 cm. There are 20 rows in the flow direction with 10 tubes in each row. Determine the rate of heat transfer.

Solution:

- Open "*Flow over Tube Banks*" Tool





Staggered Arrangement

$D = 1.6\ cm$

$S_T = S_L = 4\ cm$

$N_T = 10; N_L = 20$

$V = 5.2\ m/s$

Document last modified: 9/21/2023

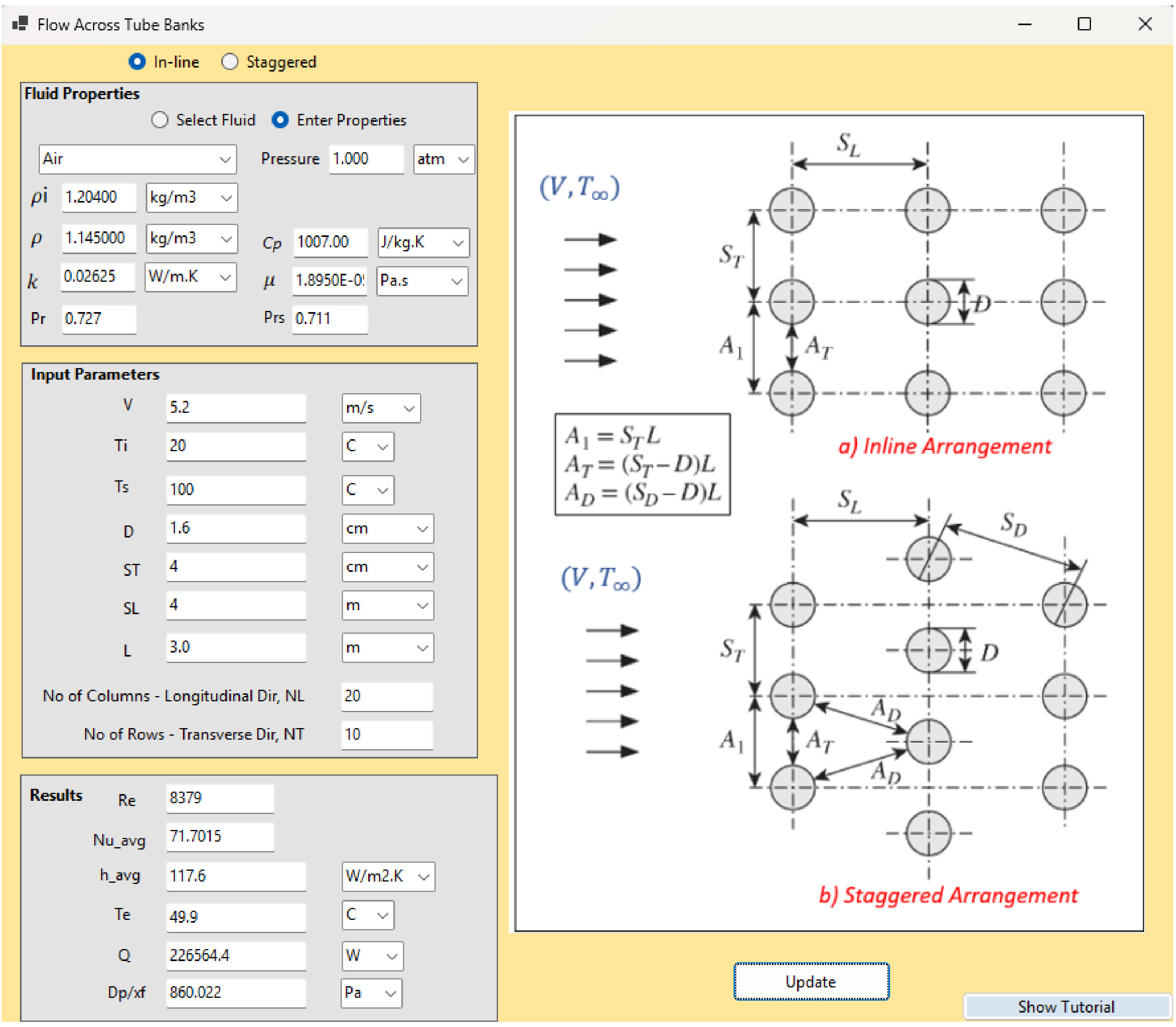
$T_i = 20\ °C$

$T_S = 100\ °C$

$L = 3\ m$

Note that, for this example, user properties (fixed properties) are used instead of fluid selection.

- Click Update and Solve



Comments:

1. The Reynolds number, *Re*=8379 is based on V_{max}
2. Average heat transfer coefficient is 117.6 W/m.K
3. Air exit temperature is 49.9 °C
4. Heat transfer into air is 226564.4 W (22.7 kW)

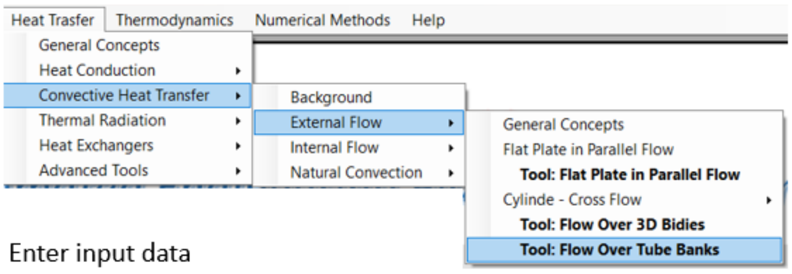
5. Pressure drop across the bank can be calculated to be 283.8 Pa, by multiplying *Dp/xf* value of 860.022 Pa by the product of the friction factor and the correction factor (roughly 0.33 and 1.0 from the chart).

Example 7.7– Space Heating Using Pressurized Water [Source: Bergman-Lavine Example 7.7]

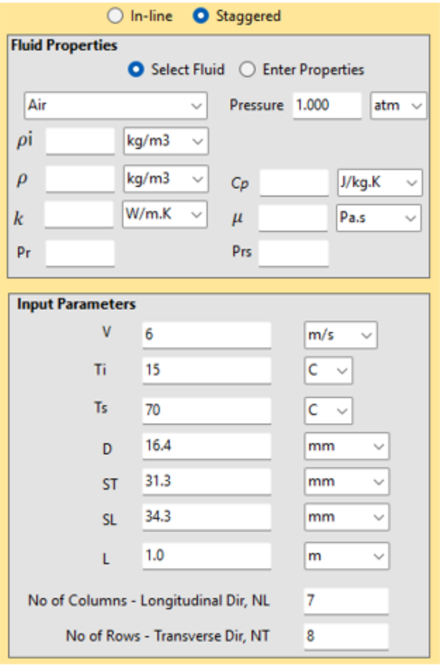
Pressurized water is often available at elevated temperatures and may be used for space heating or industrial process applications. In such cases it is customary to use a tube bundle in which the water is passed through the tubes, while air is passed in cross flow over the tubes. Consider a staggered arrangement for which the tube outside diameter is 16.4 mm and the longitudinal and transverse pitches are S_L = 34.3 mm and S_T = 31.3 mm. There are seven rows of tubes in the airflow direction and eight tubes per row. Under typical operating conditions the cylinder surface temperature is at 70 °C, while the air upstream temperature and velocity are 15 °C and 6 m/s, respectively. Determine the air-side convection coefficient and the rate of heat transfer for the tube bundle.

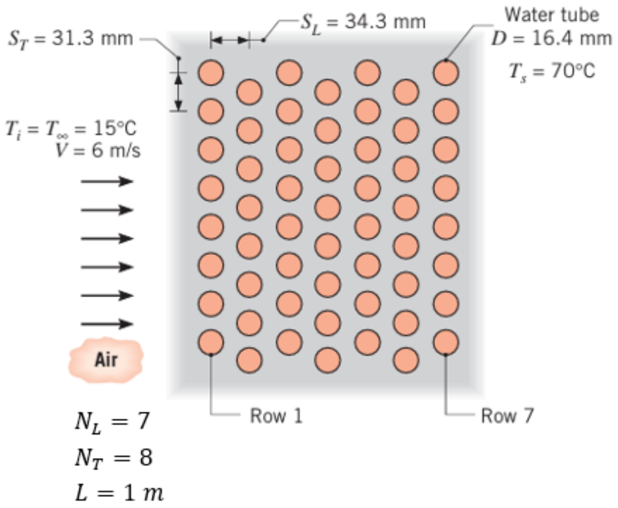
<u>Solution:</u>

- Open "*Flow over Tube Banks*" Tool

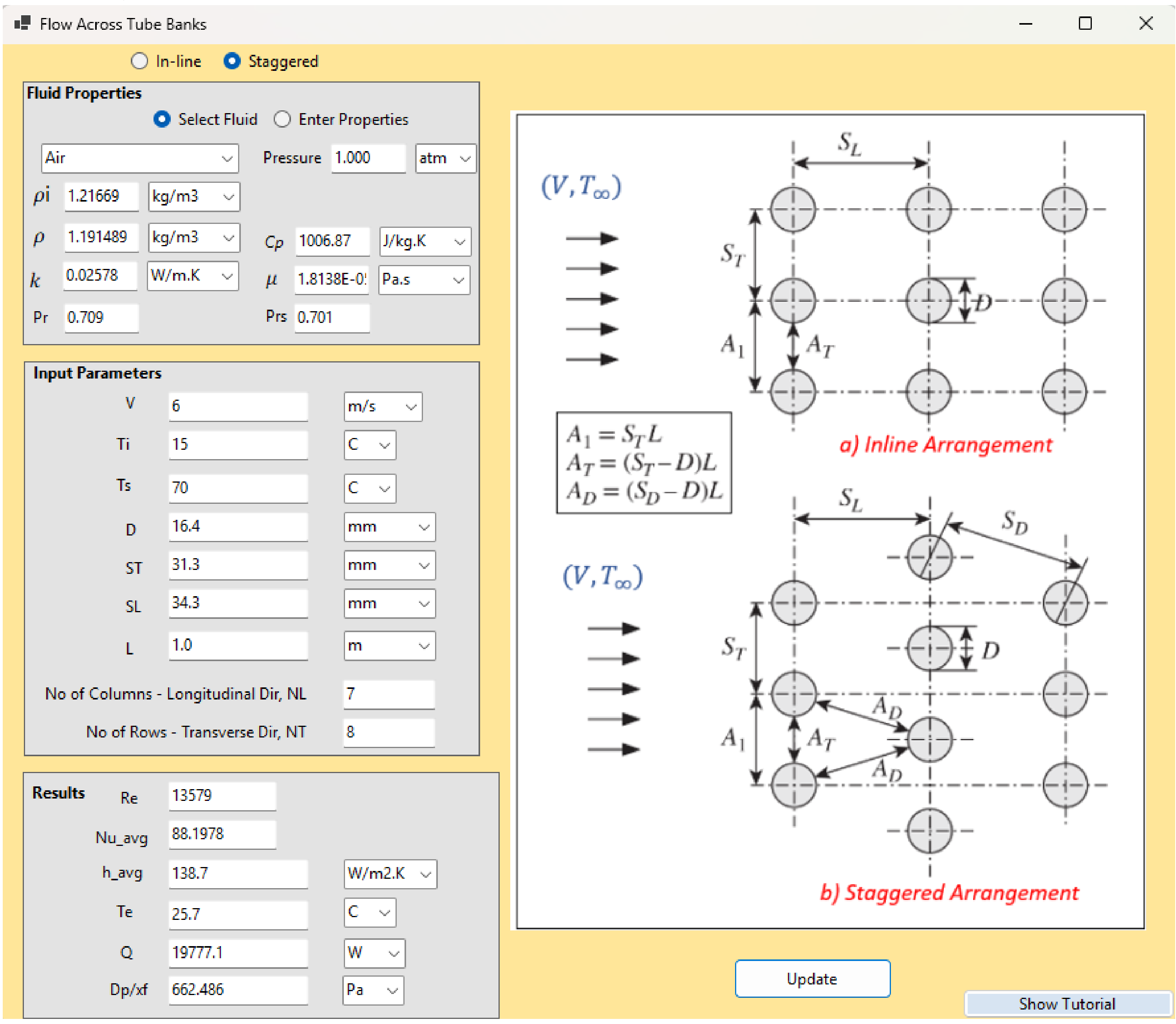


Enter input data





- Click Update and Solve



Comments:

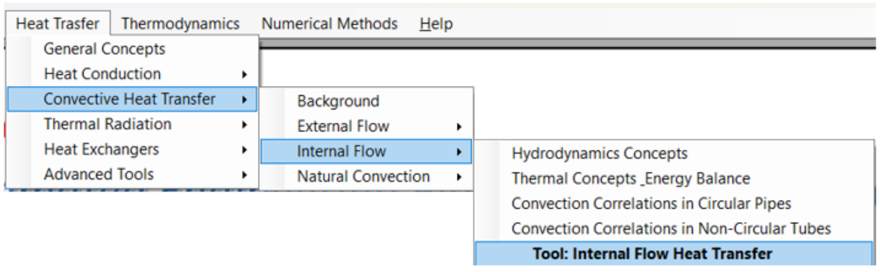
1. The Reynolds number, *Re*, is 13,579 and it is based on V_{max}
2. Average heat transfer coefficient is 138.7 W/m.K
3. Air exit temperature is 25.7 °C
4. Heat transfer into air is 19,777 W (19.8 kW)
5. Pressure-drop across the bank can be calculated to be 241.15 Pa, by multiplying *Dp/xf* value of 662.49 Pa by the product of the friction factor and the correction factor (roughly 0.332 and 1.04 from the chart).
6. Note that, following the solution process, values fluid thermal properties are displayed corresponding to the latest updated film temperature.

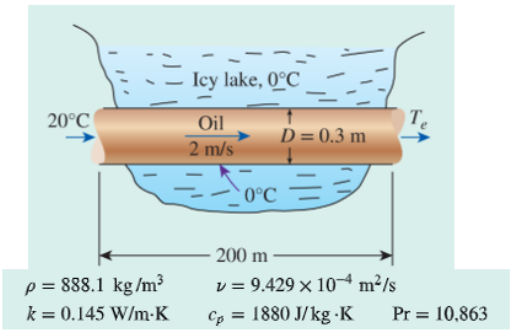
HT-15: Internal Flow Heat Transfer

Example – Developing laminar Flow of Oil in a Pipeline Through a Lake [Source: Cengel-Ghagar Example 8.3]

Consider the flow of oil at 20 °C in a 30-cm-diameter pipeline at an average velocity of 2 m/s. A 200-m-long section of the horizontal pipeline passes through icy waters of a lake at 0C. Measurements indicate that the surface temperature of the pipe is very nearly 0 °C. Disregarding the thermal resistance of the pipe material, determine (a) the temperature of the oil when the pipe leaves the lake, (b) the rate of heat transfer from the oil, and (e) the pumping power required to overcome the pressure losses and to maintain the flow of the oil in the pipe.

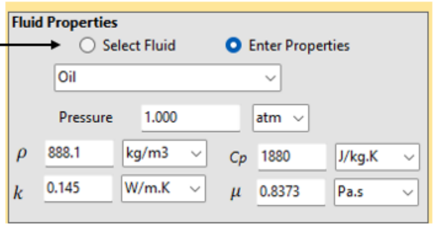
Solution:



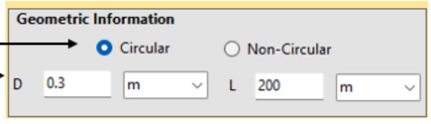


Enter input data

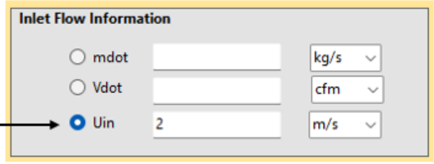
- *Select "Enter Properties" and enter values.*



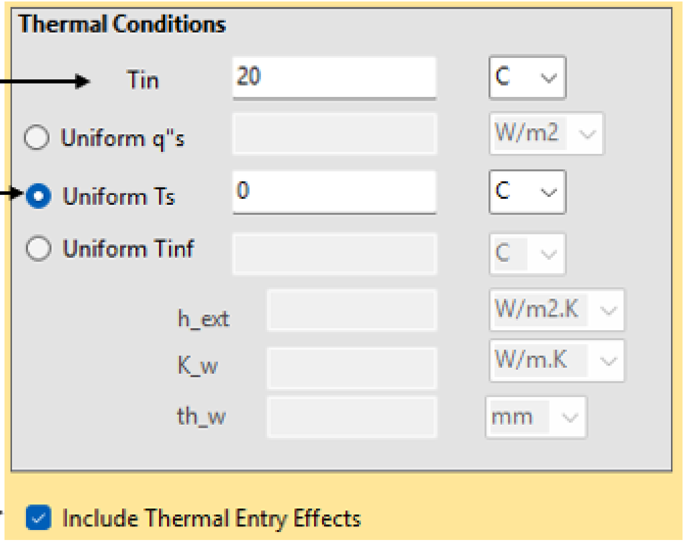
- *Keep "Circular" cross-section.*
- *Enter given values for diameter and tube length in given units.*



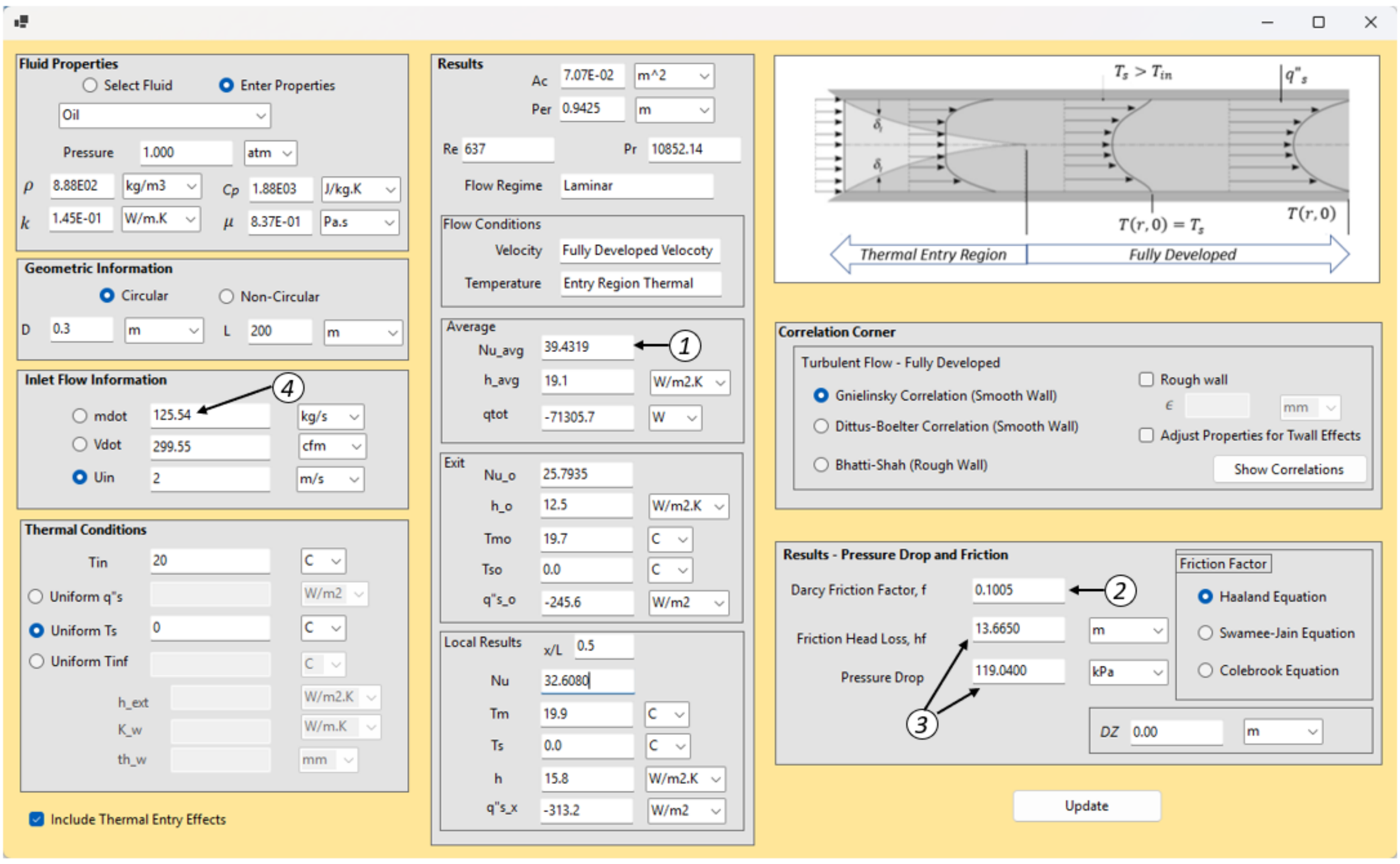
- *Select "Uin" to enter inlet velocity.*



- *Enter the Inlet Temperature.*
- *Select "Uniform Surface Temperature" option and Enter the value provided.*
- *Make sure Thermal Entry Effects are included.*



Click Update to Solve



Comments:

1. It is evident from the Nusselt number value (being much larger than the fully-developed value of 3.66) that the thermal profile is not developed and we are operating in the thermal entry region. This is typical of "high Prandtl number" fluids (here *Pr* = 10852).
2. The friction coefficient is calculated to be 0.01005. This value corresponds to laminar flow in circular pipes $\left(f = \frac{64}{Re}\right)$.
3. The head loss (h_f) and corresponding Δp are calculated to be 13.66 m and 119.04 kPa
4. The mass flow rate is 125.54 kg/s. The required pumping power can be obtained from:

$$\dot{W}_P = \frac{\dot{m}\Delta p}{\rho} = \frac{(125.54)(119.04)}{(888.1)} = 16.8\ kW$$

Example – Developing Laminar Flow with Uniform Heat Flux – [Source: Lienhard Example 7.2]

A fully developed flow of air at 27C moves at 2 m/s in a 1 cm I.D. pipe. An electric resistance heater surrounds the last 20 cm of the pipe and supplies a constant heat flux to bring the air out at Tb = 40C. What power input is needed to do this? What will be the wall temperature at the exit?

Example – Low Prandtl Number Flow – Liquid Mercury Flow in a Pipe [Source: Cengel-Ghagar Problem 8.125]

Liquid mercury flows at 0.6 kg/s through a 5-cm diameter tube, with inlet mean temperature of 100 °C. T surface temperature is kept constant at 250 °C.

a) Determine the outlet mean temperature at x=50 cm.

b) Determine the rate of heat transfer to mercury for this length of pipe.

Example – Prescribed External Temperature and Convection Coefficient – Hot Air in Cold Ambient Environment [Source: Bergman-Lavine Example 8.6]

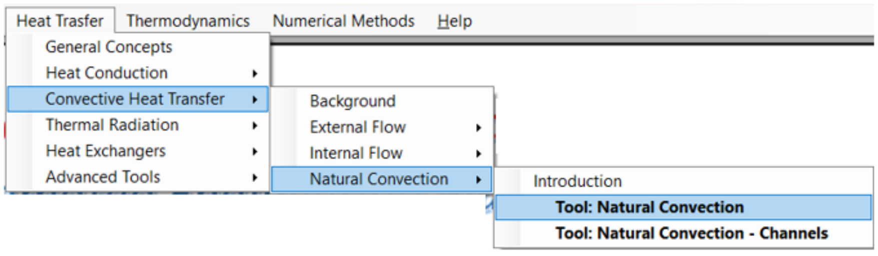
Hot air, with an inlet temperature of 103°C, flows with a mass rate of 0.050 kg/s through an uninsulated sheet metal duct of diameter D = 0.15 m and L = 5 m, which is in the crawlspace of a house. The heat transfer coefficient between the duct outer surface and the ambient air at T_∞ = 0°C is known to be h_{ext} = 6 W/(m^2 · K).

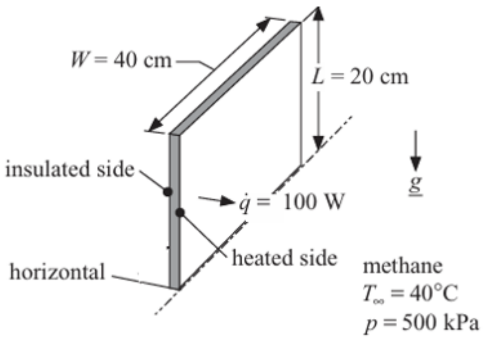
1) Calculate the rate of heat loss (W) from the duct over the length L.
2) Determine the heat flux and the duct surface temperature at x = L.

HT-16: Natural Convection over Bodies

Example – Vertical Flat Plate – [Source: Nellis and Klein Example 6.2-1 Modified]

A rectangular plate heater is placed in the ullage space of a fuel tank on a military aircraft (as shown below). One side of the heater is insulated and the other is heated. The heater is oriented vertically with respect to gravity and achieves a nearly uniform temperature. The length of the heater is L = 20 cm, and the width is W = 40 cm. The plate is exposed to fuel that has properties consistent with methane at T_∞ = 40 ◦C. Assuming heater power of 100 W, determine the surface temperature of the heater for a) Fuel at atmospheric pressure, b) Fuel at p =500 kPa.





HT-17: Natural Convection Vertical Channels

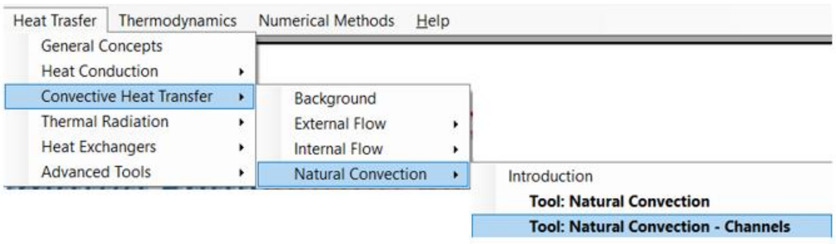
Example – Heat Sink Fin Spacing – [Source: Cengel-Ghagar Example 9.3]

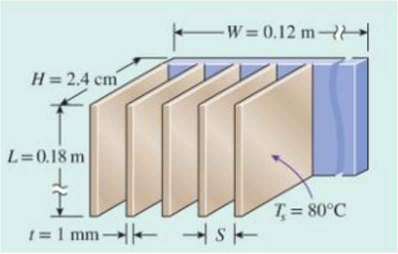
A 12-cm-wide and 18-cm-high vertical hot surface in 30 °C air is to be cooled by a heat sink with equally spaced fins of rectangular profile. The fins are 1 mm thick and 18 cm long in the vertical direction and have a height of 2.4 cm from the base. Assuming the base temperature of 80 °C.

a) Determine the rate of heat transfer by natural convection from the heat sink using 20 fins.
b) Determine the rate of heat transfer by natural convection from the heat sink using optimal fin spacing.
c) Determine the rate of heat transfer from the heat sink using optimal fin spacing and including thermal radiation (emissivity=0.25).

Solution:

a)

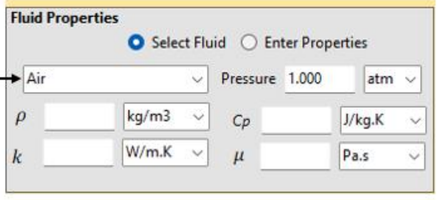




Enter input data

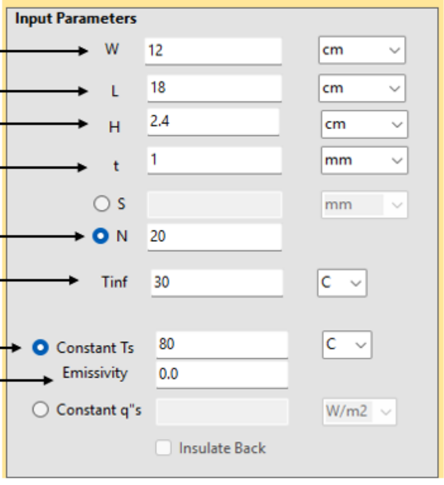
Enter Fluid Properties:

- Select "Air" from Fluid Library.



Enter Input:

- Enter the Width of Heat Sink
- Enter the Vertical Height of
- Enter the Height from Base
- Enter the Fin Thickness
- Enter Number of Fins
- Enter Ambient Temperature
- Enter Sink Temperature
- Enter $\epsilon = 0$ to Neglect Radiation



Note:
1. The used can choose to provide number of fins or fin spacing.
2. The user may select specified sink temperature or constant heat flux.

Click "Update" to Solve.

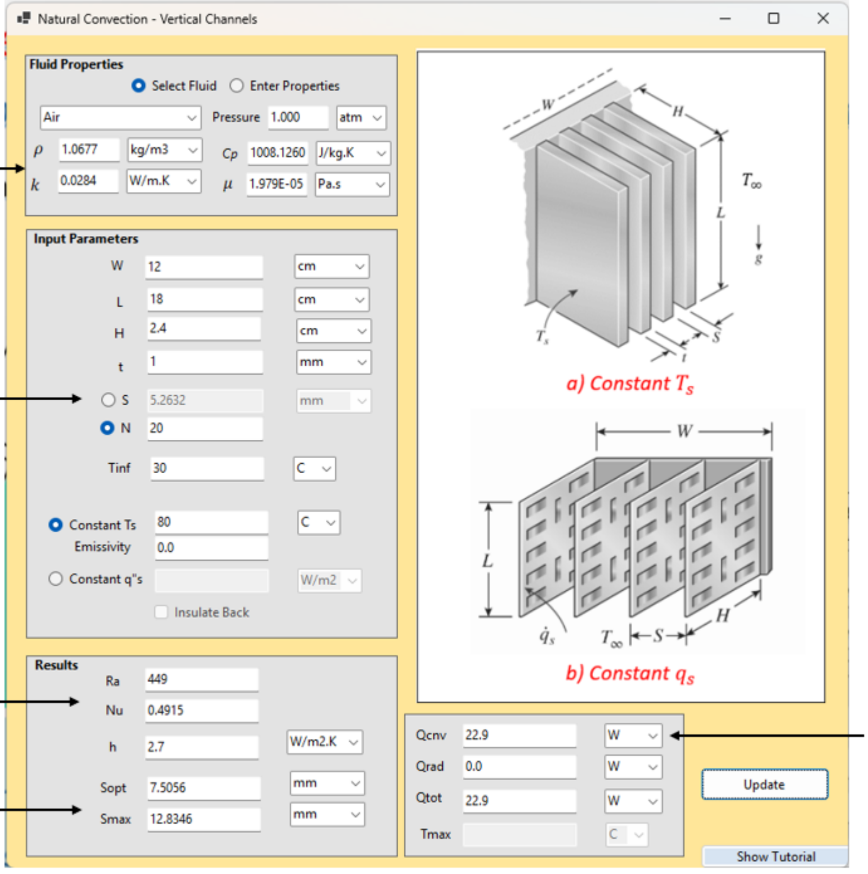
Completed Form:

Fluid properties at T_{film} are displayed.

Fin spacing is calculated and displayed.

Rayleigh number, Nusselt number and the heat transfer coefficient.

S_{opt} is the spacing that will result in max. total heat transfer rate. Smax results in max heat transfer per fin.



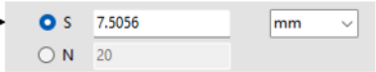
Calculated Heat Transfer rate.

b)

To use optimal fin spacing:

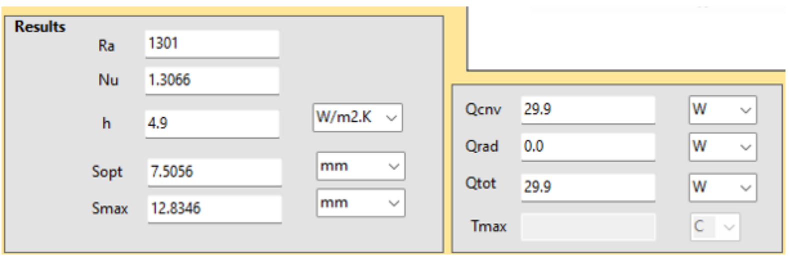
1. Change input *from Number of Fins* to *Spacing*.
2. Copy/Paste the value of the Optimal Spacing from the results to input.

Leave all other parameters unchanged.

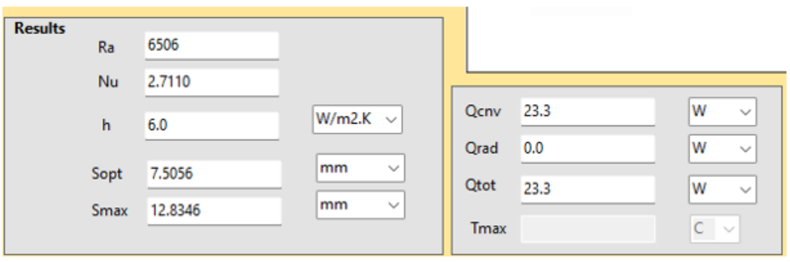


Click "Update" to Solve.

- Heat transfer coefficient is increased.
- Heat rate is increased from 22.9 W to 29.9 W.

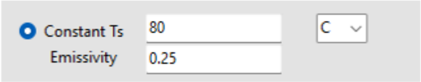


Note: If you repeat the solution using S_{max} for the spacing, the heat transfer coefficient will increase to 6 W/m.k, but the total rate of heat transfer will decrease to 23.3 W due to reduced number of fins.



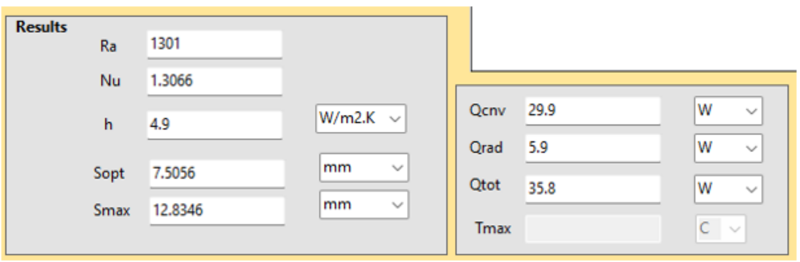
c)

To Include thermal radiation, enter a non-zero value for emissivity:



Click "Update" to Solve.

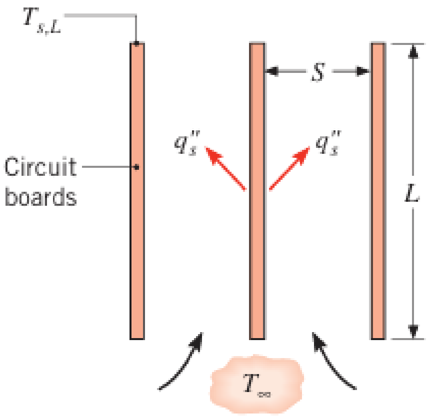
- This will result in an additional 5.9 W heat loss due to radiation, bringing the total heat transfer rate to ambient to 35.8 W.



Note: The effective radiation surface area is much smaller that the convection surface area due to close spacing between the fins. Therefore, addition of fins do not significantly enhance the radiation heat transfer here.

Example – Natural Convection Cooling of Vertical PCBs – [Source: Bergman-Lavine Problem 9.59]

A vertical array of circuit boards is immersed in quiescent ambient air at T_∞ = 17°C. Although the components protrude from their substrates, it is reasonable, as a first approximation, to assume flat plates with uniform surface heat flux q''_s. Consider a 60 cm wide heat sink with boards of height and length H = L = 40 cm with 1.5 mm thickness and spacing S = 25 mm. Assuming each board is populated on both sides and dissipates 88 W, determine maximum board temperature and total heat dissipation of this unit.



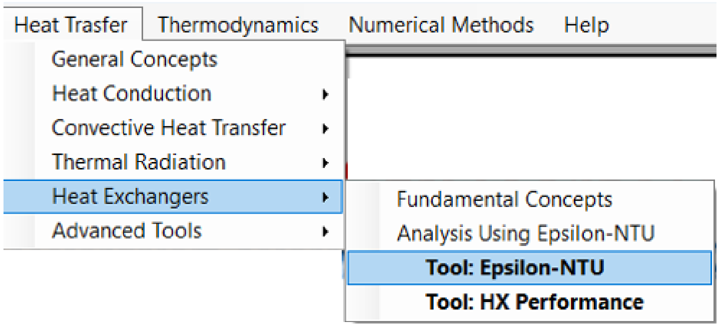
HT-18: Heat Exchanger Epsilon/NTU Calculator

Example – Heat Exchanger Effectiveness-NTU Relationships

Complete the following table using the automated Epsilon-NTU Tool in AutoTherm.

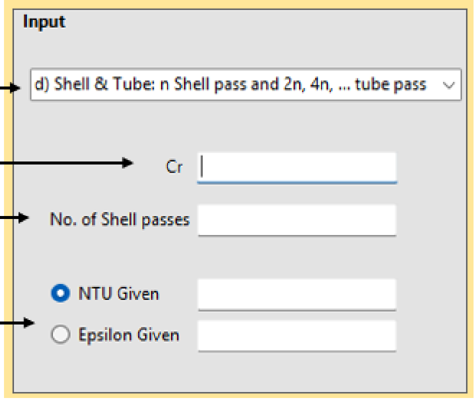
Heat Exchanger Flow Arrangement	Capacity Ratio, Cr	NTU	Epsilon
Parallel Flow	0.5	2.5	
Counter Flow	0.5	2.5	
Shell-Tube 2 Tube Passes; Single Shell Pass	0.5	2.5	
Shell-Tube 2 Tube Passes; 2 Shell Pass	0.5	2.5	
Cross-Flow Both Fluids Unmixed	0.5	2.5	
Cross-Flow Both Fluids Unmixed	0.75		0.65
Parallel Flow	0.75		0.65

Solution:



Tool utilizes a very simple user input panel shown below:

- Select heat exchanger flow arrangement.
- Enter the capacity ratio.
- Enter Number of shell passes (only for shell-tube).
- Enter NTU to calculate Effectiveness (performance mode), or epsilon to get NTU (design mode).



Heat Exchanger Flow Arrangement	Capacity Ratio, Cr	NTU	Epsilon
Parallel Flow	0.5	2.5	0.651
Counter Flow	0.5	2.5	0.8328
Shell-Tube 2 Tube Passes; Single Shell Pass	0.5	2.5	0.7237
Shell-Tube 2 Tube Passes; 2 Shell Pass	0.5	2.5	0.802
Cross-Flow Both Fluids Unmixed	0.5	2.5	0.7911
Cross-Flow Both Fluids Unmixed	0.75	2.1165	0.65
Parallel Flow	0.75	No Solution	0.65

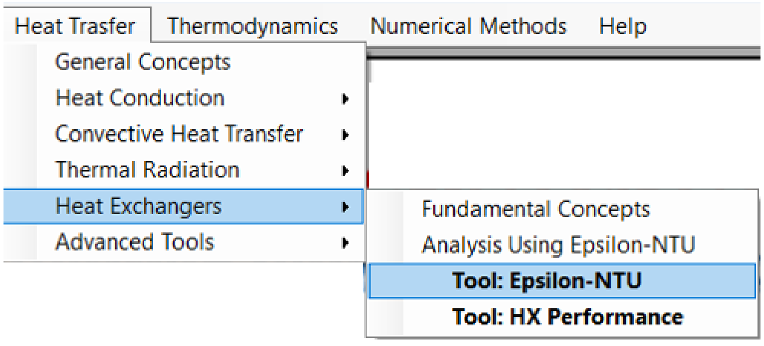
Note: Epsilon-NTU relations always provide value for effectiveness for all positive NTU's. However, as seen from the last row, not every positive epsilon will result in a valid NTU.

HT-19: Heat Exchanger Performance Analysis Tool

Example – Heat Exchanger Performance – [Source: Nellis and Klein Example 8.3-1]

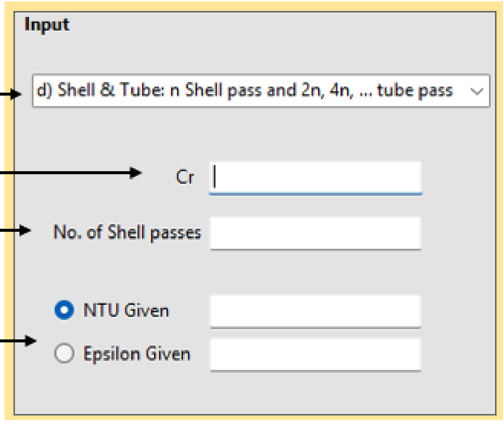
The cross-flow heat exchanger with both fluids unmixed is used to heat air with hot water. Water enters the heat exchanger tubing with a mass flow rate, of 0.03 kg/s and 60 °C. Air at 20 °C and atmospheric pressure is blown across the heat exchanger with a volumetric flowrate of 0.06 m^3/s. The conductance of this heat exchanger has been calculated to be 58.4 W/K, based on the compact heat exchanger correlations. Determine the outlet temperatures of the water and air and the heat transfer rate using the ε-NTU method.

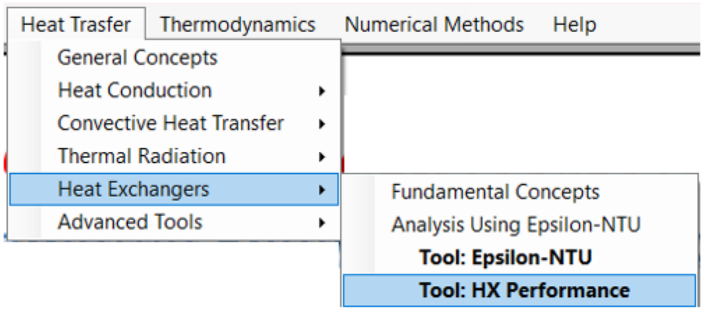
Solution:

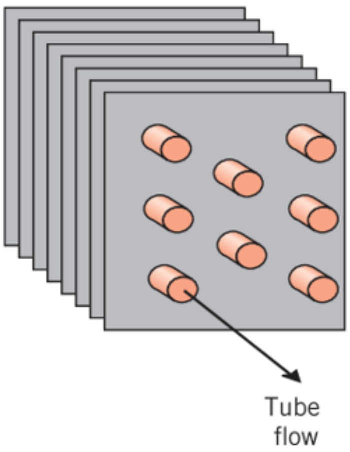


Tool utilizes a very simple user input panel shown below:

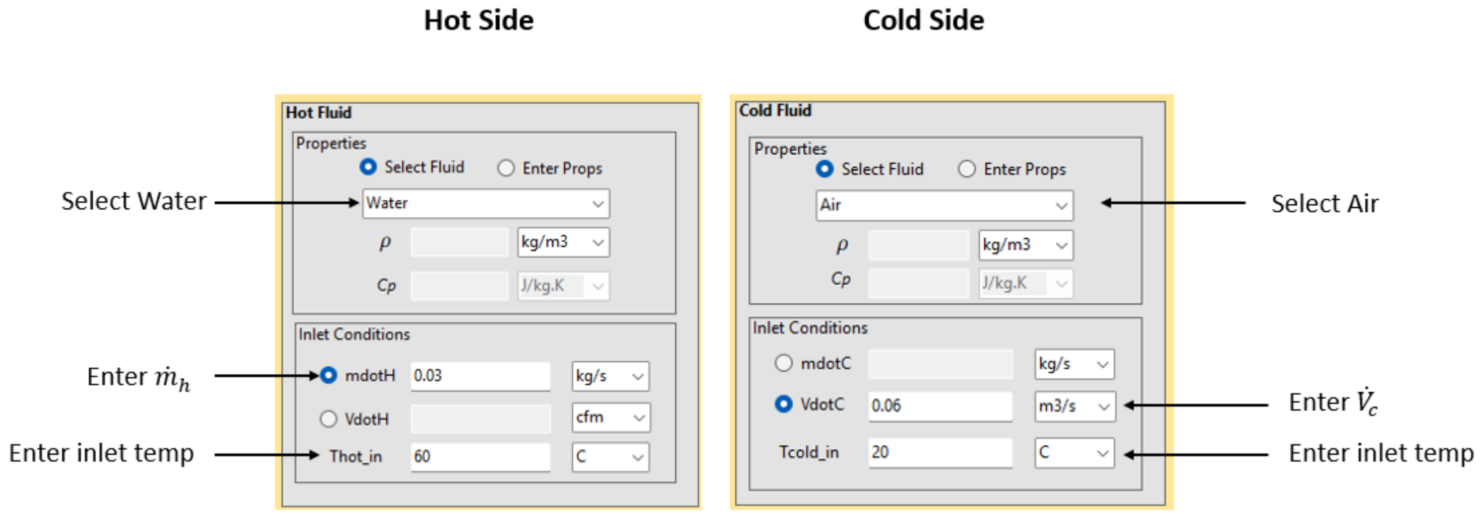
- Select heat exchanger flow arrangement.
- Enter the capacity ratio.
- Enter Number of shell passes (only for shell-tube).
- Enter NTU to calculate Effectiveness (performance mode), or epsilon to get NTU (design mode).

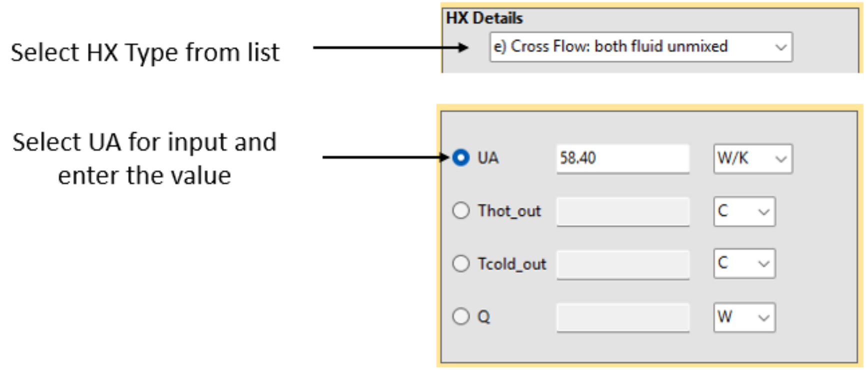




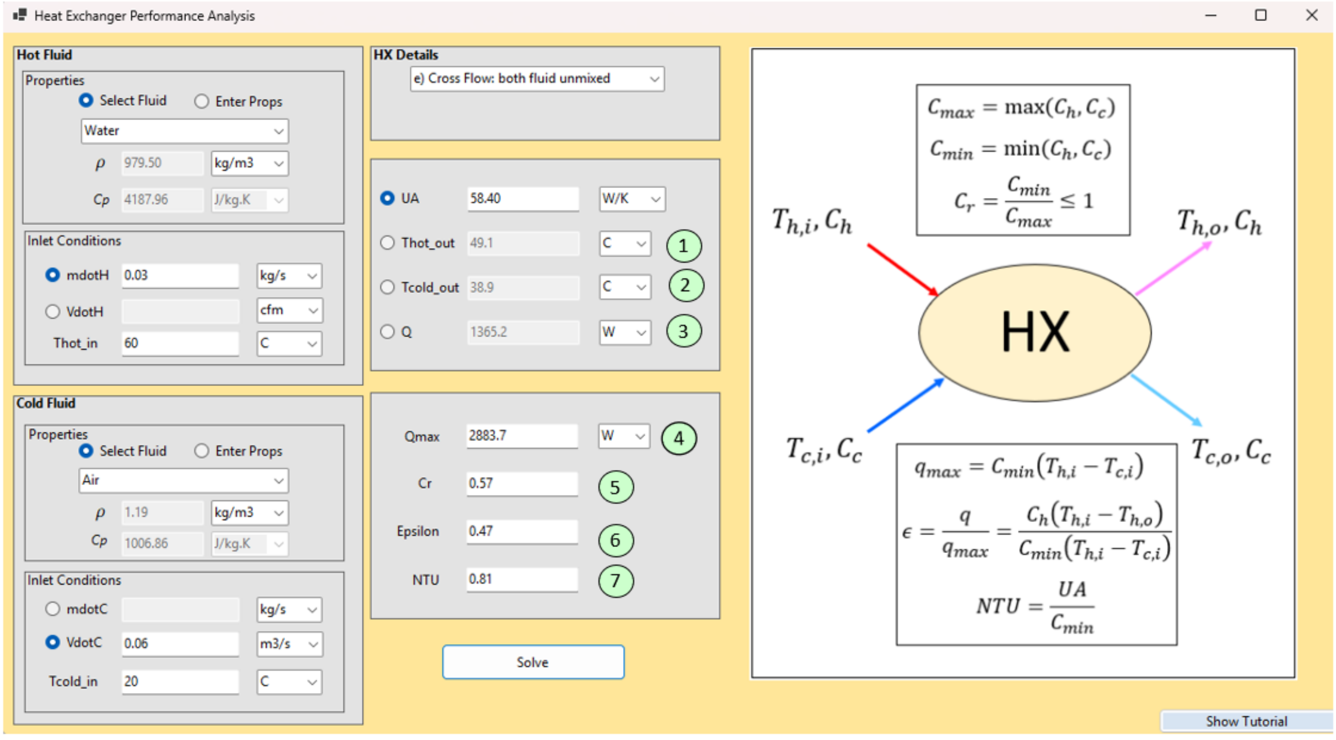


Enter input data





Click "Solve".

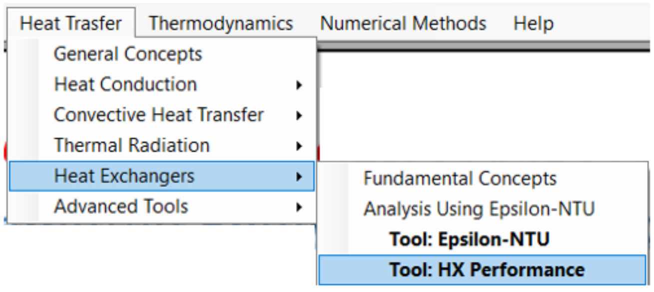


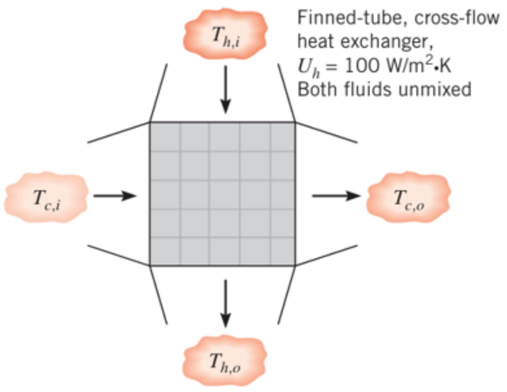
1. The outlet temperature of water is 49.1 °C.
2. The outlet temperature of air is 38.9 °C.
3. The total rate of heat transfer between water and air is 1365.2 W.
4. The maximum heat transfer rate (Q_{max}) is 2883.7 W.
5. The capacity ratio is calculated to be 0.57.
6. The heat exchanger effectiveness is calculated to be 0.47.
7. The number of transfer units (NTU) is 0.81.

Example – Heat Exchanger Design – [Source: Bergman-Lavine Problem 9.59]

Hot exhaust gases (c_p=*1080* J/kg.K), which enter a finned-tube, cross-flow heat exchanger with a flow rate of 1.75 kg/s at 300°C and leave at 100°C, are used to heat pressurized water at a flow rate of 1 kg/s from 35 °C. The overall heat transfer coefficient based on the gas-side surface area is U_h = 100 W/m2.K. Determine the required gas-side surface area A_h and outlet temperature of water.

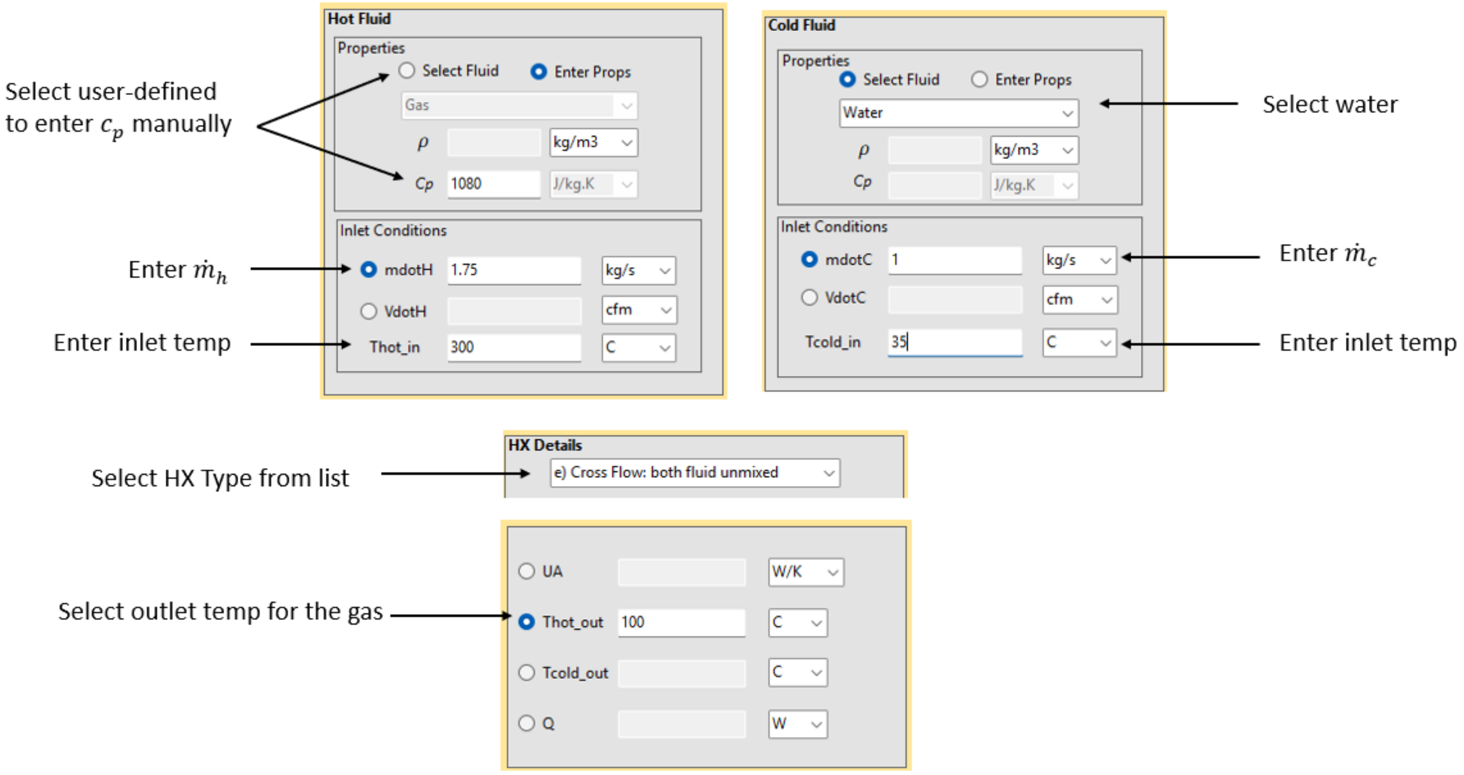
Solution:



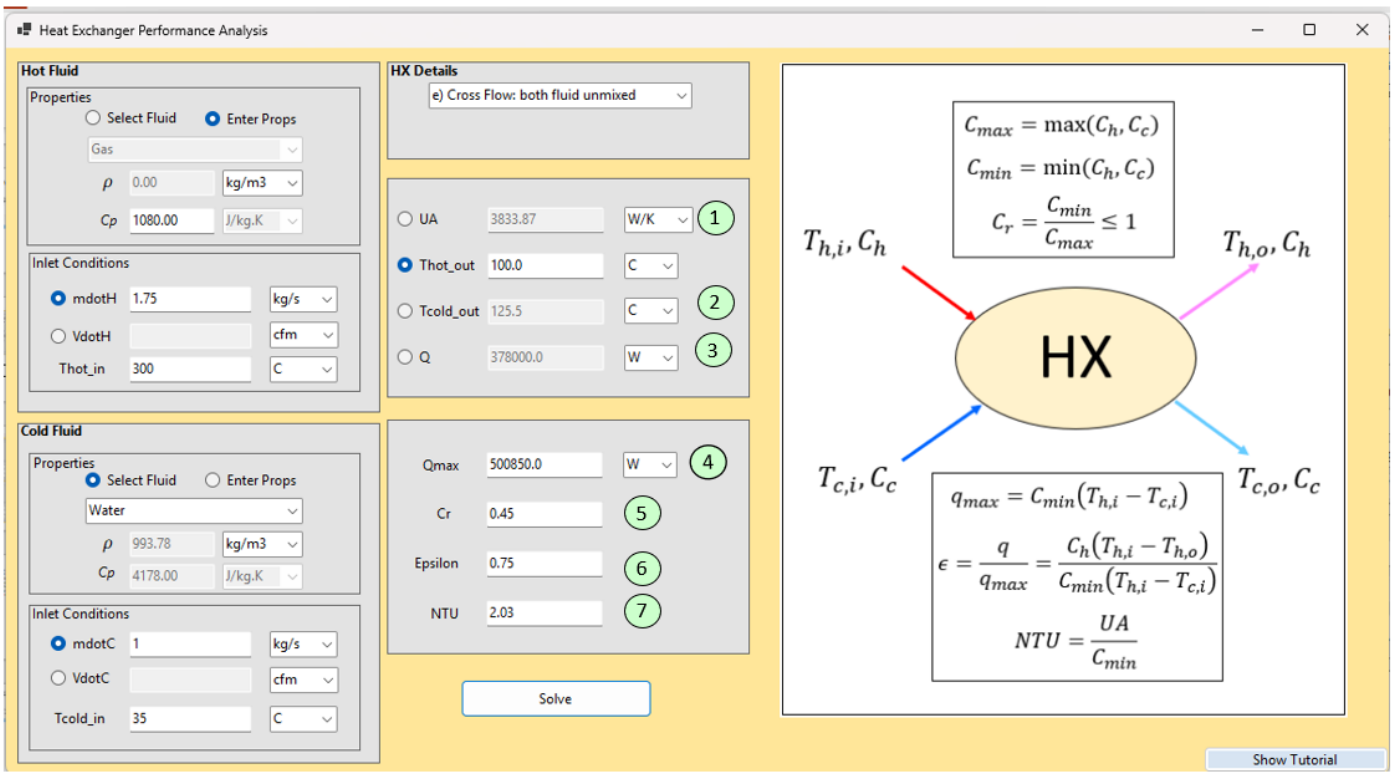


Enter input data

Hot Side **Cold Side**



Click "Solve".

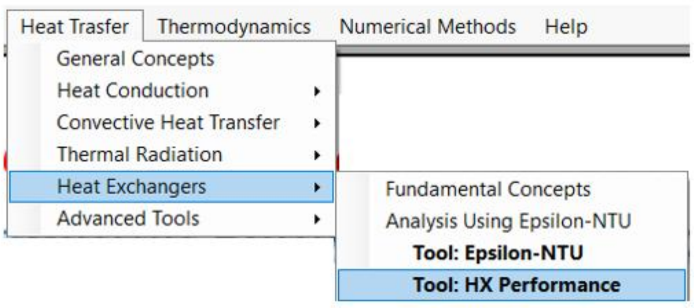


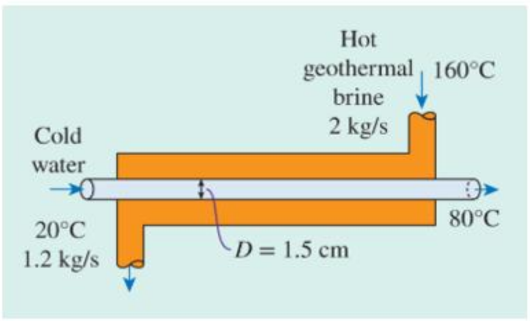
1. The UA is calculated to be 3833.37 W/K.
   Since U is given in the problem statement to be 100 W/m2.K, heat transfer area may be determined from:
   $$A_h = \frac{UA}{U_h} = \frac{3833}{100} \quad \rightarrow \quad A_h = \mathbf{38.33\ m^2}$$
2. The outlet temperature of water is 125.5 °C.
3. The total rate of heat transfer between hot gas and water is 378 kW.
4. The maximum heat transfer rate (Q_{max}) is 500850 W.
5. The capacity ratio is calculated to be 0.45.
6. The heat exchanger effectiveness is calculated to be 0.75.
7. The number of transfer units (NTU) is 2.03.

Example – Heat Exchanger Design – [Source: Cengel-Ghagar Example 11.8]

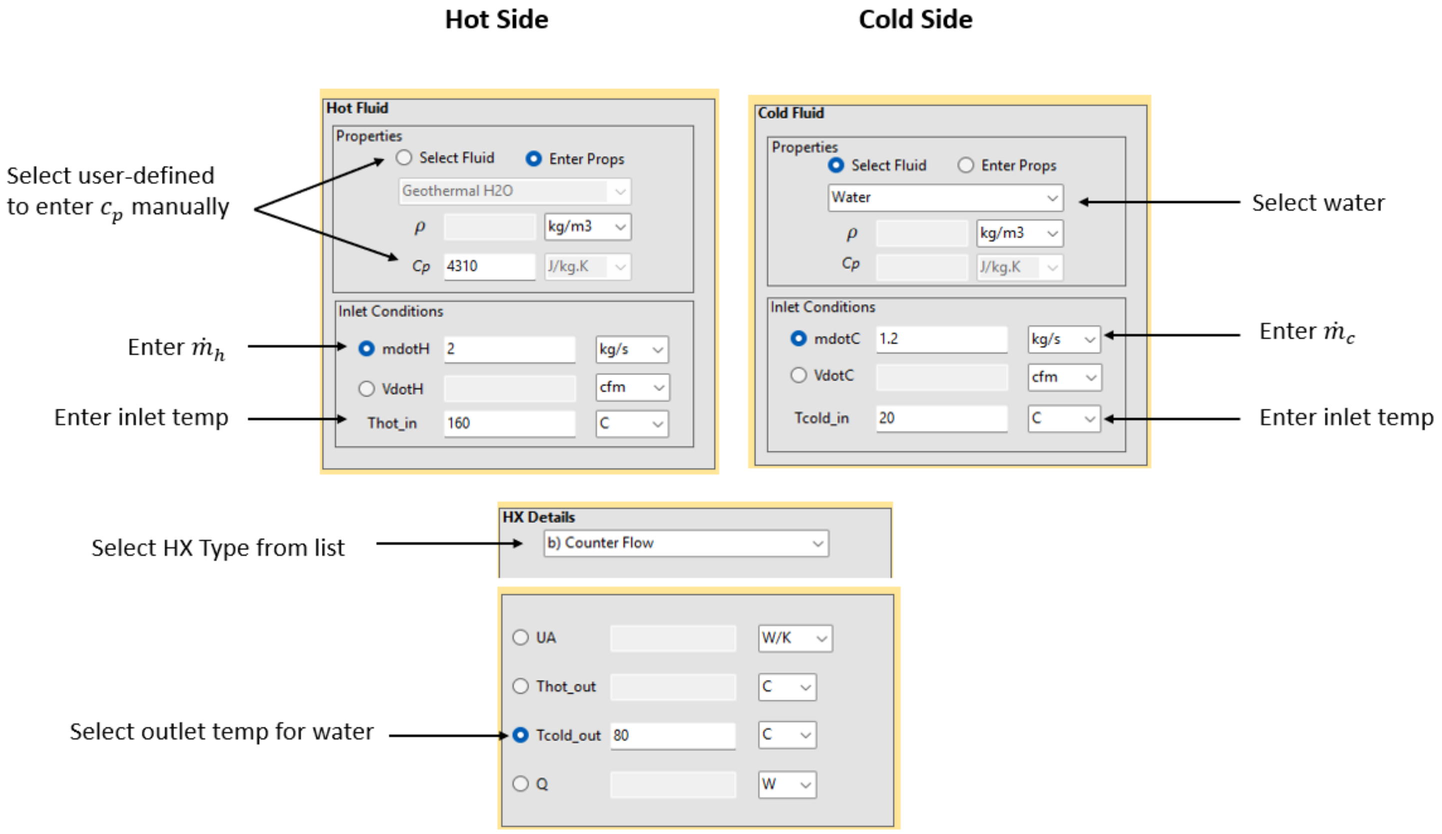
A counterflow double-pipe heat exchanger is to heat water from 20 C to 80 C at a rate of 1.2 kg/s. The heating is to be accomplished by geothermal water available at 160 C at a mass flow rate of 2 kg/s (assume c_p=*4310* J/kg.K). The inner tube is thin-walled and has a diameter of 1.5 cm. The overall heat transfer coefficient of the heat exchanger is 640 W/m2. K, determine the length of the heat exchanger required to achieve the desired heating.

Solution:

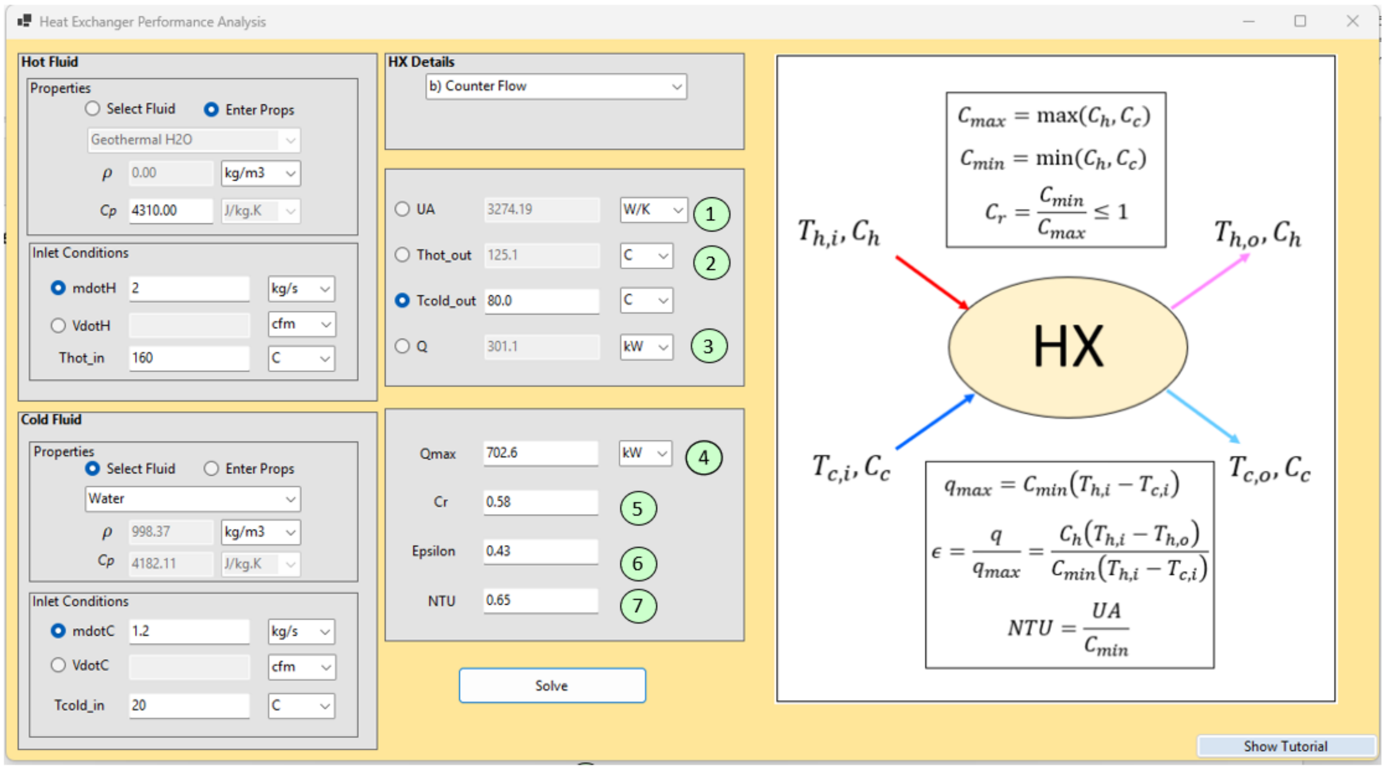




Enter input data



Click "Solve".



1. The UA is calculated to be 3274.2 W/K.
   Since U is given in the problem statement to be 640 W/m2.K, The length can be found:

$$A_s = \pi DL = \frac{UA}{U_h} \Rightarrow L = \frac{UA}{(U)(\pi D)} = \frac{(3574.2)}{(640)(\pi)(0.015)} \Rightarrow \boldsymbol{L = 108.6\ m}$$

2. The outlet temperature of geothermal brine is 125.1 °C.
3. The total rate of heat transfer between hot gas and water is 301.1 kW.
4. The maximum heat transfer rate (Q_{max}) is 702.6 kW.
5. The capacity ratio is calculated to be 0.58.
6. The heat exchanger effectiveness is calculated to be 0.43.
7. The number of transfer units (NTU) is 0.65.

HT-20: Radiation View factor Calculator