



AutoTherm

Heat Transfer Validation Cases & Tutorial

AutoTherm Heat Transfer Tools – Validation and Tutorials

- 01: Heat Conduction Composite Systems (Walls/Cylinders/Spheres)
- 02: K effective Parallel/Series
- 03: K effective Vias
- 04: Conduction with Uniform Heat Generation (Walls/Cylinders/Spheres)
- 05: Fin Efficiency/Effectiveness
- 06: Heat Sink Analysis
- 07: Multidimensional Heat Transfer in Common Configurations
- 08: Lumped Capacity (Constant Ambient Temperature/ varying Ambient Temperature)
- 09: Transient Conduction (Walls, Cylinders, Spheres)
- 10: Semi-Infinite Solids
- 11: Contact of Two Semi-Infinite Solids
- 12: Flat Plate in Parallel Flow
- 13: Flow Over 3D Bodies
- 14: Flow Over Tube Banks
- 15: Internal Flow Heat Transfer
- 16: Natural Convection over Bodies
- 17: Natural Convection Vertical Channels
- 18: Heat Exchanger Epsilon/NTU Calculator
- 19: Heat Exchanger Performance Analysis Tool
- 20: Radiation View Factor Calculator

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

Input panel showing geometry type selection (Plane Wall, Cylinder, Sphere) and number of layers (3). The "Initialize" button is visible.

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*.

Boundary condition panels for Right BC (x=0) and Right BC (x=L). Both panels show Convection selected with h and T_{amb} inputs. Units are W/m².K and C.

3. Geometric Parameters

Wall: Minimum x value (Default is zero); Wall Size (Default 1m x 1m)

Cylinder: Inner Radius; Cylinder Length

Sphere: Inner Radius

Geometric parameters panel showing X0 (0.0000 cm), L (1.00 m), and W (1.00 m).

4. Thermal Contact Resistance

Interfacial contact resistance may be applied between any layers. Film resistance may also be applied at inner/outer surfaces.

Thermal Contact Resistance panel showing Layer_1/Layer_2 and Value (0.095 C.in²/W) inputs. An "Apply" button is visible.

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.

Layers panel showing a table of layer details. The table has columns for Layer Name, Therm. Cond., and Thickness. The units are mm and W/m.K.

Layer Name	Therm. Cond.	Thickness
Layer_1	0.78	4
Layer_2	0.026	8
Layer_3	0.78	4

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.

Layers					
X	mm	T	C	Rth	C.m2/W
	X Left	T Left	Rth	X Right	T Right
▶	0.000	14.23	0.005	4.000	13.93
	4.000	13.93	0.385	14.000	-8.26
	14.000	-8.26	0.005	18.000	-8.56

2. Overall Results

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

Results		
q"tot	57.7065	W/m2
qtot	69.2478	W
R"tot	0.5199	C.m2/W
U	1.9236	W/m2.K

Example – Composite Wall:

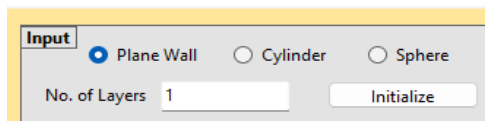
A 0.8 m by 1.5 m glass window (thermal conductivity of 0.78 W/m.K) with a thickness of 8 mm. Assume the interior of the room is maintained at 20 °C with a natural convection heat transfer coefficient of 10 W/m².K, and the external ambient at -10 °C with a heat transfer coefficient of 40 W/m².K (which includes 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.

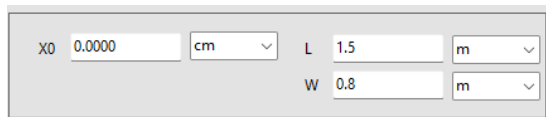
Solution:

a) Single pane

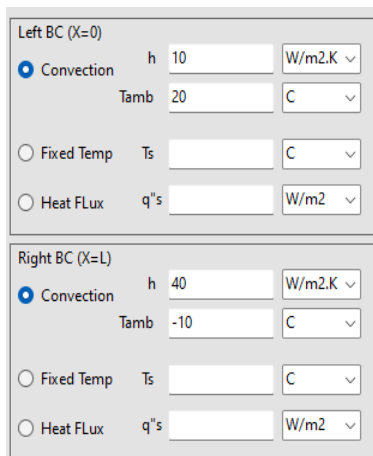
- Open “Heat Conduction Composite Systems” Tool:
- Top menu/Heat Transfer/Heat Conduction/Steady-State 1D/Tool: Composite Wall
- Select **Plane Wall** (default) and set number of layers to “1” (for Case-a) and click **Initialize**.



- Set Width and Length to 0.8 m and 1.5 m



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



- 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:

Heat Conduction - Composit Systems

Input ☒ Plane Wall ☐ Cylinder ☐ Sphere

No. of Layers: 1 Initialize

Right BC (x=0): ☒ Convection h: 10 W/m².K T_{amb}: 20 C ☐ Fixed Temp T_s: C ☐ Heat FLux q''s: W/m²

Right BC (x=L): ☒ Convection h: 40 W/m².K T_{amb}: -10 C ☐ Fixed Temp T_s: C ☐ Heat FLux q''s: W/m²

Thermal Contact Resistance: Xmin: Value: 0.0000 C.m²/W Apply

Layers: Thickness: mm k: W/m.K

Layer Name	Therm. Cond.	Thickness
Layer_1	0.78	8

Results

q''_{tot}: 221.8009 W/m² q_{tot}: 221.8009 W R''_{tot}: 0.1353 C.m²/W U: 7.3934 W/m².K

Layers: X: mm T: C R_{th}: C.m²/W

	X Left	T Left	R _{th}	X Right	T Right
	0.000	-2.18	0.010	8.000	-4.45

Update Show Tutorial

The heat flux is 221.8 W/m². 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

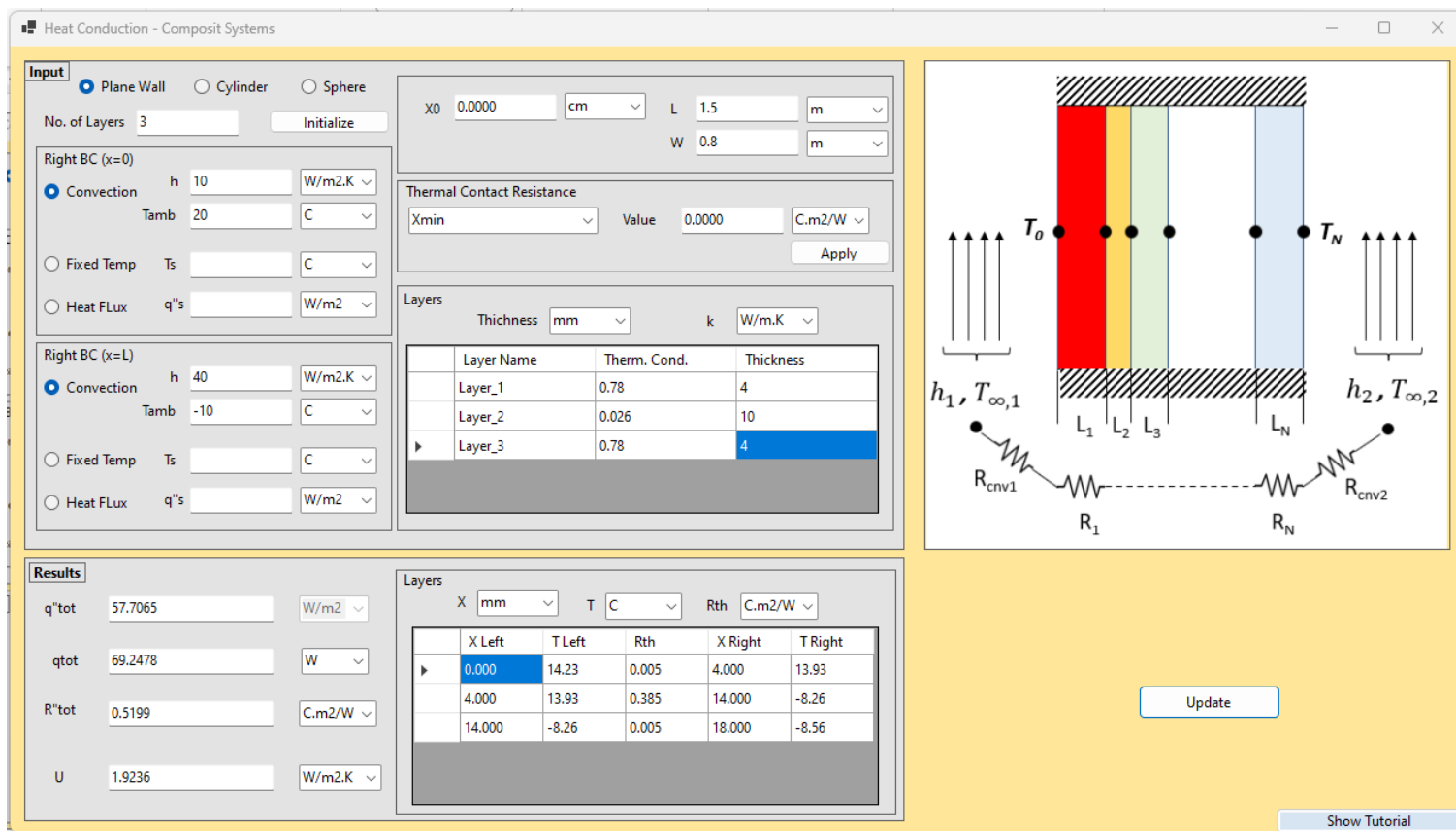
Change **No. of Layers** to **3** and click **Initialize**.

Now, enter thermal conductivity and thickness for all layers, as shown below.

Layers: Thickness: 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 $14.23 \text{ }^\circ\text{C}$, as shown under T_{Left} in the first row above. Note the considerably better performance with the double-pane window.

<<End-of-Tutorial>>

Example – Composite Cylinder:

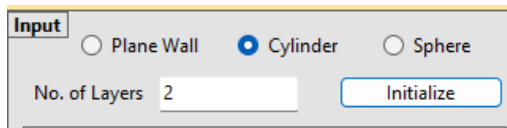
Steam at 320 °C flows in a 5-cm-diameter cast iron pipe ($k = 80 \text{ W/m} \cdot \text{K}$) with a wall thickness of 0.25 cm. The pipe is covered with 3-cm-thick glass wool insulation ($k = 0.05 \text{ W/m} \cdot \text{K}$). Assuming an internal heat transfer coefficient of 60 $\text{W/m}^2\text{°C}$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

Solution:

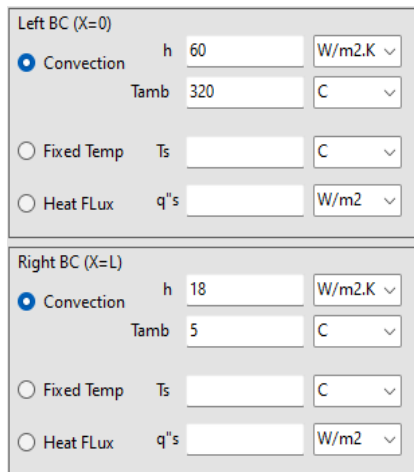
Open “Heat Conduction Composite Systems” Tool:

Top menu/Heat Transfer/Heat Conduction/Steady-State 1D/Tool: Composite Wall

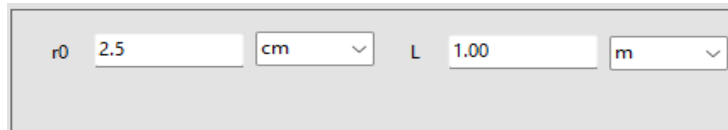
Select **Cylinder** and set the number of layers to “2” and click **Initialize**.



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

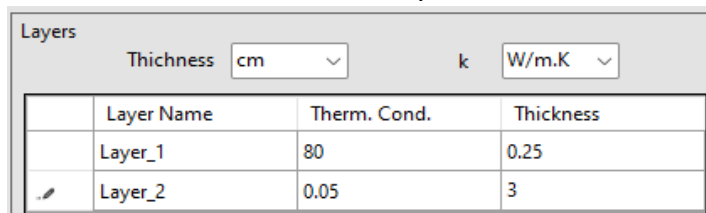


Set $r_0=2.5 \text{ cm}$ and length, $L=1 \text{ m}$ (default).



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:



	Layer Name	Therm. Cond.	Thickness
	Layer_1	80	0.25
	Layer_2	0.05	3

Click [Update](#) to solve. The finished form with results is shown below

Heat Conduction - Composit Systems

☐ Plane Wall
☒ Cylinder
☐ Sphere

No. of Layers 2
Initialize

r0 2.5 cm
L 1.00 m

Left BC (X=0)
☒ Convection
h 60 W/m2.K
Tamb 320 C
☐ Fixed Temp
Ts C
☐ Heat FLux
q"s W/m2

Right BC (X=L)
☒ Convection
h 18 W/m2.K
Tamb 5 C
☐ Fixed Temp
Ts C
☐ Heat FLux
q"s W/m2

Thermal Contact Resistance
rmin Value 0.0000 C.m2/W
Apply

Layers
Thickness cm
k W/m.K

Layer Name	Therm. Cond.	Thickness
Layer_1	80	0.25
Layer_2	0.05	3

Results

qr 120.7861 W
R"tot 2.6079 C.m2/W
R"tot C.m2/W
U 0.3834 W/m2.K

Layers
X cm
T C
Rth C.m2/W

	r Left	T Left	Rth	r Right	T Right
	2.500	307.18	0.000	2.750	307.16
	2.750	307.16	2.348	5.750	23.57

Update

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

<<End-of-Tutorial>>

Example – Composite Sphere:

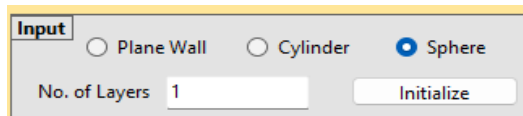
A stainless-steel spherical tank ($k=15 \text{ W/m.K}$), with a 3 m internal diameter and 2 cm thickness, is used to store ice water at 0°C . If the environment is at 22°C , and assuming $80 \text{ W/m}^2\text{C}$ and $16 \text{ W/m}^2\text{C}$ for internal and external heat transfer coefficients, determine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

Solution:

Open “Heat Conduction Composite Systems” Panel:

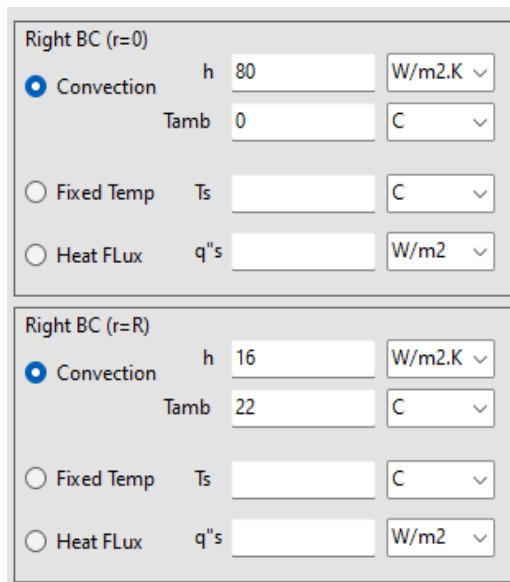
Top menu/Heat Transfer/Heat Conduction/Steady-State 1D/Tool: Composite Wall

Select *Sphere* and set number of layers to “1” and click *Initialize*.



Input panel showing the selection of **Sphere** and the number of layers set to **1**. The **Initialize** button is visible.

Set values for *Left BC* ($h=80 \text{ W/m}^2\text{.K}$, $T_{amb}=0^\circ\text{C}$), and *Right BC* ($h=16 \text{ W/m}^2\text{.K}$, $T_{amb}=22^\circ\text{C}$)



Two panels for setting boundary conditions. The top panel is for **Right BC (r=0)** and the bottom panel is for **Right BC (r=R)**. Both panels have **Convection** selected.

Parameter	Value	Unit
h	80	W/m2.K
Tamb	0	C

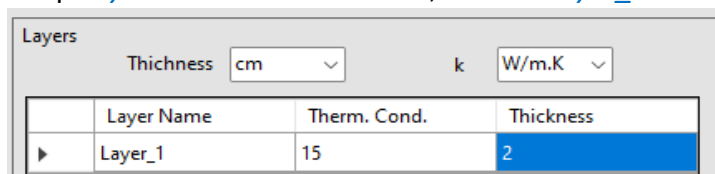
Parameter	Value	Unit
h	16	W/m2.K
Tamb	22	C

Set $r0=1.5 \text{ cm}$.



Input field for $r0$ set to **1.5** with unit **m**.

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



Layers panel showing the unit **cm** for thickness and **W/m.K** for thermal conductivity. A table lists the layer properties.

Layer Name	Therm. Cond.	Thickness
Layer_1	15	2

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

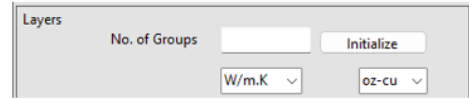
HT-02: K effective Parallel/Series

This tool 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 are 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.



Layers

No. of Groups Initialize

W/m.K oz-cu

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).

	Nu. of Layers	Therm. Cond.	Thickness
	2	390	2
▶	3	390	1

3. Background (Dielectric) Definition

- Enter thermal conductivity and total thickness for the background (board).

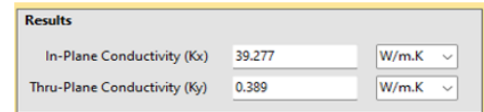


tb 0.096 in

Kb 0.35 W/m.K

Results:

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



Results

In-Plane Conductivity (Kx) 39.277 W/m.K

Thru-Plane Conductivity (Ky) 0.389 W/m.K

Example:

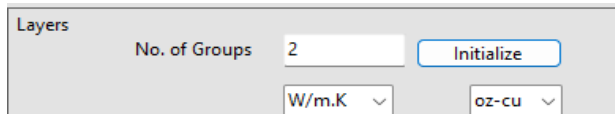
Consider a 0.096-inch-thick PCB, with two 2-oz-cu layers and three 1-oz-cu layers. Determine in-plane and thru-plane effective thermal conductivities for this board. Assume the board is made of FR4 ($k=0.35 \text{ W/m.K}$) and copper conductivity is 390 W/m.K .

Solution:

Open “Effective Thermal Conductivity” Panel:

Top menu/Heat Transfer/Heat Conduction/Steady-State 1D/Tool: K Effective Parallel/Series

Enter “2” for *No. of Groups* and click *Initialize*.



Layers

No. of Groups: 2

Initialize

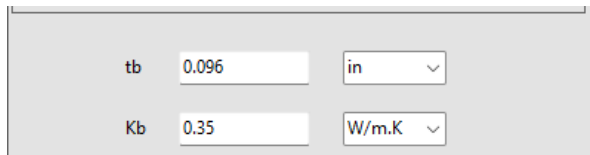
W/m.K

oz-cu

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

	Nu. of Layers	Therm. Cond.	Thickness
	2	390	2
▶	3	390	1

Enter values for board thickness and conductivity.



tb: 0.096 in

Kb: 0.35 W/m.K

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

Effective Thermal Conductivity

Input

Layers

No. of Groups

2

Initialize

W/m.K

oz-cu

	Nu. of Layers	Therm. Cond.	Thickness
	2	390	2
▶	3	390	1

tb

0.096

in

Kb

0.35

W/m.K

Results

In-Plane Conductivity (Kx)

39.277

W/m.K

Thru-Plane Conductivity (Ky)

0.389

W/m.K

Update

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:

Top menu/Heat Transfer/Heat Conduction/Steady-State 1D/Tool: K Effective Vias

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

Effective Conductivity - Vias

Input

☐ Number of Vias 160

☒ Via Spacing 1 mm

Vias Inner Diameter 0.3 mm

Vias Wall Thickness 1 oz-cu

Vias Conductivity (Kv) 395 W/m.K

Filler Conductivity (Kf) 0.03 W/m.K

Back Conductivity (Kb) 0.35 W/m.K

L 0.65 in

W 0.4 in

Results

Keff 14.107 W/m.K

Update

Diagram labels:

- k_v, D_v - Via
- k_f, D_f - Filler
- K_b - Background

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).

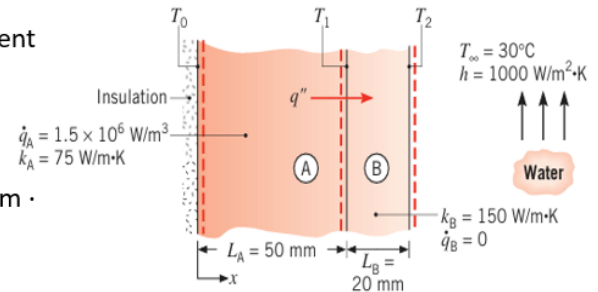
<<End-of-Tutorial>>

HT-04: Conduction with Uniform Heat Generation (Walls/Cylinders/Spheres)

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

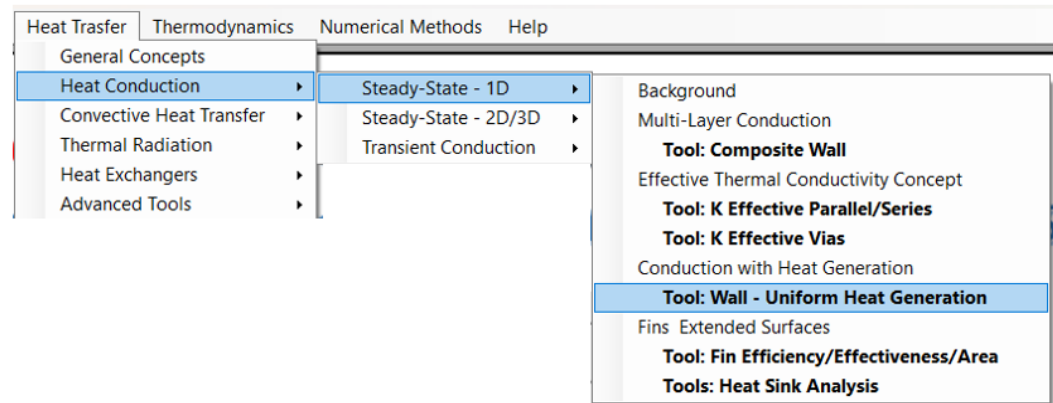
A plane wall of material A has uniform heat generation 1.50 MW/m^3 , $k_A = 75 \text{ W/m} \cdot \text{K}$, and thickness $L_A = 50 \text{ mm}$. The outer surface of the wall (larger x value) is cooled by a water stream with a temperature of 30°C and heat transfer coefficient of $1000 \text{ W/m}^2\cdot\text{K}$. Determine the inner wall temperature for the following three scenarios.

- The inner surface is exposed to air at 30°C with heat transfer coefficient of $250 \text{ W/m}^2\cdot\text{K}$
- The Inner surface is insulated.
- The inner surface is insulated, and the outer surface is attached to another wall with material B that has no generation with $k_B = 150 \text{ W/m} \cdot \text{K}$ and thickness $L_B = 20 \text{ mm}$.



Solution:

- Open "Solids with Uniform Heat Generation" Panel:



Enter input data

Geometric Parameters:

Geometry: Plane Wall
Heat Generation: Note can select 1.5 MW/m^3 , or $1.50\text{E}+06 \text{ W/m}^3$
Wall Thickness and Area: Note half thickness must be entered.
Surface area 1.0 m^2 is assumed since the value is not given.

The input form for the 'Plane Wall' geometry shows the following parameters: Heat Generation (q''') = 1.5 MW/m³, Thermal Conductivity (K) = 75 W/m.K, L = 25 mm, and A_s = 1 m².

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.

The 'Left Side Boundary Condition' form shows 'Prescribed Transport Coefficient and Ambient Temperature at $x=-L$ ' selected. The convection coefficient U_1 is 250 W/m².K and the ambient temperature T_{amb1} is 30 C.

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.

Right Side Boundary Condition

☐ Uniform Heat Flux at $x=L$

T_{s2} 115 C

☒ Prescribed Transport Coefficient and Ambient Temperature at $x=L$

$U2$ 1000 W/m².K

T_{amb2} 30 C

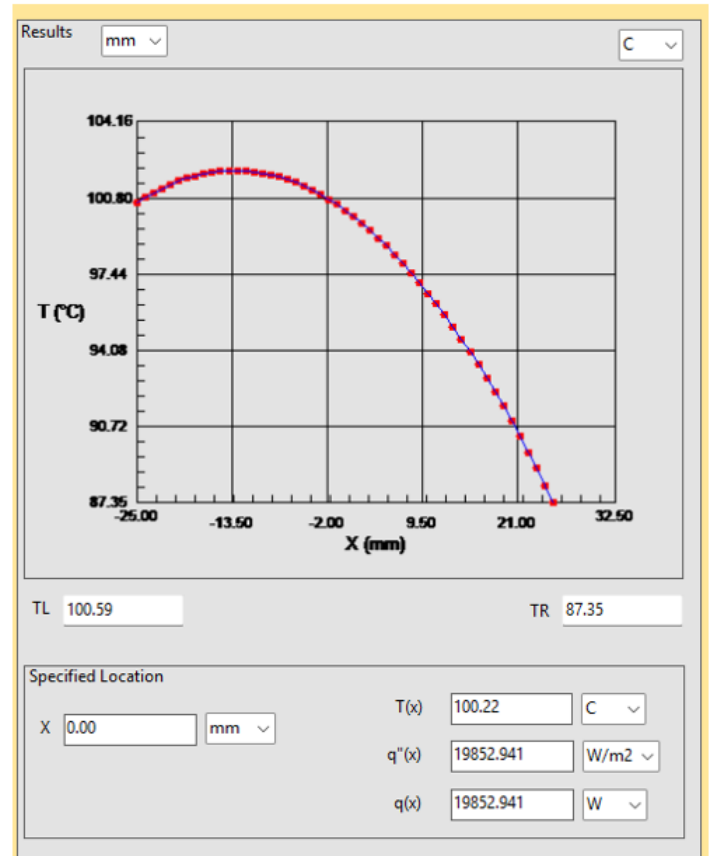
☐ Uniform Heat Flux at $x=L$

$q''s$ W/m²

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 m².



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 heat transfer coefficient $U1$.

Left Side Boundary Condition

☐ Uniform Heat Flux at $x=-L$

T_{s1} C

☒ Prescribed Transport Coefficient and Ambient Temperature at $x=-L$

$U1$ 0 W/m².K

T_{amb1} 30 C

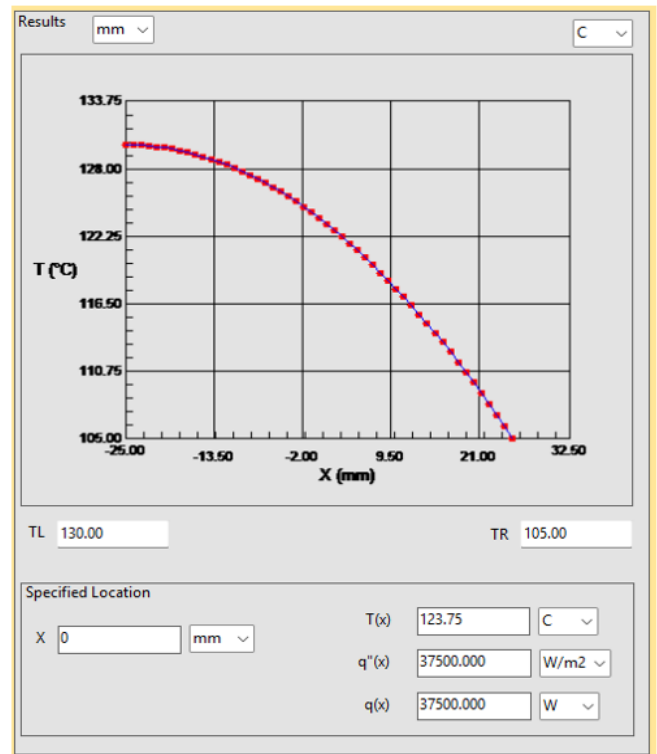
☐ Uniform Heat Flux at $x=-L$

$q''s$ W/m²

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 $k_B = 150 \text{ W/m} \cdot \text{K}$ and thickness $L_B = 20 \text{ mm}$.

The Only change to be made is to the right-side Boundary by adjusting U_2 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.

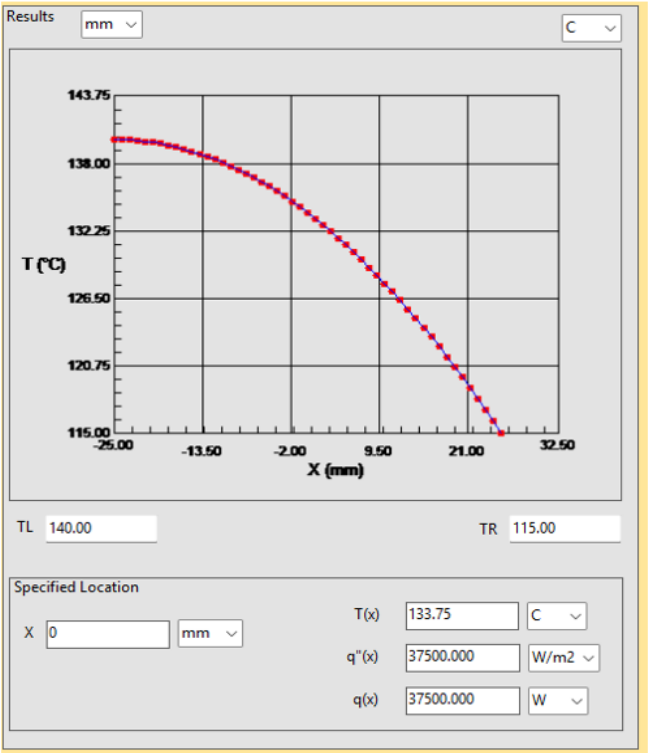
$$R''_{th,2} = \frac{L_B}{k_B} + \frac{1}{h_2} = \frac{0.02\text{m}}{150 \text{ W/(m} \cdot \text{K)}} + \frac{1}{1000 \text{ W/(m}^2 \cdot \text{K)}} = 0.00113333 \text{ K} \cdot \text{m}^2 / \text{W}$$

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

Results

Click Update to solve.

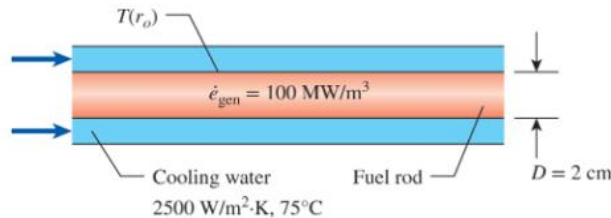
Plot of temperature distribution through the slab indicates insulated BC on the left face and updated temperature and heat flux values.



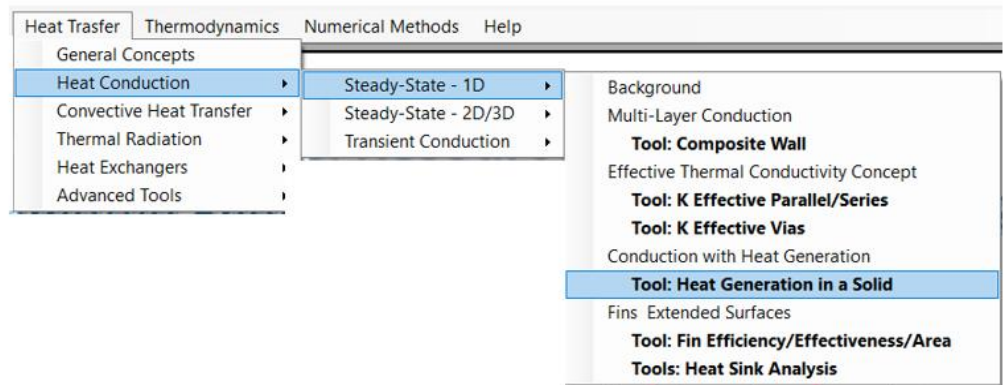
Example: Water-Cooled Fuel Rod [Source: Cengel-Ghajar Problem 2.104]

A cylindrical fuel rod ($k = 30 \text{ W/m}\cdot\text{K}$) with a diameter of 2 cm is encased in a concentric tube and cooled with water. The fuel rod generates heat uniformly at a rate of 100 MW/m^3 , and the average temperature of the cooling water is 75°C with a convection heat transfer coefficient of $2500 \text{ W/m}^2\cdot\text{K}$. The operating pressure of the cooling water is set so that the fuel rod surface temperature remains 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, causing overheating of the fuel rod surface and leading to damage. Determine the temperature variation in the fuel rod and at its 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 ($r=r_1$) Boundary Conditions:

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

Left Side Boundary Condition

☒ Uniform Heat Flux at $r=r_1$

T_{s1} C

☐ Prescribed Transport Coefficient and Ambient Temperature at $r=r_1$

$U1$ W/m².K

T_{amb1} C

☐ Uniform Heat Flux at $r=r_1$

$q''s$ W/m²

Right side ($r=r_2$) 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.

Right Side Boundary Condition

☐ Specified Temperature at $r=r_2$

T_{s2} C

☒ Prescribed Transport Coefficient and Ambient Temperature at $r=r_2$

$U2$ W/m².K

T_{amb2} C

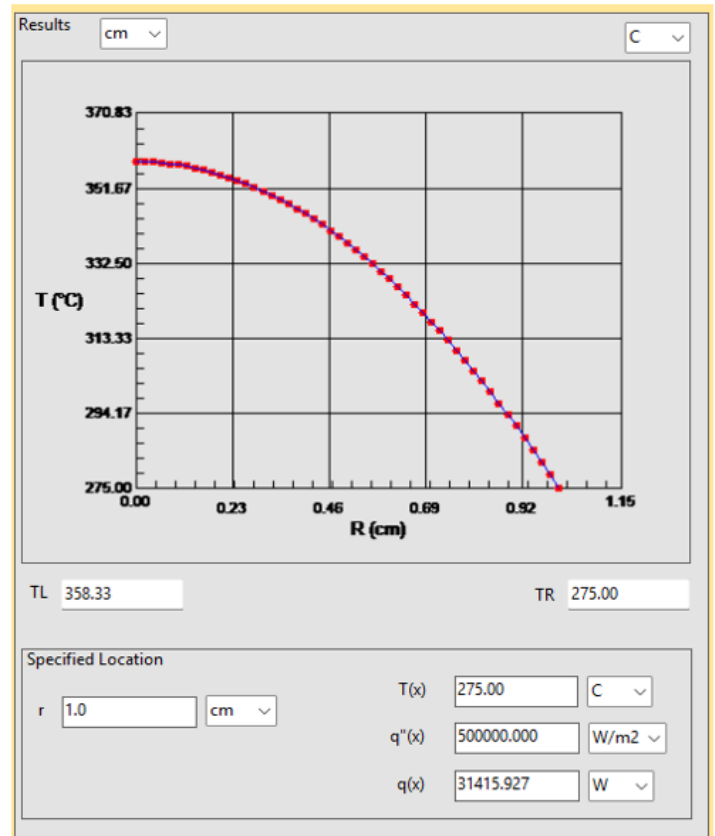
☐ Uniform Heat Flux at $r=r_2$

$q''s$ W/m²

Results

Click Update to solve.

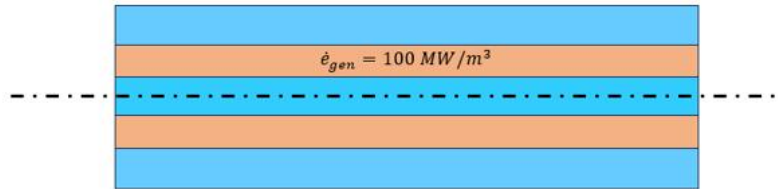
1. Plot of temperature distribution through the slab is provided.
2. Left-side (centerline) and Right-Side ($r=1$ cm) temperatures are shown.
3. The user can select the units for temperature and length for the plot are from unit boxes at the top.
4. The surface temperature of 275 °C is 75°C higher than 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.



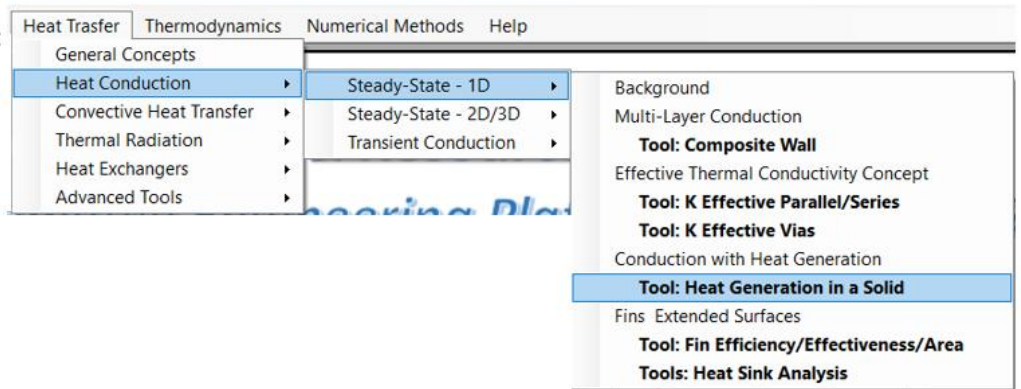
Example: Water-Cooled Fuel Rod [Source: Cengel-Ghagar Problem 2.104 - Modified]

Repeat the above example with an added inner cooling channel of 2 cm, cooling water at 75 °C, with a convection heat transfer coefficient of 2500 W/m².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.

$$\pi r_2^2 L = \pi(r_2^2 - r_1^2)L \rightarrow r_2^2 = r^2 + r_1^2 \rightarrow r_2 = \sqrt{r^2 + r_1^2}$$
$$r_1 = 1 \text{ cm}; r = 1 \text{ cm} \rightarrow r_2 = \sqrt{2} = 1.41421356 \text{ cm}$$

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; r₂ = √2 = 1.41421356 cm

Left side ($r=r_1$) 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.

Left Side Boundary Condition

☐ Specified Temperature at $r=r_1$

T_{s1} C

☒ Prescribed Transport Coefficient and Ambient Temperature at $r=r_1$

$U1$ W/m².K

T_{amb1} C

☐ Uniform Heat Flux at $r=r_1$

q''_s W/m²

Right side ($r=r_2$) 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.

Right Side Boundary Condition

☐ Specified Temperature at $r=r_2$

T_{s2} C

☒ Prescribed Transport Coefficient and Ambient Temperature at $r=r_2$

$U2$ W/m².K

T_{amb2} C

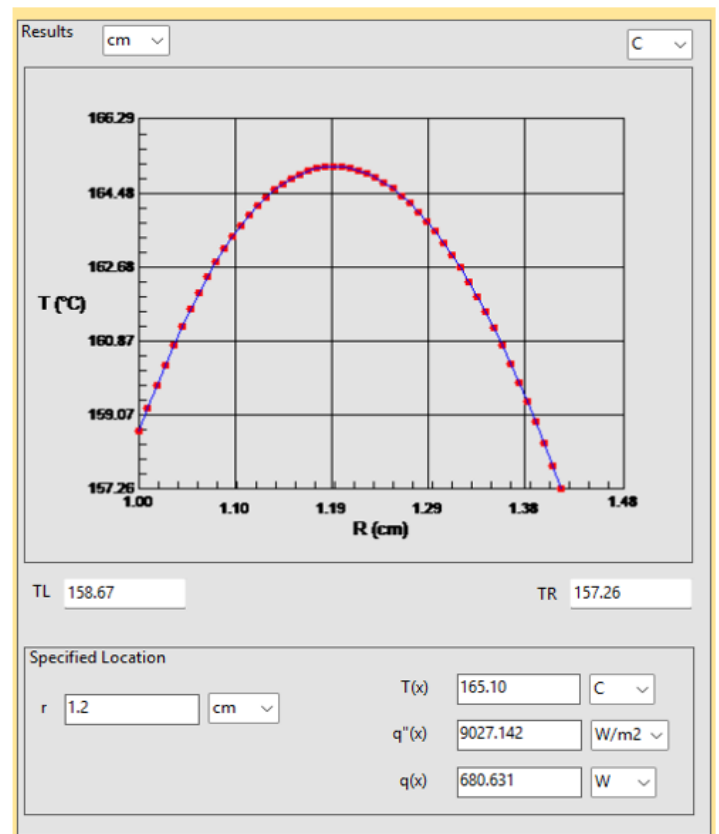
☐ Uniform Heat Flux at $r=r_2$

q''_s W/m²

Results

Click Update to solve.

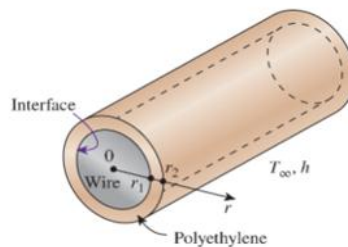
1. Plot of temperature distribution through the slab is provided.
2. Left-side (centerline) and Right-Side ($r=1$ cm) temperatures are shown.
3. The user can select the units for temperature and length for the plot are from unit boxes at the top.
4. The surface temperature of 158.57 °C and 157.26 °C 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.



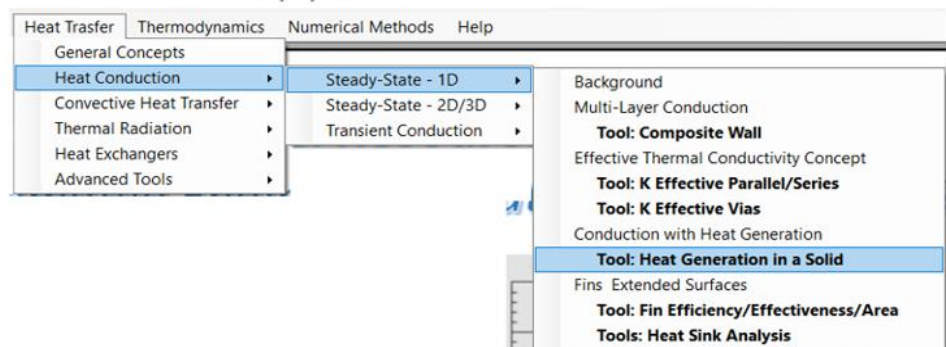
Example: Electrical Resistance wire with Insulation [Source: Cengel-Ghagar Problem 2.105 - Modified]

A long electrical resistance wire of radius $r_1 = 0.2$ 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/cm³. 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/m².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 75 °C. Under these conditions, does the polyethylene insulation for the wire meet the ASTM D1351 standard?

Solution:

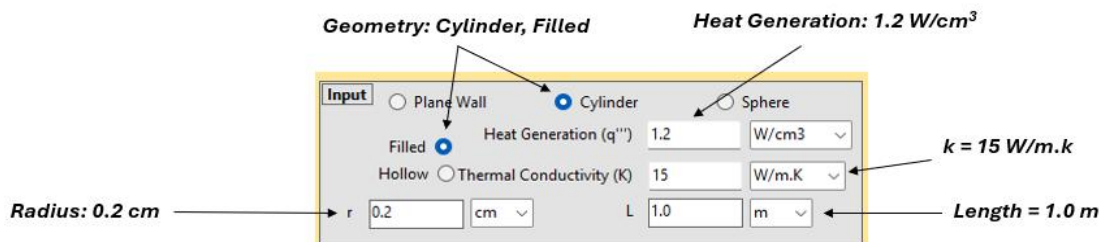


Open "Heat Generation in a Solid" Panel:



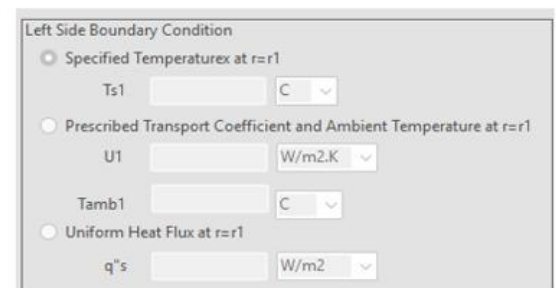
Enter input data

Geometric Parameters:



Left side ($r=r_1$) 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 U_2 :

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

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/m².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/m².K** is the value we need for the analysis; it is the U value using the radius of 0.2 cm.

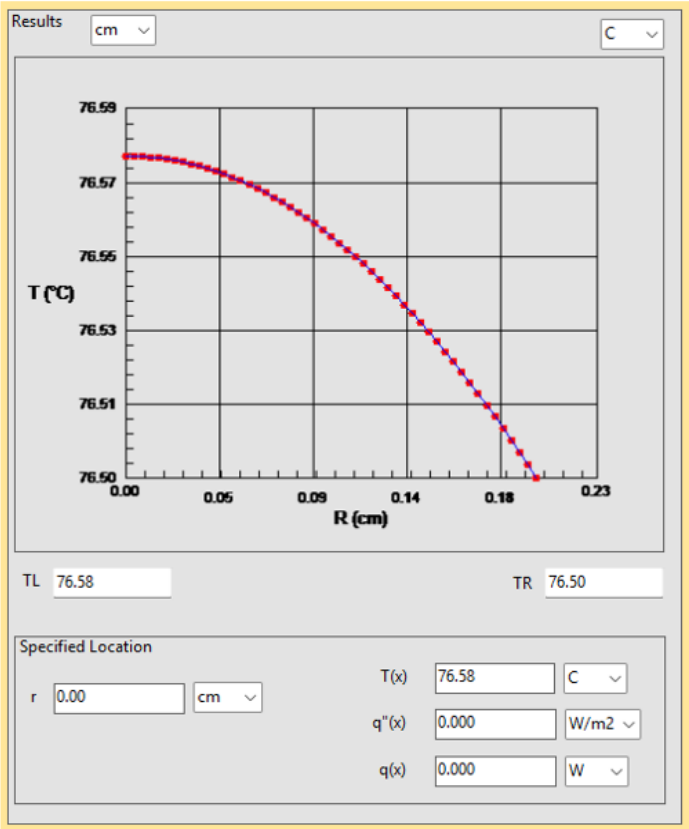
Now, we get back to "Heat Generation in a Solid" and enter the calculated value of U_2 in the Right-Side BC Panel.

- 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.

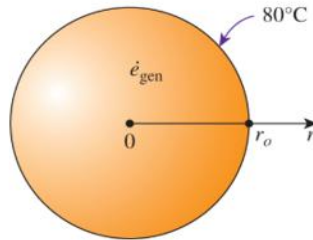
- 1. Plot of temperature distribution through the slab is provided.
- 2. Left-side (centerline) and Right-Side ($r=1\text{ cm}$) temperatures are shown.
- 3. The user can select the units for temperature and length for the plot are from unit boxes at the top.
- 4. The surface temperature of $76.5\text{ }^{\circ}\text{C}$ is $1.5\text{ }^{\circ}\text{C}$ 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 $0.1\text{ }^{\circ}\text{C}$ higher than the interface temperature.



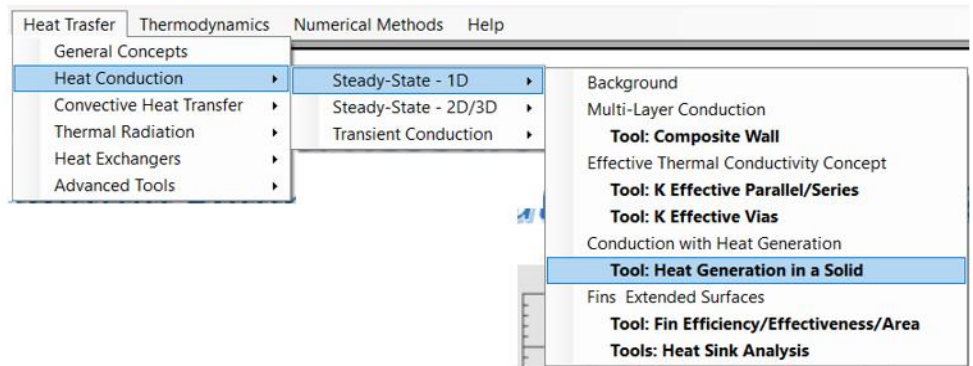
Example:

Consider a homogeneous spherical piece of radioactive material of radius $r_o = 0.04$ m that is generating heat at a constant rate of $e_{\text{gen}} = 4 \times 10^7$ W/m³. The heat generated is dissipated to the environment steadily. The outer surface of the sphere is maintained at a uniform temperature of 80°C, 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:



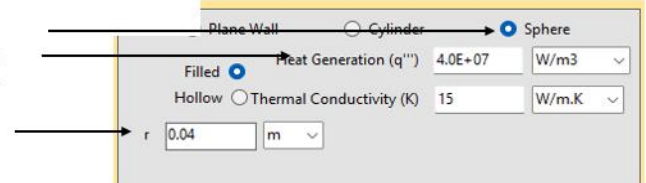
Open "Heat Generation in a Solid" Panel:



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 ($r=r_1$) Boundary Conditions:

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

Left Side Boundary Condition

☒ Specified Temperature at $r=r_1$
Ts1 C

☐ Prescribed Transport Coefficient and Ambient Temperature at $r=r_1$
U1 W/m2.K
Tamb1 C

☐ Uniform Heat Flux at $r=r_1$
q"s W/m2

Right side ($r=r_2$) Boundary Conditions:

Select Specified Temperature.

Enter 80 °C.

Right Side Boundary Condition

☒ Specified Temperature at $r=r_2$
Ts2 80 C

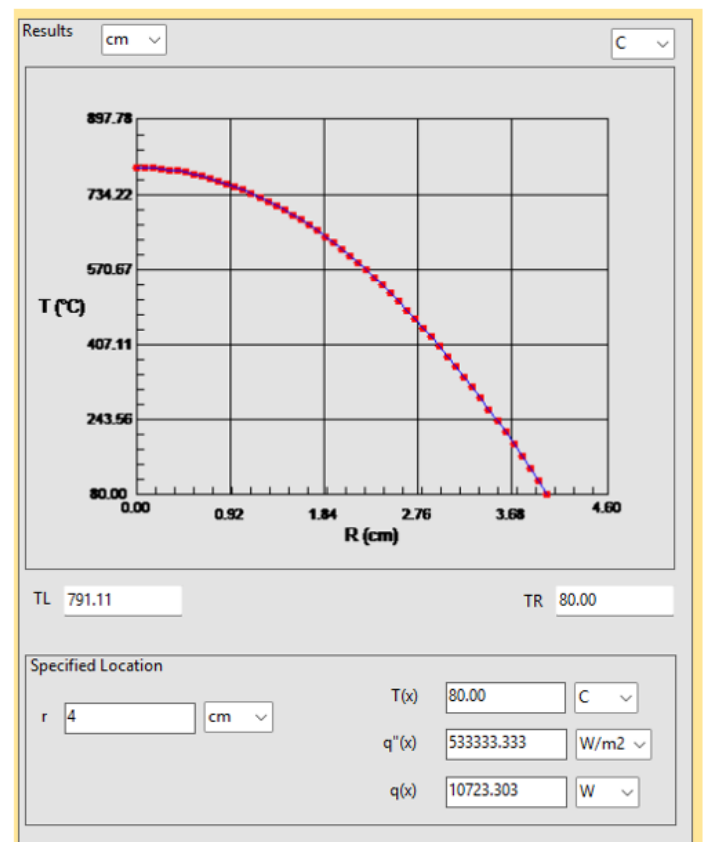
☐ Prescribed Transport Coefficient and Ambient Temperature at $r=r_2$
U2 W/m2.K
Tamb2 C

☐ Uniform Heat Flux at $r=r_2$
q"s W/m2

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 ($k = 186 \text{ W/m} \cdot \text{K}$) 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 $20 \text{ W/m}^2 \cdot \text{K}$, determine the efficiency and effectiveness for each fin.

Solution:

Open “Fin Efficiency” Panel:

Top menu/Heat Transfer/Heat Conduction/Steady-State 1D/Tool: Fin EfficiencyEffectiveness/Area

Choose **f) Cone with blunt tip** from the fin type combo list

Enter the following parameters:

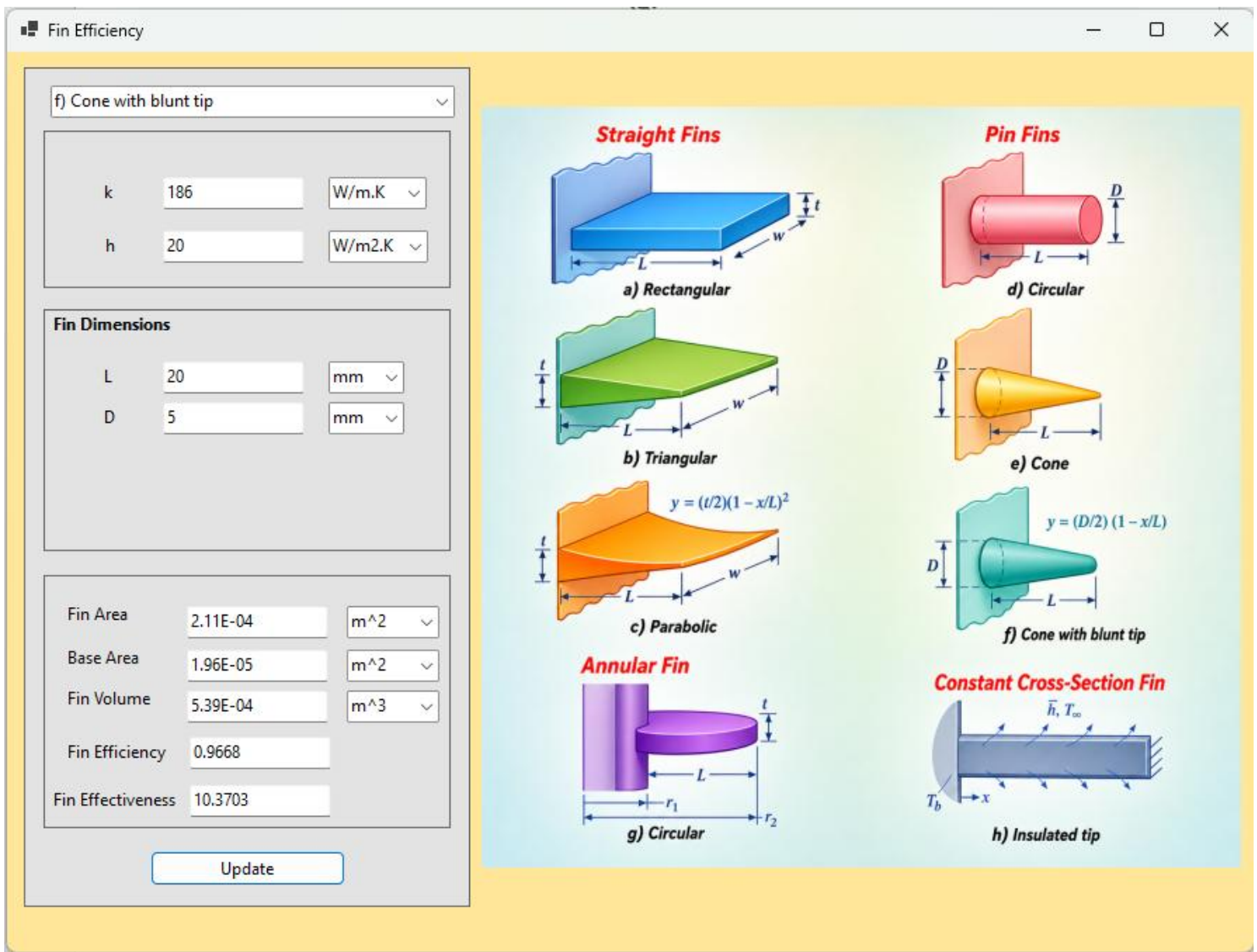
$$k = 186 \text{ W/m.K}$$

$$h = 20 \text{ W/m}^2 \cdot \text{K}$$

$$L = 20 \text{ mm}$$

$$D = 5 \text{ mm}$$

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.

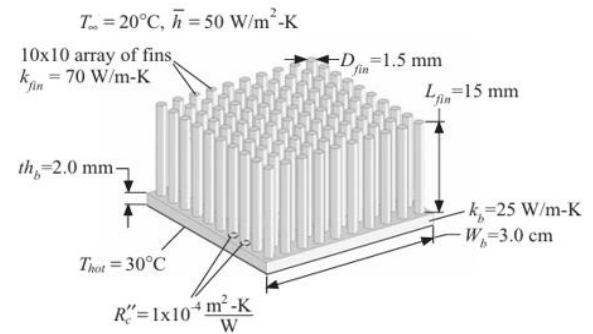
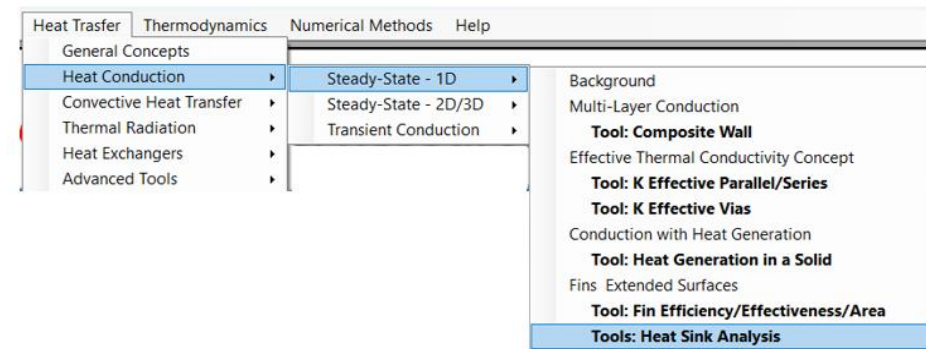
HT-06: Heat Sink Analysis

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

A thermoelectric cooling device rejects heat using a 10×10 array of 1.5-mm-diameter pin fins, each 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 fin material has a conductivity of 70 W/m-K, and the base material has a thermal conductivity of 25 W/m-K. There is a contact resistance of $1 \times 10^{-4} \text{ m}^2\text{-K/W}$ 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 heat sink surface is $h = 50 \text{ W/m}^2 \cdot \text{K}$.

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

Solution:



Enter input data

1-Heat Sink Input

Heat Sink Input

☒ Rectangular Heat Sink ☐ Circular Heat Sink

Base Plate Length (L) 3 cm

Base Plate Width (w) 3 cm

Base Plate Thickness (t) 2 mm

Base Plate Conductivity (k_b) 25 W/m.K

Contact base-source ($R''_{c,b}$) 0.00 C.m2/W

Number of Fins (Nfin) 100

Contact Base-Fin ($R''_{c,fin}$) 0.0001 C.m2/W

Avg Heat Tran Coeff (h) 50 W/m2.K

Ambient Temp. (Tamb) 20 C

Fin Details

Fin surface area m^2

Fin base area m^2

Fin Efficiency

Use Fin Tool

2-Click "Use Fin Tool" to open the Fin Tool and get fin properties.

3-Choose circular fin and enter fin's thermal and geometric data. Click on Update..

d) Circular

k 70 W/m.K

h 50 W/m2.K

Fin Dimensions

L 15 mm

D 1.5 mm

Fin Area m^2

Base Area m^2

Fin Volume m^3

Fin Efficiency

Fin Effectiveness

Update

4-The Fin Tool will Automatically close, and the results will be entered into the Fin Details section.

Fin Details

Fin surface area 7.07E-05 m^2

Fin base area 1.77E-06 m^2

Fin Efficiency 0.8780

Use Fin Tool

frmHeatSink

Heat Sink Input

☒ Rectangular Heat Sink
☐ Circular Heat Sink

Base Plate Length (L) 3 cm

Base Plate Width (w) 3 cm

Base Plate Thickness (t) 2 mm

Base Plate Conductivity (kb) 25 W/m.K

Contact base-source (R"cb) 0.00 C.m2/W

Number of Fins (Nfin) 100

Contact Base-Fin (R"cf,fin) 0.0001 C.m2/W

Avg Heat Tran Coeff (h) 50 W/m2.K

Ambient Temp. (Tamb) 20 C

Fin Details

Fin surface area 7.07E-05 m^2

Fin base area 1.77E-06 m^2

Fin Efficiency 0.8780

Use Fin Tool

$$R_{fin} = \frac{1}{\eta_{fin} h A_{s,fin}} \quad (\text{thermal resistance - single fin})$$

$$R_{unfinned} = \frac{1}{h (A_b - N_{fin} A_{c,fin})} \quad (\text{thermal resistance - unfinned section})$$

$$R_{cond,b} = \frac{t_b}{k_b A_b} \quad (\text{thermal resistance - baseplate})$$

$$R_{c,b}, R_{c,fin} \quad (\text{contact resistances - baseplate, fin})$$

$$A_{s,fin} \equiv \text{Fin total surface area}$$

$$A_{c,fin} \equiv \text{Fin cross-section area}$$

$$A_b \equiv \text{Baseplate Area}$$

Results

Total Thermal Resistance (Rtot) 3.41980 C.m2/W

☒ Source Temperature (Thot) 30 C

Overall Surface Efficiency (eta_o) 0.889

☐ Heat Transfer Rate (Qs) 2.924 W

Update

Show Tutorial

5-Enter 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.

 AutoTherm

ETR, LLC. Proprietary and Confidential. All rights reserved.

Example – Motorcycle Cylinder [Bergman & Levin Example 3.10]

The engine cylinder of a motorcycle is constructed of 2024-T6 aluminum alloy and is of height $H = 100$ mm and outer diameter $D = 2r_1 = 50$ mm. Under typical operating conditions, $q_t = 2$ kW of heat is transferred from the cylinder to ambient air at 300 K, with a convection coefficient of $75 \text{ W/m}^2 \cdot \text{K}$. Annular fins are integrally cast with the cylinder to reduce the cylinder temperature. Consider ten equally spaced fins, each of thickness $t = 4$ mm and length $L = 20$ mm. What reduction in the cylinder temperature can be achieved by using the fins?

1. Choose the “Circular” option
2. Enter values for the heat sink dimensions and properties.
3. Click **Use n Tool**: This will open the Fin Efficiency tool to obtain Fin Details.
4. Choose “Circular Fin”
5. The fin conductivity and heat transfer coefficient will be filled out automatically – make sure they are correct.
6. Enter Fin Dimensions.
7. Click Update.
8. The Fin Tool will close automatically, and the required values will be entered back into the Heat Sink tool.

The image shows three panels from the AutoTherm software interface:

- Heat Sink Tool:**
 - Step 1: Select "Circular Heat Sink" (radio button).
 - Step 2: Enter dimensions and properties:
 - Base Cylinder Length (L): 100 mm
 - Cylinder Outer Radius (r1): 25 mm
 - Cylinder wall Thickness (t): 2 mm
 - Base conductivity (kb): 186 W/m.K
 - Contact base-source ($R''_{c,b}$): 0.00 C.m2/W
 - Number of Fins (Nfin): 10
 - Contact Base-Fin ($R''_{c,fin}$): 0.00 C.m2/W
 - Avg Heat Tran Coeff (h): 75 W/m2.K
 - Ambient Temp. (Tamb): 300 K
 - Fin Details (pre-filled):
 - Fin surface area: 9.95E-03 m^2
 - Fin base area: 6.28E-04 m^2
 - Fin Efficiency: 0.9608
 - Step 3: Click "Use Fin Tool".
- Fin Tool:**
 - Step 4: Select "Circular" (dropdown).
 - Step 5: Confirm material properties:
 - k: 186 W/m.K
 - h: 75 W/m2.K
 - Step 6: Enter Fin Dimensions:
 - L: 20 mm
 - t: 4 mm
 - r1: 25 mm
 - r2: 45 mm
 - Step 7: Click "Update".
- Back to Heat Sink Tool:**
 - Shows the updated Fin Details:
 - Fin surface area: 9.95E-03 m^2
 - Fin base area: 6.28E-04 m^2
 - Fin Efficiency: 0.9608
 - Click "Use Fin Tool" to return to the Heat Sink Tool.

Click “ON” the option button for the “Heat Transfer Rate” in the Results panel and enter 2000 W for the value.

Click Update to solve. The base temperature is calculated to be 282 °C.

Heat Sink Analysis

Heat Sink Input

Rectangular Heat Sink

Circular Heat Sink

Base Cylinder Length (L)

100

mm

Cylinder Outer Raduis (r1)

25

mm

Cylinder wall Thickness (tb)

2

mm

Base onductivity (kb)

186

W/m.K

Contact base-source (R" c,b)

0.00

C.m2/W

Number of Fins (Nfin)

10

Contact Base-Fin (R" c,fin)

0.00

C.m2/W

Avg Heat Tran Coeff (h)

75

W/m2.K

Ambient Temp. (Tamb)

300

K

Fin Details

Fin surface area

9.95E-03

m^2

Fin base area

6.28E-04

m^2

Fin Efficiency

0.9608

Use Fin Tool

Results

Total Thermal Resistance (Rtot)

0.12766

C.m2/W

Overall Surface Efficiency (ηp)

0.964

Source Temperature (Thot)

282.18

C

Heat Transfer Rate (Qs)

2000

W

Update

ENHANCED FINNED HEAT SINK

Thermal Resistance Model

RECTANGULAR FIN

PIN FIN ARRAY

PIN FIN HEAT SINK

PIN FIN GEOMETRY

THERMAL RESISTANCE MODEL

Diagram illustrating the thermal resistance model for a pin fin heat sink. The model shows heat flow from a hot source (T_{hot}) through a contact resistance ($R_{c,b}$), a conduction resistance in the base plate ($R_{cond,b}$), and then through the thermal resistance of the fins ($R_{c,fin}$ and R_{fin}) to the ambient temperature (T_{∞}). The thermal resistance of the fins is represented by a parallel combination of the fin thermal resistance (R_{fin}) and the thermal resistance of the unfinned section ($R_{unfinned}$).

HEAT CONDUCTION

FIN RESISTANCE

CONVECTION TO AMBIENT

BASEPLATE EFFECTS

$$R_{fin} = \frac{1}{\eta_{fin} h A_{s,fin}} \quad (\text{thermal resistance - single fin})$$

$$R_{unfinned} = \frac{1}{\bar{h} (A_b - N_{fin} A_{c,fin})} \quad (\text{thermal resistance - unfinned section})$$

$$R_{cond,b} = \frac{t_b}{k_b A_b} \quad (\text{thermal resistance - baseplate})$$

$$R_{c,b}, R_{c,fin} \quad (\text{contact resistances - baseplate, fin})$$

$$A_{s,fin} = \text{Fin total surface area}$$

$$A_{c,fin} = \text{Fin cross-section area}$$

$$A_b = \text{Baseplate Area}$$

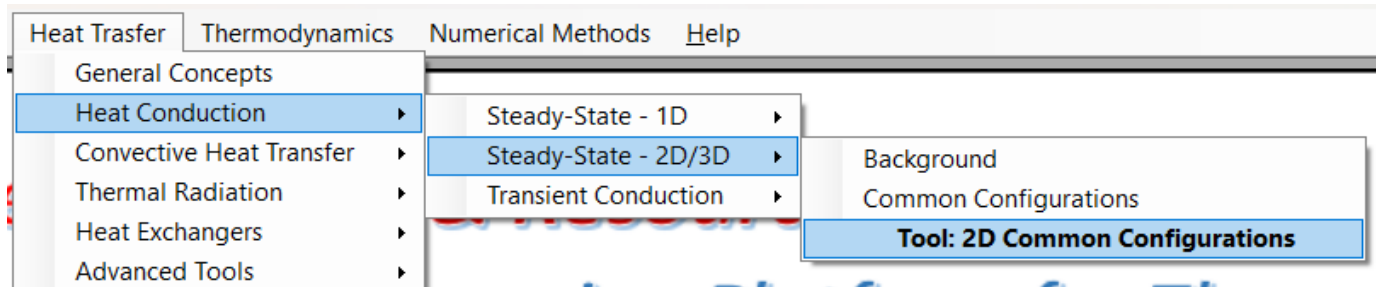
$$T_{b, \text{wo}} = T_{\infty} + \frac{q_t}{h(2\pi r_j H)} = 27^{\circ}\text{C} + \frac{2000 \text{ W}}{75 \text{ W/m}^2 \cdot \text{K}(2\pi \times 0.025 \text{ m} \times 0.10 \text{ m})} = 1725^{\circ}\text{C}$$
$$\Delta T_b = T_{b,wo} - T_b = 1725^{\circ}\text{C} - 282^{\circ}\text{C} = 1443^{\circ}\text{C}$$

HT-07: Multidimensional Heat Transfer in Common Configurations

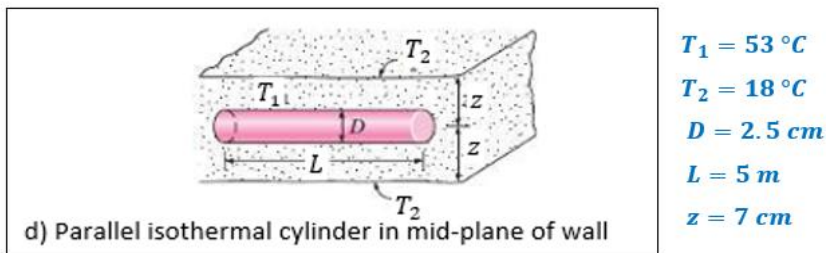
Example: Hot water at $53\text{ }^{\circ}\text{C}$ flows through a 5-m-long section of a thin-walled pipe at an average velocity of 5 m/s. The pipe passes through the center of a 14-cm-thick wall filled with fiberglass insulation ($k = 0.035\text{ W/m}\cdot\text{K}$). Assuming the surfaces of the wall are at $18\text{ }^{\circ}\text{C}$, 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.

Solution:

Open “Common Configuration/Shape Factor” Panel:



Choose configuration **d)** from the Shape Factors combo list



Enter input in the given units:

Dimensions		
D	2.5	cm
L	5	m
z	7	cm

Thermal Input		
k	0.035	W/m.K
T1	53	C
T2	18	C

a) Click [Update](#) to get the rate of heat transfer from the pipe to the air in the rooms.

Results		
S	15.993	m ▾
Rth	1.786	K/W ▾
Q12	19.592	W ▾
Update		

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:

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

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

<<End-of-Tutorial>>

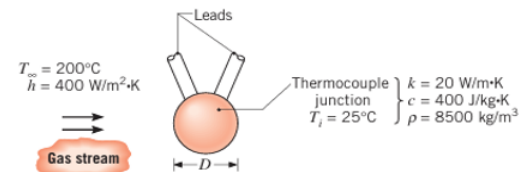
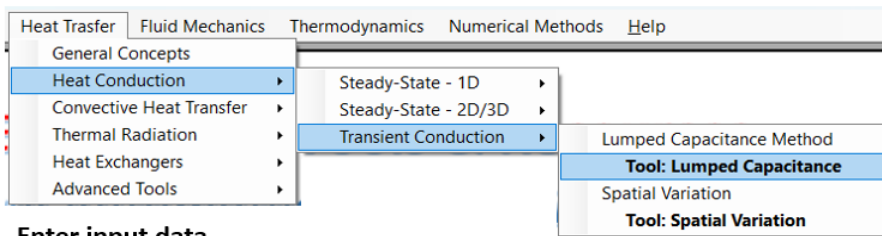
HT-08: Lumped Capacity (Constant Ambient Temperature/ Varying Ambient Temperature)

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

A thermocouple junction, which may be approximated as a sphere, is used to measure temperature in a gas stream. The convection coefficient between the junction surface and the gas is $h = 400 \text{ W/m}^2\cdot\text{K}$, and the junction thermophysical properties are $k = 20 \text{ W/m}\cdot\text{K}$, $c = 400 \text{ J/kg}\cdot\text{K}$, and $\rho = 8500 \text{ kg/m}^3$. Assume the thermocouple junction diameter is 0.8 mm , and the initial temperature is $T_i = 25^\circ\text{C}$.

- Temperature versus time for eight seconds.
- Determine the junction temperature at time $t=1 \text{ s}$.
- Determine how long it will take for the junction to reach 198°C .

Solution:



Enter input data

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

Geometric Parameters

Choose 3D Shape

Area m^2

Volume m^3

Input

f-Solid Sphere

0.4 mm

Geometric Parameters

Choose 3D Shape

Area 2.0106e-06 m^2

Volume 2.6808e-10 m^3

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

- Enter solid thermal properties.

Material

Library Aluminum_Pure

User

ρ 8500 kg/m^3

c 400 $\text{J/kg}\cdot\text{K}$

k 20 $\text{W/m}\cdot\text{K}$

- In "Ambient Conditions" section:
 - Enter values for heat transfer coefficient and initial temperature.
 - Keep Constant T_{inf} option "on" and enter the ambient temperature

Ambient Conditions

Heat Trans. Coeff. 400 $\text{W/m}^2\cdot\text{K}$

Initial Temp 25 $^\circ\text{C}$

☒ Constant T_{inf} 200 $^\circ\text{C}$

☐ Time Dependent with Heat Generation

- In "Plot Temp vs Time" section:
 - Enter 8 second for "End Time" and click on Update.

Plot Temp vs time

End Time 8 s

Update

Part b):

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

Temperature at Specified Time

Time
1
s

Temperature
C

Update

Part c):

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

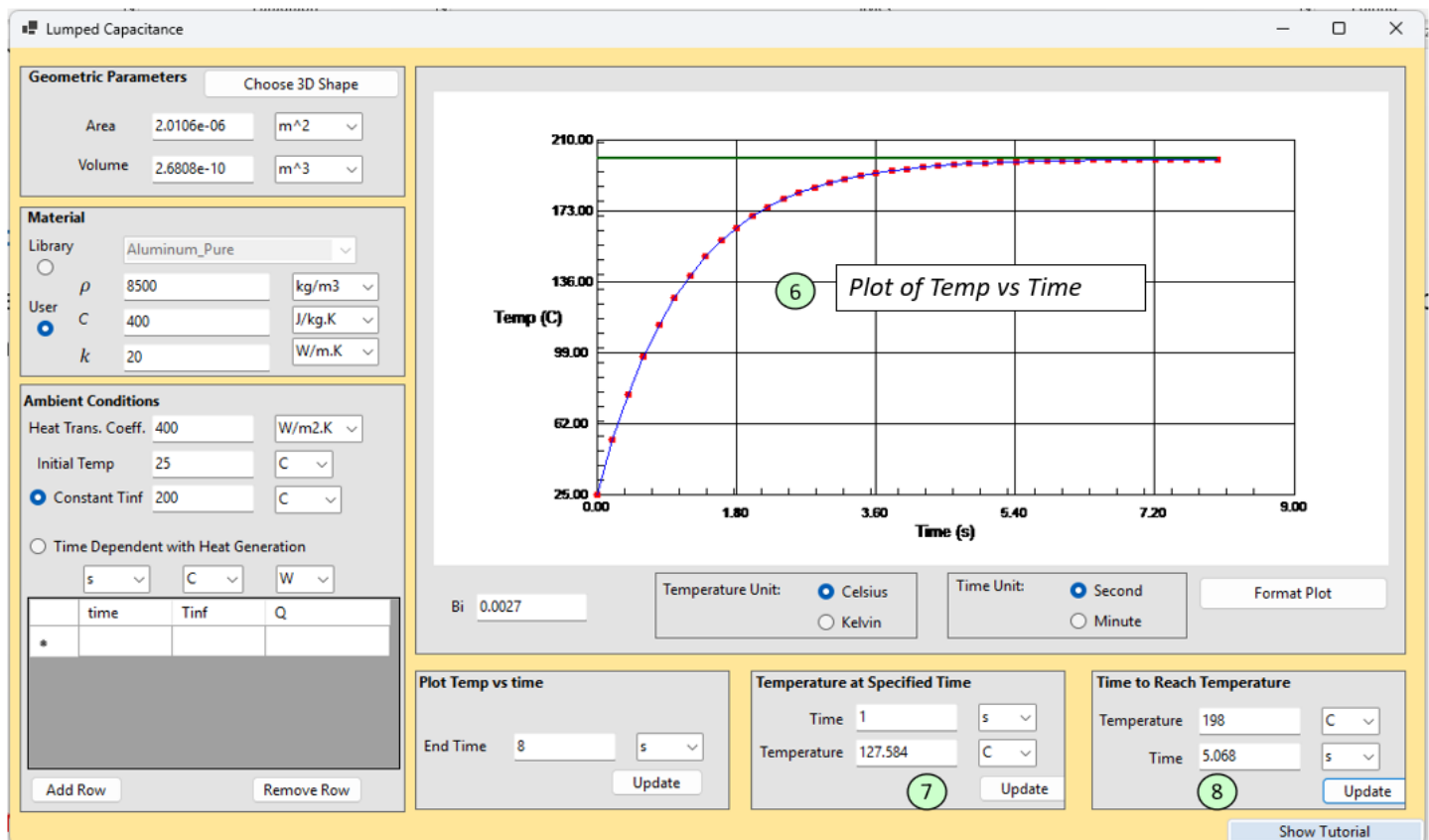
Time to Reach Temperature

Temperature
198
C

Time
s

Update

Finished form is shown below.

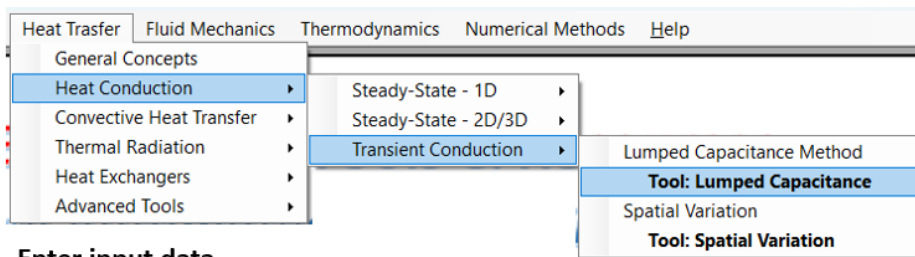


Note: the value of $Bi = 0.0027 \ll 0.1$, indicates lumped capacitance assumption is valid.

Example – Thermal Response of a Solid Ball in a Thermal Chamber

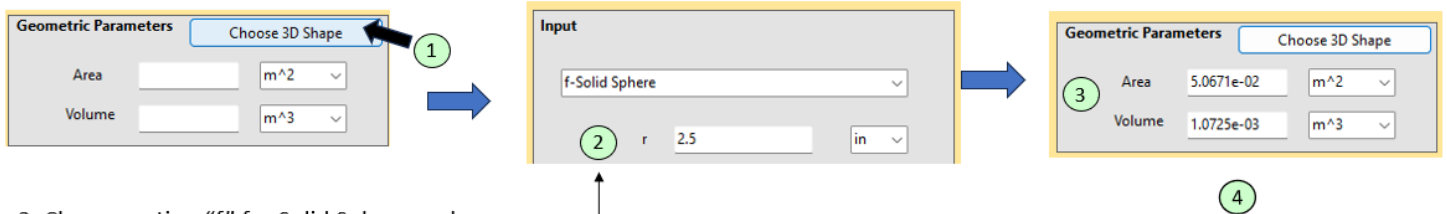
Consider a 5-inch solid metallic ball made of Aluminum_6061-T6, initially at 85 °C, in a thermal chamber with a heat transfer coefficient $h = 250 \text{ W/m} \cdot \text{K}$. Plot the sphere temperature versus time with the varying chamber temperature profile shown below.

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



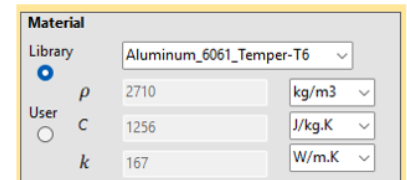
Enter input data

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



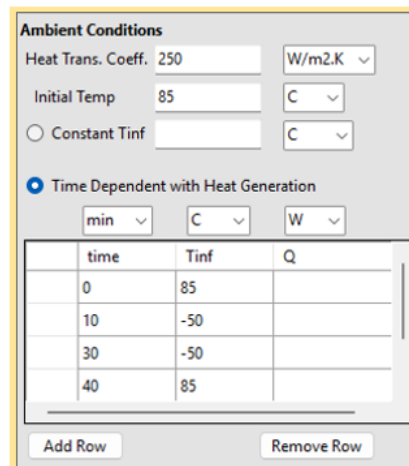
2. Choose option “f” for Solid Sphere and enter the radius.
3. Click update to automatically enter these parameters into “Lumped Capacity Form”.

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:

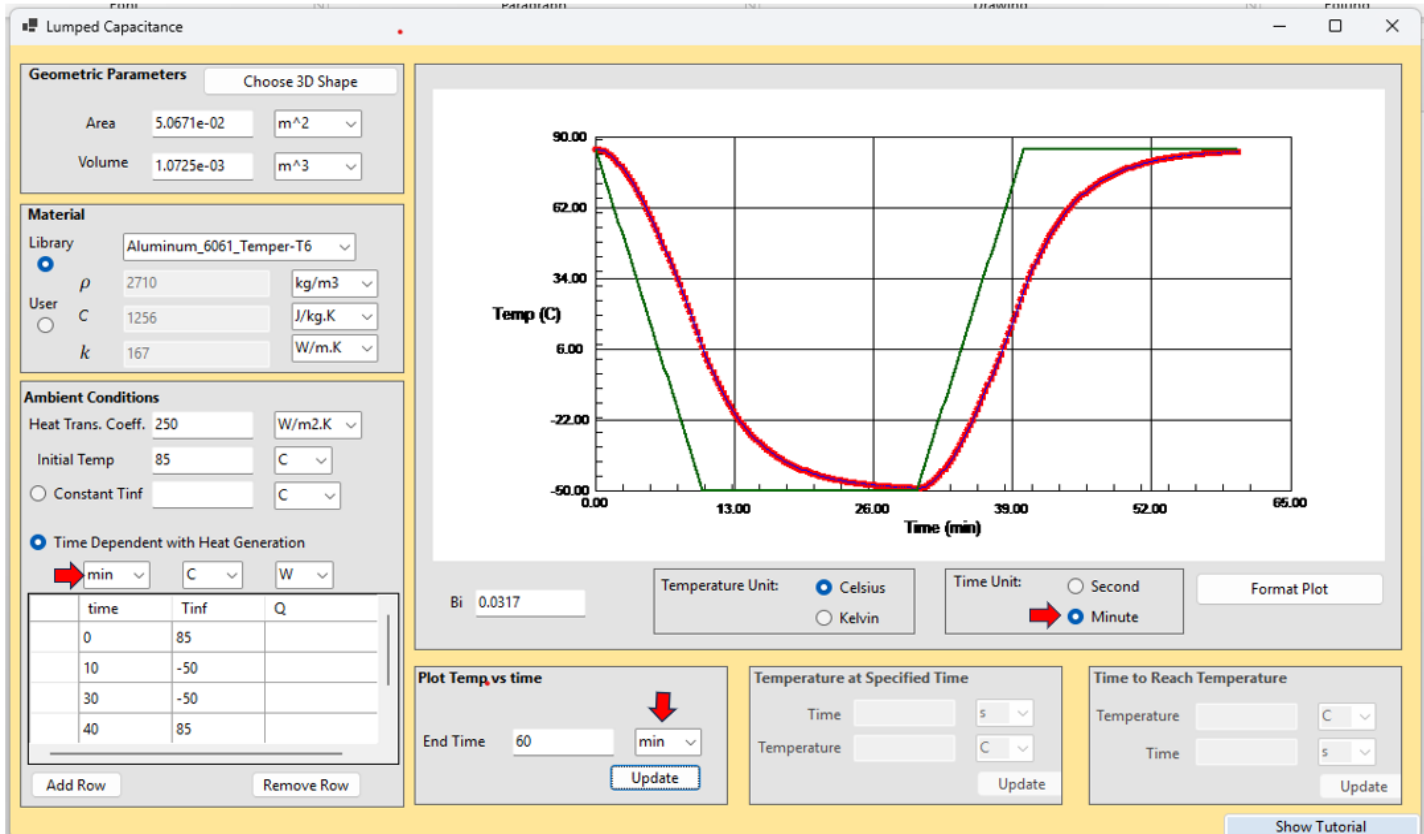
- Enter 60 minutes for "End Time"
- Change Time unit in the plot area to minutes
- click on Update.

Plot Temp vs time

End Time
60
min

Update

Finished form is shown below.



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

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

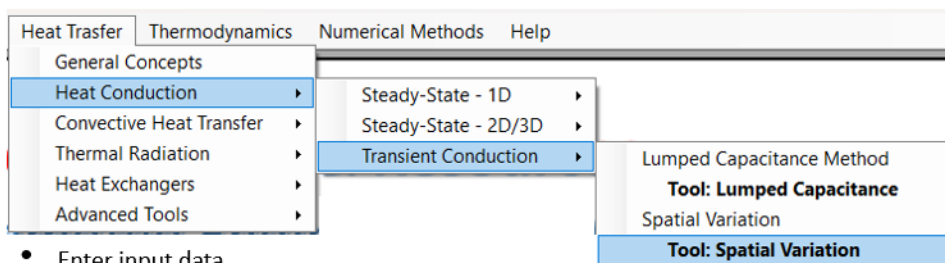
Example – Plate: Heating of a Steel Pipeline [Source: Bergman-Lavine Example 7.7]

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 initially has a uniform temperature of -20°C . When flow begins, hot oil at 60°C is pumped through the pipe, creating convection with $h = 500 \text{ W/m}^2 \cdot \text{K}$ at the inner surface of the pipe.

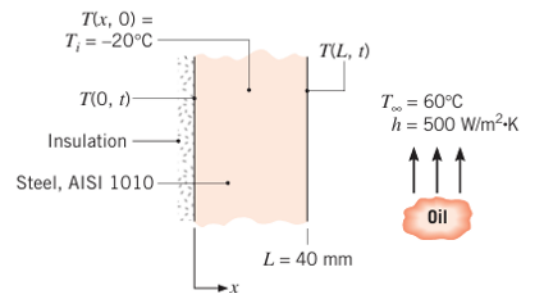
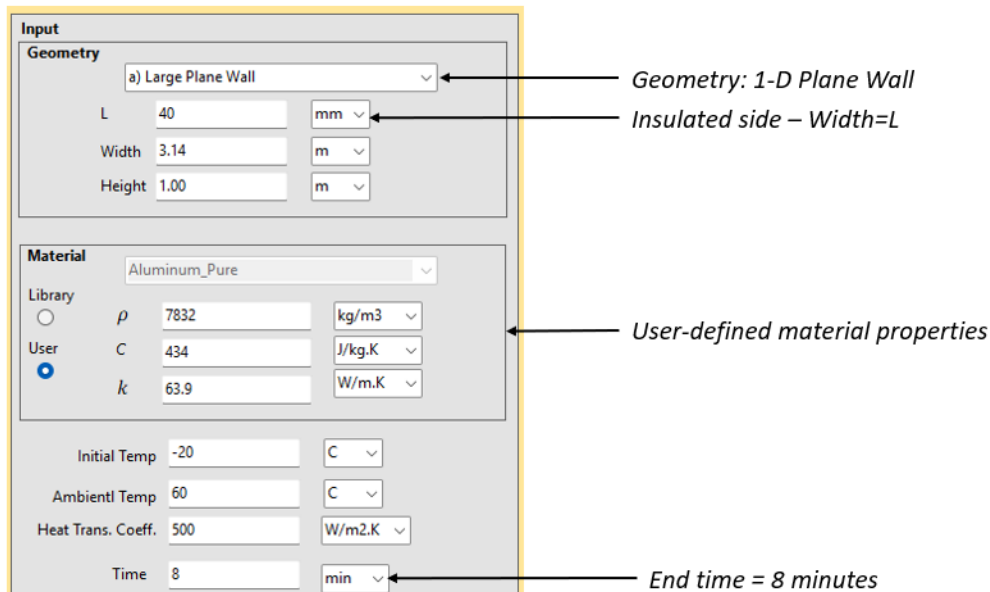
- Determine the temperature of the exterior pipe surface covered by the insulation at $t = 8 \text{ min}$.
- How much energy per meter of pipe length has been transferred from the oil to the pipe at $t = 8 \text{ min}$?

Solution:

- Open “Transient Conduction” Tool



- Enter input data



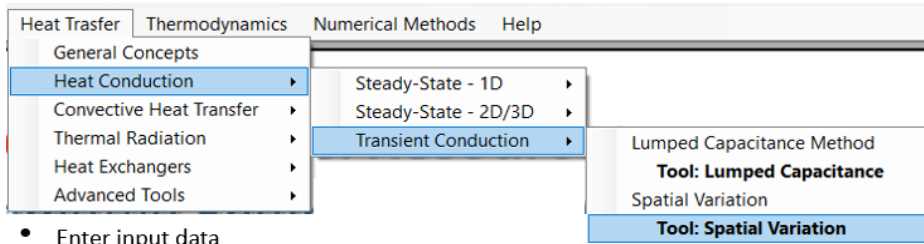
- Click Update and Solve

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 with a heat transfer coefficient of 80 W/m².K. Determine a) the temperature at the center of the shaft after 45 minutes and b) the 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

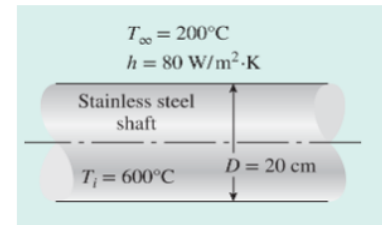
The input data form is shown with the following fields and values:

- Geometry:** b) Long Cylinder (selected), r0 = 10 cm, Height = 1.00 m.
- Material:** Steel_Stainless_316 (selected).
- Library:** $\rho = 7900$ kg/m³, $c_p = 477$ J/kg.K, $k = 14.9$ W/m.K.
- User:** (selected).
- Initial Temp:** 600 C.
- Ambientl Temp:** 200 C.
- Heat Trans. Coeff.:** 80 W/m².K.
- Time:** 45 min.

Geometry: Long Cylinder
Radius, r0

User-defined material properties

End time = 45 minutes



Properties of Stainless Steel as given:

$$\begin{aligned}\rho &= 7900 \frac{\text{kg}}{\text{m}^3} \\ c_p &= 477 \frac{\text{J}}{\text{kg} \cdot \text{K}} \\ k &= 14.9 \frac{\text{W}}{\text{m} \cdot \text{K}}\end{aligned}$$

- Click Update and Solve

Transient Conduction

Input

Geometry

b) Long Cylinder

r0

10

cm

Height

1.00

m

Material

Steel_Stainless_316

Library

ρ

7900

kg/m3

User

C

477

J/kg.K

k

14.9

W/m.K

Initial Temp

600

C

Ambientl Temp

200

C

Heat Trans. Coeff.

80

W/m2.K

Time

45

min

Results

Bi

0.537

Center Temp

362.83

C

Fo

1.068

Total Energy Gain

3.028E+07

J

Internal Temp

r/r0

1.0

Temp

326.45

C

a) Large Plane Wall

2L

0

L

x

b) Long Cylinder

0

r0

c) Sphere

r0

d) Long Rectangular Bar

2L

2W

2L

2W

2L

2W

2L

2W

e) Rectangular Bar

2L

2W

2L

2W

f) Cylinder

2H

r0

Update

Show Tutorial

Comments:

1. The Biot number, $Bi = 0.537$, and the Fourier number $Fo=1.068$
2. The "Center Temperature" refers to the temperature at the center of the cylinder, which is 362.83 °C.
3. The "Total Energy Gain" is $Q_{tot} = 3.028E+07$ J per meter of the pipe.
4. The surface temperature ($r/r_0=1$) is calculated to be 326.45.

 AutoTherm

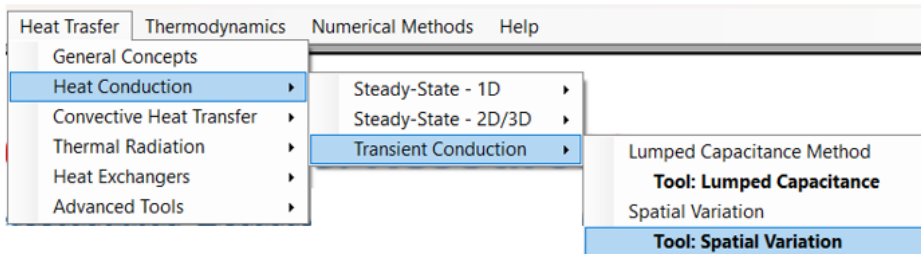
ETR, LLC. Proprietary and Confidential. All rights reserved.

Example – Sphere: Boiling of Egg [Cengel-Ghajar Example 4-5]

An egg may be modeled as a spherical object of 5-cm diameter ($\rho=1000 \text{ kg/m}^3$, $c_p=4150 \text{ J/kg} \cdot ^\circ\text{C}$, $k=0.627 \text{ W/m} \cdot \text{K}$). 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 $1200 \text{ W/m}^2 \cdot \text{K}$.

Solution:

- Open “Transient Conduction” Tool



- Enter input data

The 'Input' dialog box contains the following fields:

- Geometry:** A dropdown menu set to 'c) Sphere'. Below it, a text field for 'r0' is set to '2.5' with a unit dropdown set to 'cm'.
- Material:** A dropdown menu set to 'Aluminum_Pure'. Below it, a table for material properties is shown:

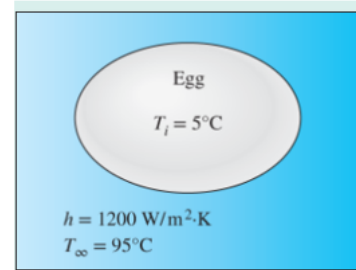
Library	ρ		
☐	994	kg/m ³	
☐	C	4178	J/kg.K
☒	k	0.623	W/m.K
- Initial Temp:** 5 C
- Ambient Temp:** 95 C
- Heat Trans. Coeff.:** 1200 W/m².K
- Time:** 14 min

Geometry: Long Cylinder

Radius, r0

User-defined material properties

End time = 45 minutes



Assumptions:

- Egg modeled as a sphere with $D=2.5\text{cm}$
- Egg is mostly water, so its properties are assumed as water the mean temperature.

Thermal properties of egg:

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

$$c_p = 4178 \frac{\text{J}}{\text{kg} \cdot \text{K}}$$

$$k = 0.623 \frac{\text{W}}{\text{m} \cdot \text{K}}$$

- Click Update and Solve

Transient Conduction
— □ ×

Input

Geometry

r0

Material

☐	ρ	994	kg/m<sup>3</sup>
☒	C	4178	J/kg.K
	k	0.623	W/m.K

Initial Temp
Ambientl Temp
Heat Trans. Coeff.
Time

Results

Bi	48.154		Center Temp	71.71	C
Fo	0.202		Total Energy Gain	-2.244E+04	J

Internal Temp

r/r0

Temp

a) Large Plane Wall b) Long Cylinder c) Sphere

d) Long Rectangular Bar e) Rectangular Bar f) Cylinder

Comments:

Example – Short Cylinder: Cooling of a Brass Bar

A short brass cylinder ($\rho = 8530 \text{ kg/m}^3$, $C_p = 0.389 \text{ kJ/kg.K}$, $k = 110 \text{ W/m.K}$) of diameter 4 cm and height 20 cm is initially at a uniform temperature of 150°C. The cylinder is now placed in atmospheric air at 20°C, where heat transfer takes place by convection with a heat transfer coefficient of 40 W/m².K. Calculate

- A. the center temperature of the cylinder.
- B. the center temperature of the top surface of the cylinder 15 min after the start of the cooling.

Solution:

The screenshot shows the AutoTherm software interface. The 'Heat Conduction' menu is open, and 'Transient Conduction' is selected. The 'Tool: Spatial Variation' is chosen. The 'Geometry' section shows a cylinder with radius $r_0 = 2 \text{ cm}$ and height $H = 10 \text{ cm}$. The 'Material' section shows 'Brass' selected, with properties $\rho = 8530 \text{ kg/m}^3$, $C = 389 \text{ J/kg.K}$, and $k = 110 \text{ W/m.K}$. The 'Initial Conditions' section shows 'Initial Temp = 150 C', 'Ambient Temp = 20 C', 'Heat Trans. Coeff. = 40 W/m².K', and 'Time = 15 min'.

Geometry: Cylinder
Radius, $r_0 = 2 \text{ cm}$ (4-cm diameter)
Height, $H = 10 \text{ cm}$ (half of total height)

Since the material properties are specified, choose User option and enter the properties.

Enter the initial temperature, ambient temperature, heat transfer coefficient, and cooling time.

Diagram of the cylinder: A brass cylinder with a diameter of 4 cm and a height of 20 cm. The initial temperature is 150°C. The ambient air is at 20°C.

- Click update to solve.

The finished form is shown.

Transient Conduction

Input

Geometry

f) Cylinder

r0 2 cm

H 10 cm

Material

Brass

Library

ρ 8530 kg/m³

User

C 389 J/kg.K

k 110 W/m.K

Initial Temp 150 C

Ambient Temp 20 C

Heat Trans. Coeff. 40 W/m².K

Time 15 min

Results

Bi 0.007 Center Temp 60.45 C

Fo 74.589 Total Energy Gain 7.498E+04 J

Internal Temp

r/r0 1.0 z/H 1.0

Temp 59.57 C

Location of center of the top surface

B) Temperature at center of the top surface

Update

Show Tutorial

a) Large Plane Wall

b) Long Cylinder

c) Sphere

d) Long Rectangular Bar

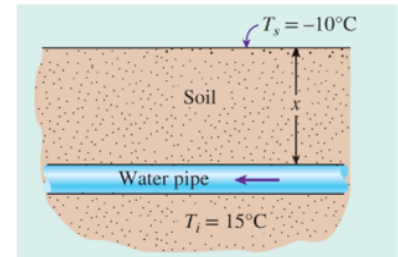
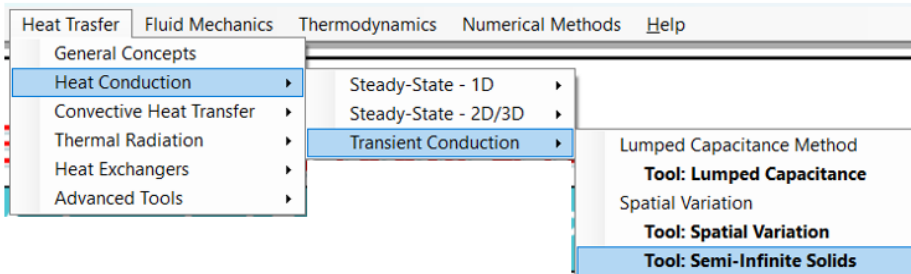
e) Rectangular Bar

f) Cylinder

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 ($90 \times 24 = 2160 \text{ hours}$), and the average soil properties at that location are $k = 0.4 \text{ W/m.K}$, $\rho = 8333.3 \text{ kg/m}^3$, and $c_p = 320 \text{ J/kg.K}$. Assuming an initial uniform temperature of 15°C for the ground, determine whether 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

Material	
Library	Aluminum_Pure
☐ User	
ρ	8333.3 kg/m ³
c	320 J/kg.K
k	0.4 W/m.K

Boundary Condition	
☒ Specified Temperature	
T_s	-10 C
☐ Heat Transfer Coef	
h	W/m ² .K
☐ Uniform Heat Flux	
$q''s$	W/m ²

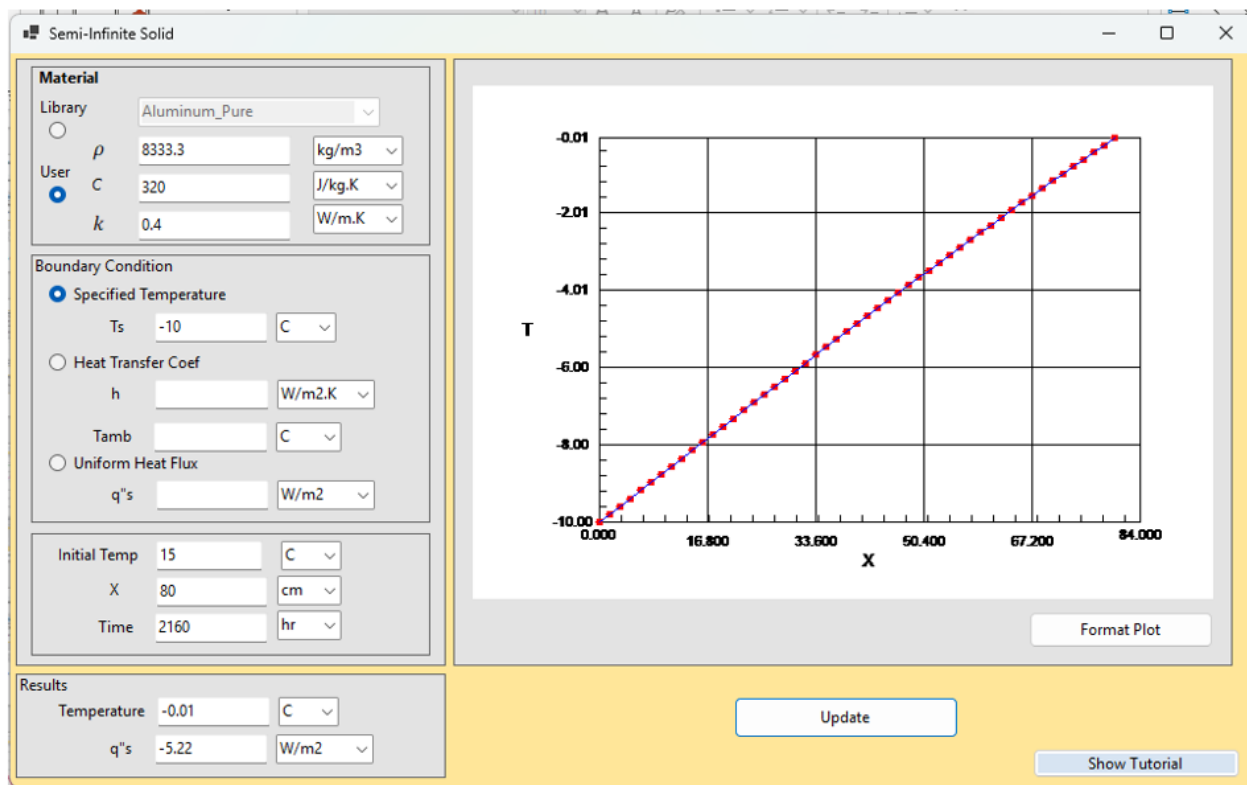
Initial Temp	15	C
X	80	cm
Time	2160	hr

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

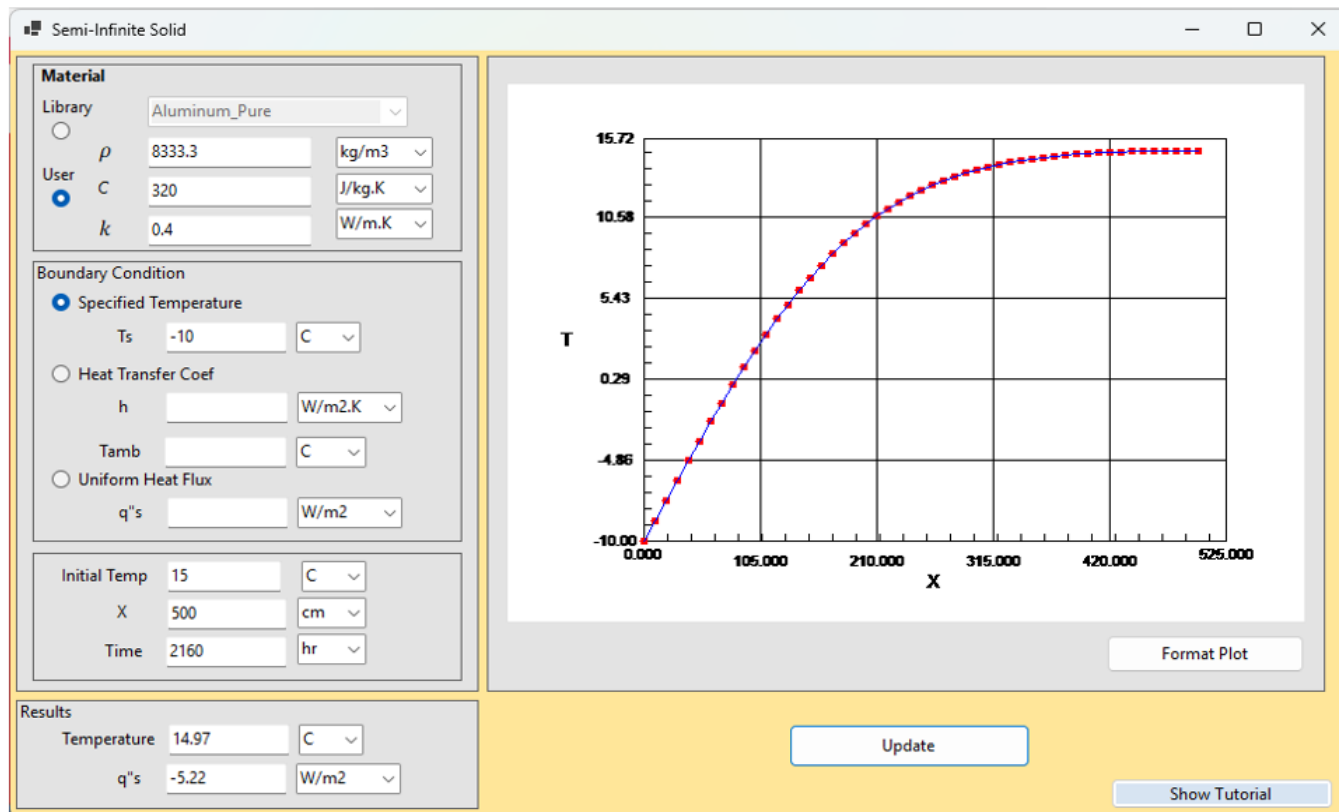
3. Choose "Specified Temperature" BC and enter the given value.

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

Finished form is shown below.



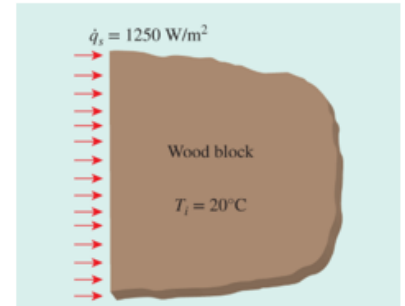
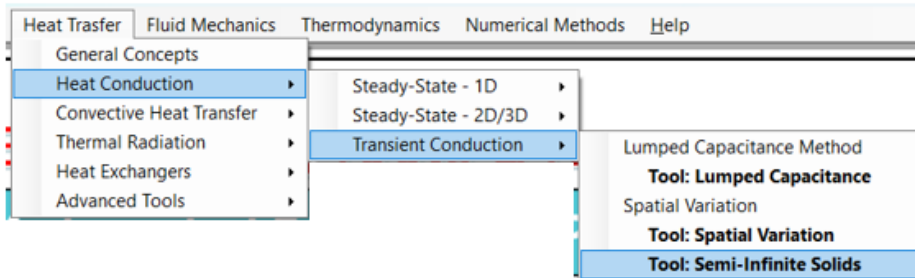
Note: 1. The value of 80 cm is the minimum burial depth of pipes, since the pipe temperature reaches 0 °C after 3 months.
 2. As evident from the plot, the at this depth the variation is nearly linear. If you rerun with $x=500$ cm, we can see the decay.



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 $20^\circ C$ is subjected to constant solar heat flux of $1250 W/m^2$.

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



Enter input data

The 'Material' input form has two tabs: 'Library' and 'User'. The 'User' tab is selected. It contains three input fields: density (ρ) with a value of 721 and units of kg/m^3 , specific heat (C) with a value of 1260 and units of $J/kg.K$, and thermal conductivity (k) with a value of 0.159 and units of $W/m.K$.

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

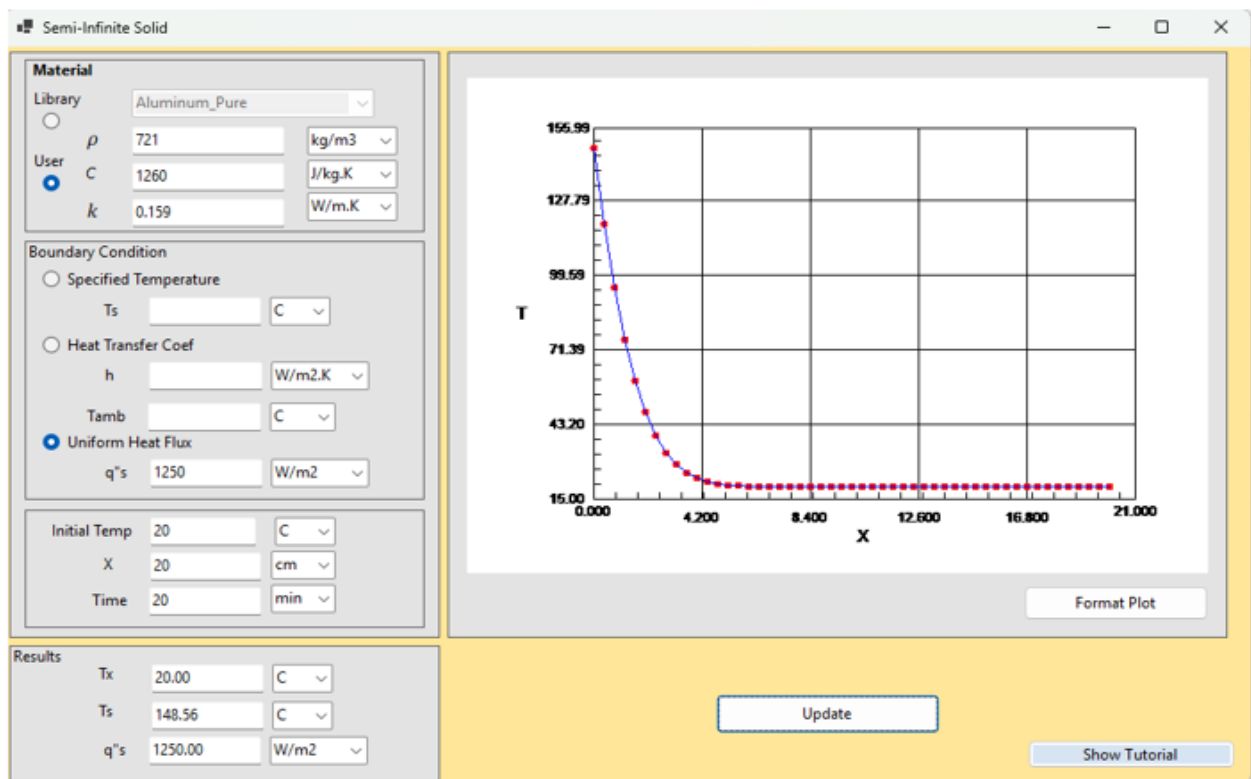
The 'Boundary Condition' input form has two radio buttons: 'Specified Temperature' and 'Uniform Heat Flux'. The 'Uniform Heat Flux' option is selected. It contains a field for heat flux (q''_s) with a value of 1250 and units of W/m^2 .

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

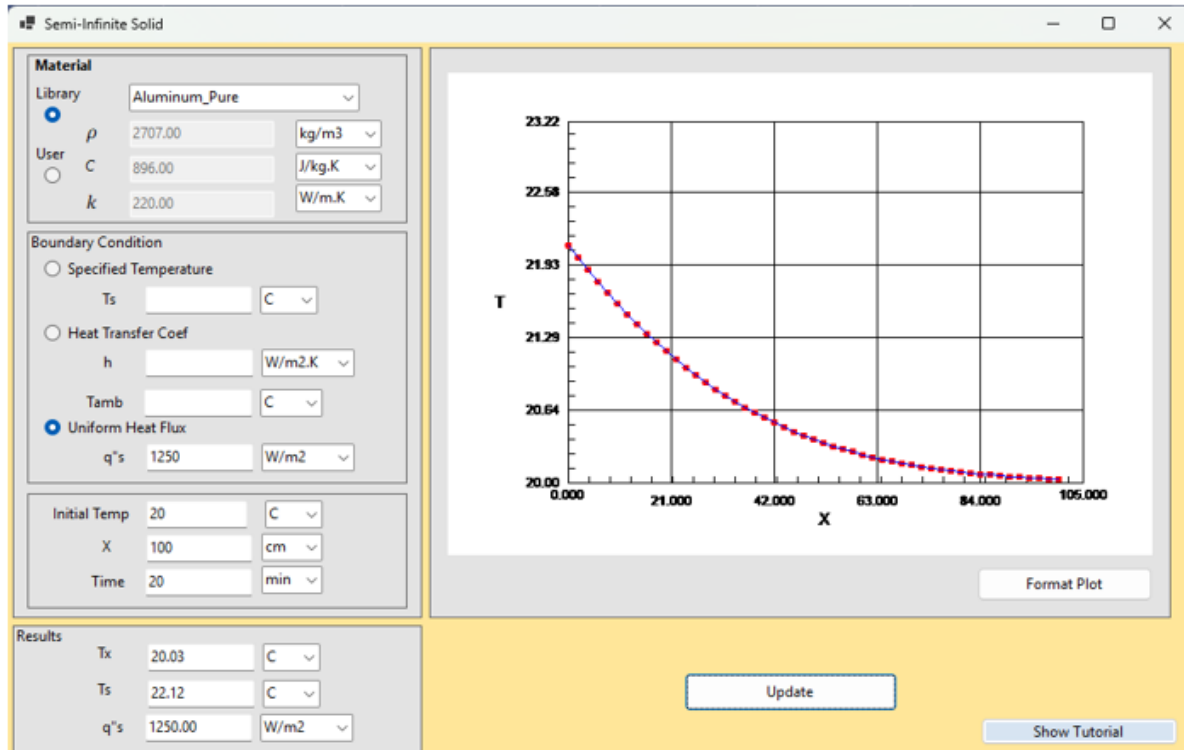
The 'Initial conditions' input form has three input fields: 'Initial Temp' with a value of 20 and units of $^\circ C$, 'X' with a value of 0 and units of cm , and 'Time' with a value of 20 and units of min .

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

Finished form is shown below.



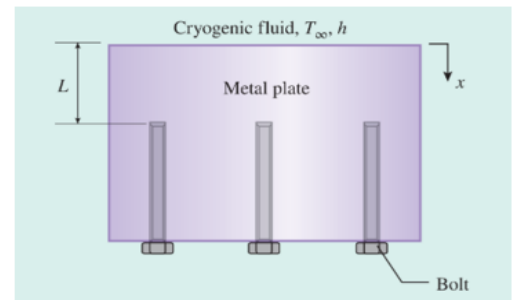
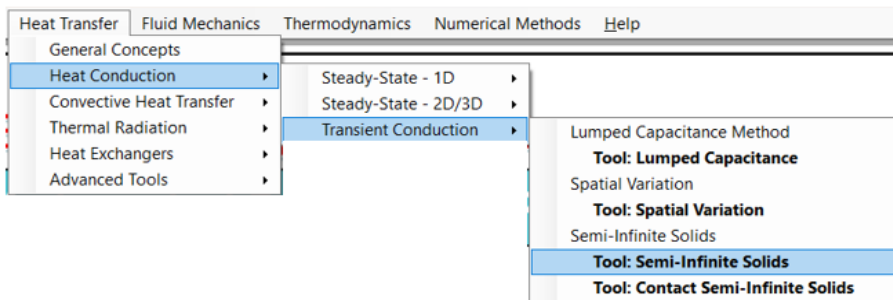
- Wood Block:**
1. The surface temperature is 148.6 °C.
 2. The plot shows thermal penetration within the wooden block to the depth of 4.2 cm.



- 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?



- Select User Materials and enter properties in given units.

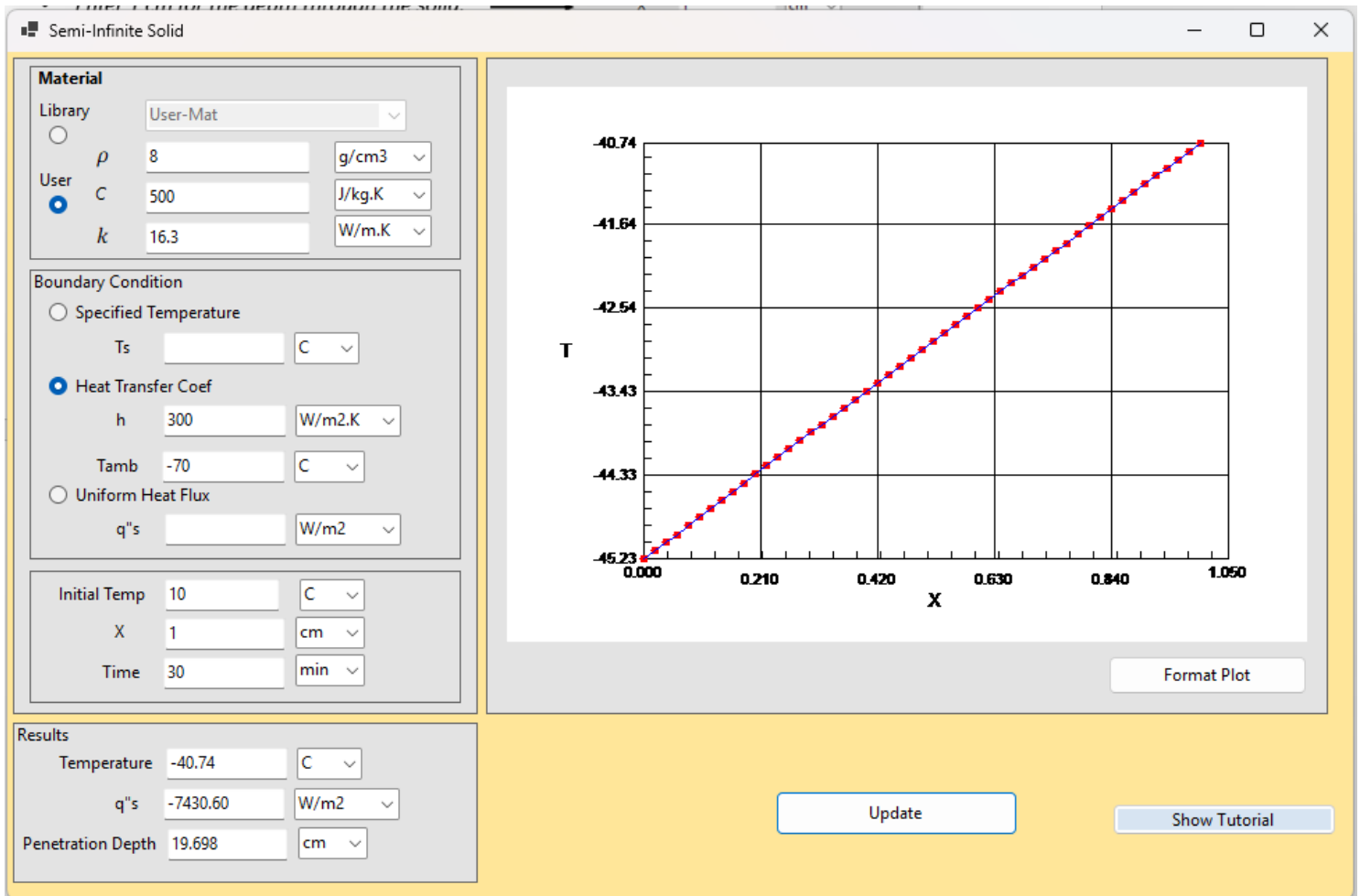
- Choose the 2nd type of boundary conditions and enter 300 W/m².K for the heat transfer coefficient and -70 ° for the ambient temperature.

- Enter 10 °C for the initial temperature.
- Enter 1 cm for the depth through the solid.
- Enter 10 min for time.

The screenshot shows the software's input form with the following settings and arrows indicating where instructions point to:

- Material:** Library: User-Mat. Properties: $\rho = 8$ g/cm³, $C = 500$ J/kg.K, $k = 16.3$ W/m.K.
- Boundary Condition:** Specified Temperature (unselected), Heat Transfer Coef (selected). $h = 300$ W/m².K, $T_{amb} = -70$ °C.
- Initial Temp:** 10 °C.
- X:** 1 cm.
- Time:** 30 min.

Click update to solve.



- The temperature at a depth of 1 cm after 10 minutes is predicted to be -40.74 °C. This indicates that the bolt tips are below the minimum temperature of -30 °C, as required by the ASME Code.
- The penetration depth is 19.7 cm at this time in the transient process.
- The plot shows a linear temperature profile through this depth at 30 minutes.

HT-11: Contact of Two Semi-Infinite Solids

A barefoot person whose feet are at 32 °C steps on a large aluminum block at 20 °C. Treating both the feet and the aluminum block as semi-infinite solids, determine the contact surface temperature.

For human flesh assume: $\rho=1000 \text{ kg/m}^3$, $C_p=3500 \text{ J/kg}\cdot\text{K}$, $k=0.3 \text{ W/m}\cdot\text{K}$.

Solution:

From the “Heat Transfer/Heat Conduction/Transient Conduction” menu, click on “Tool: Contact Semi-Infinite Solids.”

1. Leave “Solid 1” as Aluminum (Default material).
 - Enter 20 °C for the initial temperature
2. For “Solid 2”:
 - Enter 1000 kg/m^3 for density
 - Enter 3500 $\text{J/kg}\cdot\text{K}$ for specific heat
 - Enter 0.3 $\text{W/m}\cdot\text{K}$ for thermal conductivity
 - Enter 32 °C for the initial temperature
3. Click “Update for Interface Temperature.”

Contact of Two Semi-Infinite Solids

Solid 1

Material

Library: Aluminum_Pure

User: ρ 2707 kg/m³, C 896 J/kg.K, k 220 W/m.K

Initial Temp: 20 °C

Solid 2

Material

Library: User-Mat

User: ρ 1000 kg/m³, C 3500 J/kg.K, k 0.3 W/m.K

Initial Temp: 32 °C

Results

Interface Temp: 20.51 °C [Update]

Time: [] s

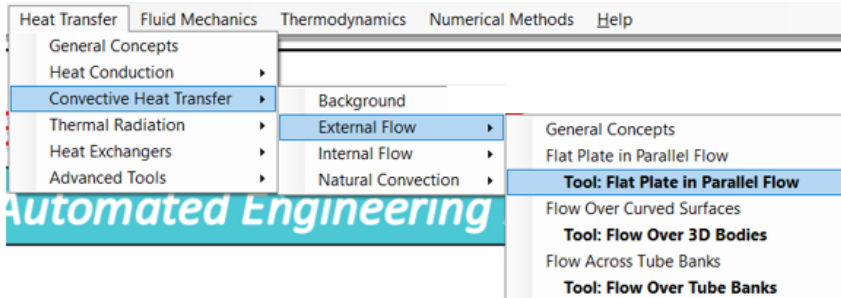
Interface Heat Flux: [] W/m² [Update]

The contact surface temperature is calculated to be 20.5 °C.

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.

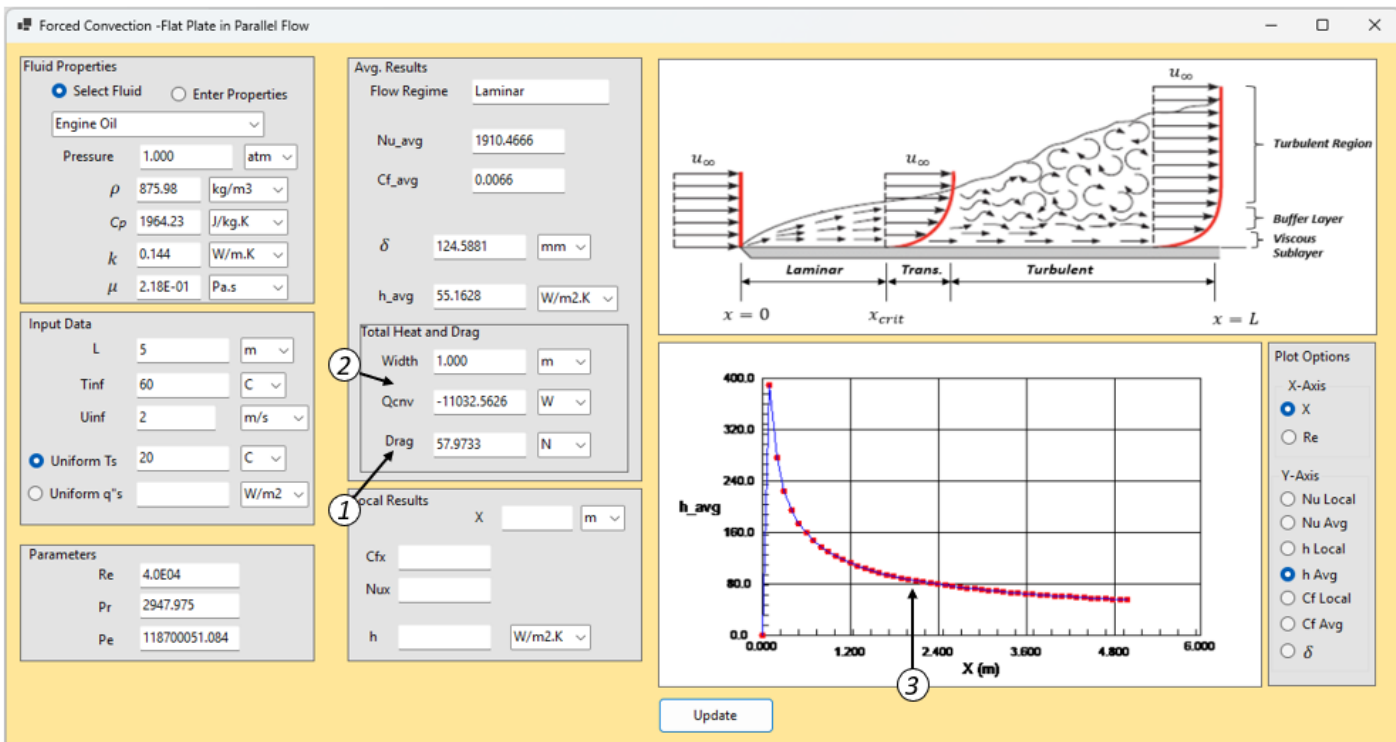


- Choose Engine Oil from the Library.

The 'Fluid Properties' dialog box is shown. The 'Select Fluid' radio button is selected, and 'Engine Oil' is chosen from the dropdown menu. The 'Pressure' is set to 1.000 atm. The properties ρ , C_p , k , and μ are listed with their respective units (kg/m³, J/kg.K, W/m.K, and Pa.s).

- Enter 5 m for Length
- Set the oil temperature to 60 °C
- Set the velocity to 2 m/s
- Set the plate temperature to 20 °C

The 'Input Data' dialog box is shown. The 'Uniform Ts' radio button is selected, and the plate temperature is set to 20 °C. The 'Length' (L) is set to 5 m, the 'Inlet Temperature' (T_{inf}) is set to 60 °C, and the 'Inlet Velocity' (U_{inf}) is set to 2 m/s. The 'Uniform q''s' option is also visible but not selected.



1. The drag force is calculated to be 58 N.
2. The rate of heat removal from the oil (heat addition to the plate) is roughly 11 kW.
3. The plot shows the variation of the average heat transfer coefficient versus the distance to the leading edge (other properties may be selected).

HT-13: Flow Over 3D Bodies

Example – Heating of Horizontal Cylinder in Cross Flow

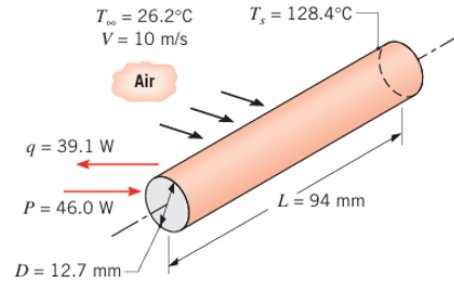
Experiments have been conducted using a metallic cylinder 12.7 mm in diameter and 94 mm long. The cylinder is heated internally by an electric heater and subjected to a cross-flow of air in a low-speed wind tunnel. Under a specific set of operating conditions for which the upstream air velocity and temperature 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 $T_s = 128.4^\circ\text{C}$. Assuming that 15% of the power dissipation is lost through the cumulative effects of surface radiation and conduction through the endpieces:

- Determine the convection heat transfer coefficient from the experimental observations.
- Compare the experimental result with the convection coefficient computed from the correlation.

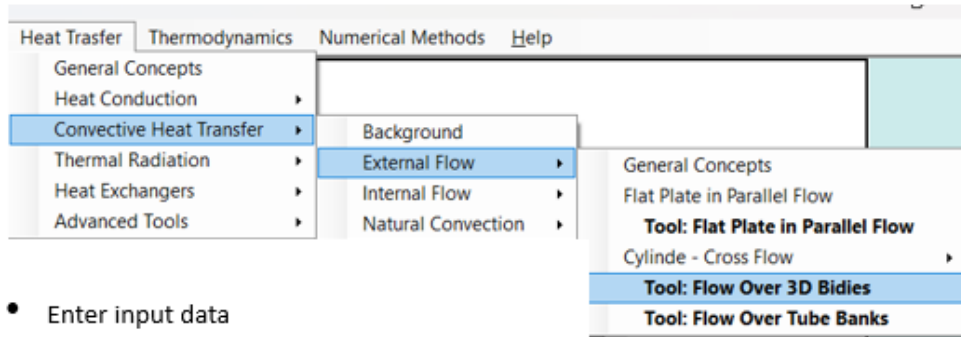
- a) The convection coefficient can be evaluated from the experimental data, assuming 85% of the total heat dissipation from the surface of the rod:

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

$$\bar{h} = \frac{\overbrace{0.85 \times 46}^q}{\underbrace{\pi(0.0127)(0.094)}_A(128.4 - 26.2)} = 102 \frac{\text{W}}{\text{m}^2 \cdot \text{K}}$$



- b) To obtain the heat transfer coefficient using the correlation, open “Flow Over 3D Bodies Tool”



- Enter input data

Geometry: Circular Cylinder → a) Circular Cylinder

Select Air from materials library → Select Fluid (Air selected)

Enter Flow and Thermal Information →

Enter Geometric Information →

Fluid Properties	
Fluid	Air
Pressure	1.000 atm
ρ	kg/m ³
C_p	J/kg.K
k	W/m.K
μ	Pa.s

Input Parameters	
U_f	10 m/s
T_{inf}	26 C
T_s	128.4 C
D	12.7 mm
L	94 mm
Emissivity	1.0

- Click Update and Solve

External Flow

a) Circular Cylinder

Fluid Properties

Select Fluid ☒ Enter Properties ☐

Air

Pressure 1.000 atm

ρ kg/m³

C_p J/kg.K

k W/m.K

μ Pa.s

Input Parameters

U_f 10 m/s

T_{inf} 26 C

T_s 128.4 C

D 12.7 mm

L 94 mm

Emissivity 1.0

Results

Re 6059.65

Pr 0.700

Nu_{avg} 40.5946

h_{avg} 95.98 W/m².K

Q_{tot} 40.685 W

Q_{conv} 36.860 W

Q_{rad} 3.826 W

(u_f, T_∞)

a) Circular Cylinder

b) Sphere

Noncircular Cylinders

c) Square

d) Tilted Square

e) Hexagon

f) Tilted Hexagon

g) Plate

Update

Comments:

- The Reynolds number, $Re_D=6059$.
- The Nusselt number, $Nu=40.6$.
- The average heat transfer coefficient is evaluated to be 95.6 W/m².K. This is very close (about 6% difference) to the experimentally determined value of 102 W/m².K.
- 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.

 AutoTherm

ETR, LLC. Proprietary and Confidential. All rights reserved.

HT-14: Flow Over Tube Banks

The following three examples illustrate how to solve heat transfer problems involving flow across tube banks in 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 the 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

Heat Transfer Thermodynamics Numerical Methods Help

General Concepts
Heat Conduction
Convective Heat Transfer
Thermal Radiation
Heat Exchangers
Advanced Tools

Background
External Flow
Internal Flow
Natural Convection

General Concepts
Flat Plate in Parallel Flow
Tool: Flat Plate in Parallel Flow
Cylinder - Cross Flow
Tool: Flow Over 3D Bodies
Tool: Flow Over Tube Banks

Enter input data

☒ In-line ☐ Staggered

Fluid Properties

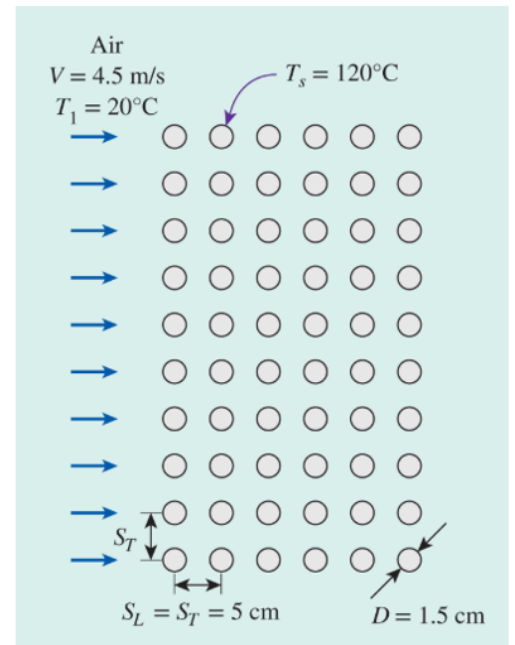
☒ Select Fluid ☐ Enter Properties

Air Pressure 1.000 atm

ρ_i kg/m³
 ρ kg/m³
 k W/m.K
 μ Pa.s
 C_p J/kg.K
 μ Pa.s
 Pr

Input Parameters

V 4.5 m/s
Ti 20 C
Ts 120 C
D 1.5 cm
ST 5 cm
SL 5 cm
L 1.0 m
No of Columns - Longitudinal Dir, NL 6
No of Rows - Transverse Dir, NT 10



$$\begin{aligned}N_L &= 6 \\N_T &= 10 \\L &= 1 \text{ m}\end{aligned}$$

- Click Update and Solve

Flow Across Tube Banks

☒ In-line ☐ Staggered

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Air Pressure 1.000 atm

ρ_i 1.19336 kg/m³

ρ 1.170883 kg/m³ C_p 1006.96 J/kg.K

k 0.02614 W/m.K μ 1.8358E-01 Pa.s

Pr 0.708 Prs 0.691

Input Parameters

V 4.5 m/s

Ti 20 C

Ts 120 C

D 1.5 cm

ST 5 cm

SL 5 cm

L 1.0 m

No of Columns - Longitudinal Dir, NL 6

No of Rows - Transverse Dir, NT 10

Results

Re 6150

Nu_avg 55.2343

h_avg 96.2 W/m².K

Te 29.6 C

Q 25888.1 W

Dp/xf 145.166 Pa

Diagram

a) Inline Arrangement

(V, T_∞)

$A_1 = S_T L$
 $A_T = (S_T - D)L$
 $A_D = (S_D - D)L$

b) Staggered Arrangement

(V, T_∞)

Update

Show Tutorial

Comments:

- The Reynolds number, $Re=6150$; it is based on V_{max}
- Average heat transfer coefficient is 96.2 W/m.K
- Air exit temperature is 29.6 °C
- Heat transfer into air is 25888.1 W (25.8 kW)
- Pressure drop across the bank can be calculated to be 23.2 Pa by multiplying the 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).
- Note that, following the solution process, the values of fluid thermal properties are displayed corresponding to the latest updated film temperature. The user can use fixed properties by selecting the "Enter Properties" option.
- ρ_i (1.193 shown in the first density box) represents the value of density at the inlet (evaluated using inlet temperature). This value determines 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 $S = S_T = 4$ 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

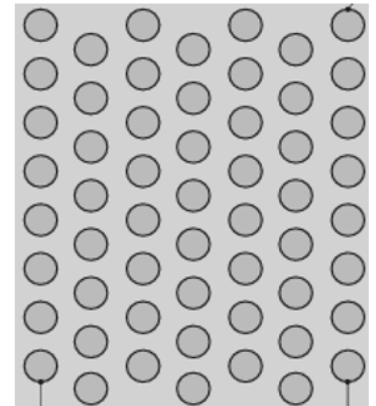
The screenshot shows the AutoTherm software interface. The 'Heat Transfer' menu is open, leading to 'Convective Heat Transfer', then 'External Flow', and finally 'Tool: Flow Over Tube Banks'. Below this, the 'Enter input data' section is visible, containing two main panels: 'Fluid Properties' and 'Input Parameters'.

Fluid Properties Panel:

- Buttons: ☒ In-line, ☐ Staggered
- Options: ☐ Select Fluid, ☒ Enter Properties
- Fluid: Air, Pressure: 1.000 atm
- ρ_i : 1.20400 kg/m³
- ρ : 1.145000 kg/m³, C_p : 1007.00 J/kg.K
- k : 0.02625 W/m.K, μ : 1.8950E-05 Pa.s
- Pr: 0.727, Prs: 0.711

Input Parameters Panel:

- V: 5.2 m/s
- T_i : 20 °C
- T_s : 100 °C
- D: 1.6 cm
- S_T : 4 cm
- S_L : 4 m
- L: 3.0 m
- No of Columns - Longitudinal Dir, NL: 20
- No of Rows - Transverse Dir, NT: 10



Staggered Arrangement

$$D = 1.6 \text{ cm}$$

$$S_T = S_L = 4 \text{ cm}$$

$$N_T = 10; N_L = 20$$

$$V = 5.2 \text{ m/s}$$

$$T_i = 20 \text{ °C}$$

$$T_s = 100 \text{ °C}$$

$$L = 3 \text{ m}$$

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

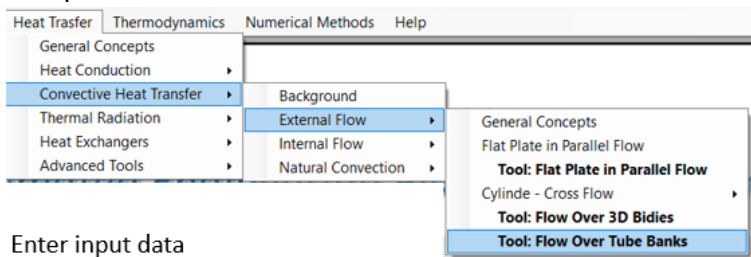
- Click Update and Solve

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 water passes through the tubes while air flows in cross-flow over them. 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 heat transfer rate for the tube bundle.

Solution:

- Open “Flow over Tube Banks” Tool



Enter input data

☐ In-line ☒ Staggered

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Air Pressure 1.000 atm

ρ_i kg/m³

ρ kg/m³ C_p J/kg.K

k W/m.K μ Pa.s

Pr Pr_s

Input Parameters

V 6 m/s

T_i 15 °C

T_s 70 °C

D 16.4 mm

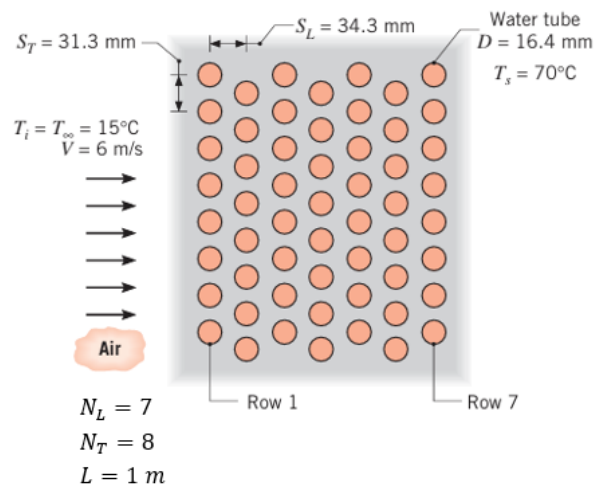
S_T 31.3 mm

S_L 34.3 mm

L 1.0 m

No of Columns - Longitudinal Dir, N_L 7

No of Rows - Transverse Dir, N_T 8



- Click Update and Solve

Flow Across Tube Banks

In-line

Staggered

Fluid Properties

Select Fluid

Enter Properties

Air

Pressure

1.000

atm

ρ_i

1.21669

kg/m³

ρ

1.191489

kg/m³

k

0.02578

W/m.K

Pr

0.709

C_p

1006.87

J/kg.K

μ

1.8138E-01

Pa.s

Prs

0.701

Input Parameters

V

6

m/s

T_i

15

C

T_s

70

C

D

16.4

mm

ST

31.3

mm

SL

34.3

mm

L

1.0

m

No of Columns - Longitudinal Dir, NL

7

No of Rows - Transverse Dir, NT

8

Results

Re

13579

Nu_{avg}

88.1978

h_{avg}

138.7

W/m².K

Te

25.7

C

Q

19777.1

W

Dp/xf

662.486

Pa

(V, T_∞)

S_L

S_T

A_1

A_T

$A_1 = S_T L$

$A_T = (S_T - D)L$

$A_D = (S_D - D)L$

(V, T_∞)

S_L

S_D

S_T

A_1

A_T

A_D

a) Inline Arrangement

b) Staggered Arrangement

Update

Show Tutorial

Comments:

1. The Reynolds number, Re , is 13,579, and it is based on V_{max}
2. The average heat transfer coefficient is 138.7 W/m².K
3. Air exit temperature is 25.7 °C
4. Heat transfer into the air is 19,777 W (19.8 kW)
5. Pressure drop across the bank can be calculated to be 241.15 Pa by multiplying the 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, the values of fluid thermal properties are displayed corresponding to the latest updated film temperature.

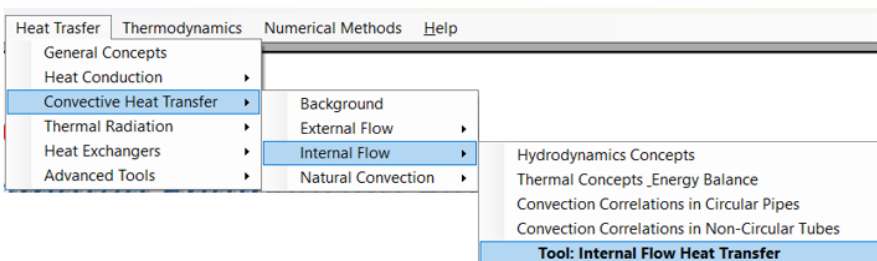
 AutoTherm

ETR, LLC. Proprietary and Confidential. All rights reserved.

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 the icy waters of a lake at 0 °C. 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.

Fluid Properties

☐ Select Fluid ☒ Enter Properties

Oil

Pressure 1.000 atm

ρ 888.1 kg/m³ C_p 1880 J/kg·K

k 0.145 W/m·K μ 0.8373 Pa·s

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

Geometric Information

☒ Circular ☐ Non-Circular

D 0.3 m L 200 m

- Select “Uin” to enter inlet velocity.

Inlet Flow Information

☐ mdot ☐ Vdot ☒ Uin

2 m/s

- Enter the Inlet Temperature.
- Select “Uniform Surface Temperature” option and Enter the value provided.

Thermal Conditions

Tin 20 C

☐ Uniform q”s ☒ Uniform Ts 0 C

☐ Uniform Tinf

h_ext W/m²·K

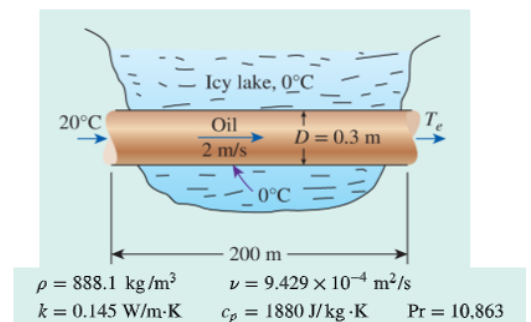
K_w W/m·K

th_w mm

- Make sure Thermal Entry Effects are included.

☒ Include Thermal Entry Effects

Click Update to Solve



Fluid Properties
☐ Select Fluid ☒ Enter Properties
 Oil
 Pressure 1.000 atm
 ρ 8.88E02 kg/m³ C_p 1.88E03 J/kg.K
 k 1.45E-01 W/m.K μ 8.37E-01 Pa.s

Geometric Information
☒ Circular ☐ Non-Circular
 D 0.3 m L 200 m

Inlet Flow Information
☐ $\dot{m}$ 125.54 kg/s
☐ $\dot{V}$ 299.55 cfm
☒ U_{in} 2 m/s

Thermal Conditions
 T_{in} 20 C
☐ Uniform q''_s
☒ Uniform T_s 0 C
☐ Uniform T_{inf}
 h_{ext} W/m².K
 K_w W/m.K
 th_w mm
☒ Include Thermal Entry Effects

Results
 A_c 7.07E-02 m²
 Per 0.9425 m
 Re 637 Pr 10852.14
 Flow Regime Laminar
 Flow Conditions
 Velocity Fully Developed Velocity
 Temperature Entry Region Thermal

Average
 Nu_{avg} 39.4319
 h_{avg} 19.1 W/m².K
 q''_{tot} -71305.7 W

Exit
 Nu_o 25.7935
 h_o 12.5 W/m².K
 T_{mo} 19.7 C
 T_{so} 0.0 C
 $q''_{s,o}$ -245.6 W/m²

Local Results
 x/L 0.5
 Nu 32.6080
 T_m 19.9 C
 T_s 0.0 C
 h 15.8 W/m².K
 $q''_{s,x}$ -313.2 W/m²

Correlation Corner
 Turbulent Flow - Fully Developed
☒ Gnielinsky Correlation (Smooth Wall)
☐ Dittus-Boelter Correlation (Smooth Wall)
☐ Bhatti-Shah (Rough Wall)
☐ Rough wall
☐ Adjust Properties for TWall Effects
 Show Correlations

Results - Pressure Drop and Friction
 Darcy Friction Factor, f 0.1005
 Friction Head Loss, h_f 13.6650 m
 Pressure Drop 119.0400 kPa
 Friction Factor
☒ Haaland Equation
☐ Swamee-Jain Equation
☐ Colebrook Equation
 DZ 0.00 m
 Update

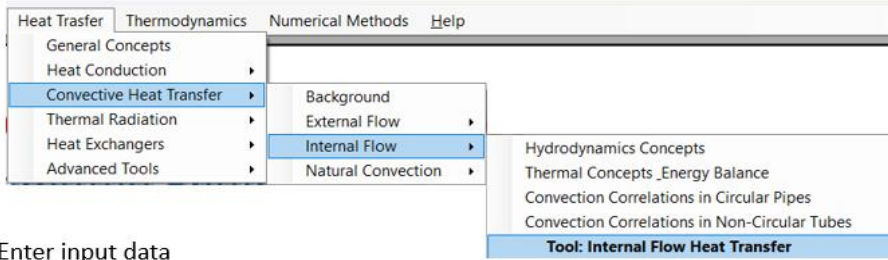
Comments:

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 ($f = \frac{64}{Re}$).
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:

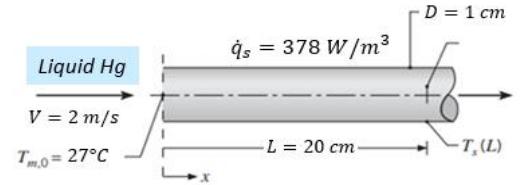
$$\dot{W}_P = \frac{\dot{m}\Delta p}{\rho} = \frac{(125.54)(119.04)}{(888.1)} = 16.8 \text{ kW}$$

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

A fully developed flow of air at 27 °C 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 of 378 W/m². What will be the mean temperature and the wall temperature at the exit?



Enter input data



- Choose the default – Select Fluid option and Air as the fluid.

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Air

Pressure 1.000 atm

ρ kg/m³ C_p J/kg.K

k W/m.K μ Pa.s

- Keep Circular option.
- Enter D = 1 cm and L = 20 cm

Geometric Information

☒ Circular ☐ Non-Circular

D 1 cm L 20 cm

- Enter 2 m/s for $\dot{m}$.

Inlet Flow Information

☐ $\dot{m}$ kg/s

☐ $\dot{V}$ cfm

☒ U_{in} 2 m/s

- Enter the 27 °C for Inlet Temperature.

Thermal Conditions

T_{in} 27 C

☒ Uniform q''_s 378 W/m²

☐ Uniform T_s C

☐ Uniform T_{inf} C

h_{ext} W/m².K

K_w W/m.K

th_w mm

- Select the “Uniform Heat Flux” option and enter 378 W/m²

- Select “Include Thermal Entry Effects”.

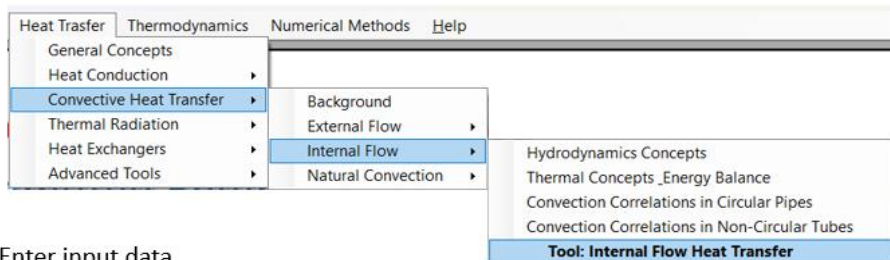
☒ Include Thermal Entry Effects

- Click Update to Solve.

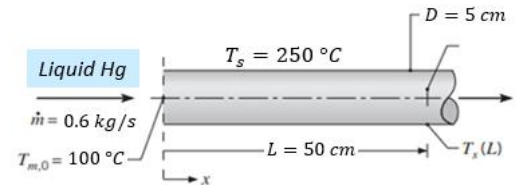
Example – Low Prandtl Number Flow – Liquid Mercury Flow in a Pipe [Source: Cengel-Ghagar Problem 8.125]

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

- Determine the outlet mean temperature at $x=50$ cm.
- Determine the rate of heat transfer to mercury for this length of pipe.



Enter input data



- Choose the default – Select Fluid option and Mercury as the fluid.

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Mercury

Pressure 1.000 atm

ρ kg/m³ C_p J/kg.K

k W/m.K μ Pa.s

- Keep Circular option.
- Enter $D = 5$ cm and $L = 50$ m

Geometric Information

☒ Circular ☐ Non-Circular

D 5 cm L 50 cm

- Enter 0.6 kg/s for $\dot{m}$.

Inlet Flow Information

☒ $\dot{m}$ 0.6 kg/s

☐ $\dot{V}$ cfm

☐ U_{in} m/s

- Enter the 100 °C for Inlet Temperature.

Thermal Conditions

T_{in} 100 C

☐ Uniform q''_s W/m²

☒ Uniform T_s 250 C

☐ Uniform T_{inf} C

h_{ext} W/m².K

K_w W/m.K

th_w mm

- Select the "Uniform Temperature" option and enter 250 °C

Note: For this problem, due to the very small Pr of mercury, the appropriate liquid metal correlation will be automatically used.

- Click Update to Solve.

Fluid Properties
☒ Select Fluid ☐ Enter Properties
Mercury
Pressure: 1.000 atm
 ρ : 13233.233 kg/m³ C_p : 136.1416 J/kg.K
 k : 10.0688 W/m.K μ : 1.128E+03 Pa.s

Geometric Information
☒ Circular ☐ Non-Circular
D: 5 cm L: 50 cm

Inlet Flow Information
☒ mdot: 0.6 kg/s
☐ Vdot: 0.10 cfm
☐ Uin: 0.02 m/s

Thermal Conditions
Tin: 100 C
☐ Uniform q''_s
☒ Uniform T_s : 250 C
☐ Uniform T_{inf}
 h_{ext} : W/m².K
 K_w : W/m.K
 th_w : mm
☐ Include Thermal Entry Effects

Results
Ac: 1.96E-03 m²
Per: 0.1571 m
Re: 13545 Pr: 0.02
Flow Regime: Turbulent
Flow Conditions:
Velocity: Entry Region Velocity
Temperature: Entry Region Thermal

Average
Nu_avg: 5.8378
 h_{avg} : 1175.6 W/m².K
qtot: 8296.0 W

Exit
Nu_o: 5.7675
 h_o : 1161.4 W/m².K
Tmo: 201.6 C
Tso: 250.0 C
 $q''_{s,o}$: 56259.1 W/m²

Local Results x/L: 0.5
Nu: 5.8168
Tm: 164.6 C
Ts: 250.0 C
h: 1171.4 W/m².K
 $q''_{s,x}$: 100050.5 W/m²

Correlation Corner
Turbulent Flow - Fully Developed
☒ Gnielinsky Correlation (Smooth Wall) ☐ Rough wall
☐ Dittus-Boelter Correlation (Smooth Wall) ϵ : mm
☐ Bhatti-Shah (Rough Wall) ☐ Adjust Properties for TWall Effects
Show Correlations

Results - Pressure Drop and Friction
Darcy Friction Factor, f: 0.0000
Friction Head Loss, hf: 0.0000 m
Pressure Drop: 0.0000 psi
Friction Factor:
☒ Haaland Equation
☐ Swamee-Jain Equation
☐ Colebrook Equation
DZ: 0.00 m
Update

Diagram

Comments:

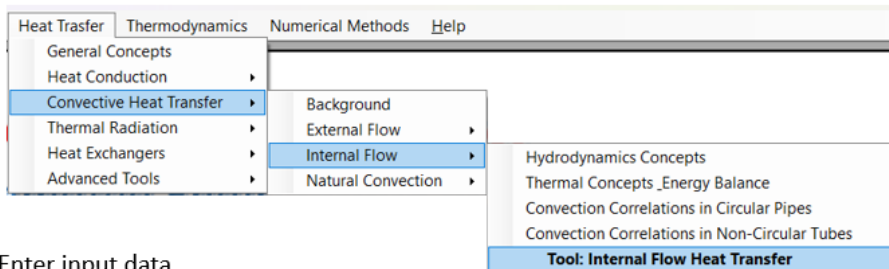
1. The uniform-surface-temperature liquid-metal correlation is used.
2. The results show that roughly 8300 W of heat is transferred to the liquid mercury. The heat transfer coefficient is 1175.6 W/m².K.
3. The air exit temperature is about 200 °C.

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 $D = 0.15\text{ m}$ and $L = 5\text{ m}$, which is in the crawlspace of a house. The heat transfer coefficient between the duct outer surface and the ambient air at $T_\infty = 0^\circ\text{C}$ is known to be $h_{\text{ext}} = 6\text{ W}/(\text{m}^2 \cdot \text{K})$.

- 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 $x = L$.



Enter input data

- Choose the default – Select Fluid option and Air as the fluid.

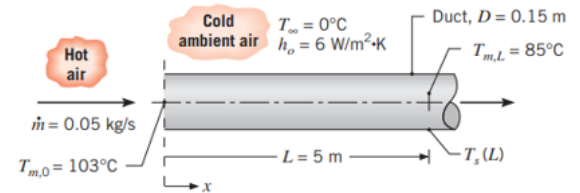
- Keep Circular option.
- Enter $D=0.15\text{m}$ and $L=5\text{m}$

- Enter 0.05 kg/s for $\dot{m}$.

- Enter the 103°C for Inlet Temperature.

- Select the “Uniform Ambient Temperature” option and enter 0°C

- Enter $6.0\text{ W}/\text{m}^2\cdot\text{K}$ for the heat transfer coefficient.



Note: The problem asks to neglect the duct wall resistance. Here, we specify a very high thermal conductivity and a thin wall, which results in negligible wall thermal resistance.

- Click Update to Solve.

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Air

Pressure 1.000 atm

ρ 0.9522 kg/m³ C_p 1010.7290 J/kg.K

k 0.0313 W/m.K μ 2.158E-05 Pa.s

Geometric Information

☒ Circular ☐ Non-Circular

D 0.15 m L 5 m

Inlet Flow Information

☒ $\dot{m}$ 0.05 kg/s

☐ $\dot{V}$ 111.27 cfm

☐ U_{in} 2.97 m/s

Thermal Conditions

T_{in} 103 C

☐ Uniform q''_s W/m²

☐ Uniform T_s C

☒ Uniform T_{in} 0 C

h_{ext} 6.0 W/m².K

K_w 400 W/m.K

th_w 1 mm

☒ Include Thermal Entry Effects

Results

Ac 1.77E-02 m²

Per 0.4712 m

Re 19669 Pr 0.70

Flow Regime Turbulent

Flow Conditions

Velocity Fully Developed Velocity

Temperature Fully Developed Thermal

Average

Nu_{avg} 57.8724

h_{avg} 12.1 W/m².K

q_{tot} -894.6 W

Exit

Nu_o 56.0550

h_o 11.7 W/m².K

T_{mo} 85.3 C

T_{so} 56.4 C

$q''_{s,o}$ -338.3 W/m²

Local Results

x/L 0.5

Nu 54.2765

T_m 93.7 C

T_s 64.2 C

h 11.3 W/m².K

$q''_{s,x}$ -334.6 W/m²

Correlation Corner

Turbulent Flow - Fully Developed

☐ Gnielinsky Correlation (Smooth Wall)

☒ Dittus-Boelter Correlation (Smooth Wall)

☐ Bhatti-Shah (Rough Wall)

☐ Rough wall

ϵ mm

☐ Adjust Properties for TWall Effects

Show Correlations

Results - Pressure Drop and Friction

Darcy Friction Factor, f 0.0262

Friction Head Loss, h_f 0.3935 m

Pressure Drop 0.0005 psi

DZ 0.00 m

Friction Factor

☒ Haaland Equation

☐ Swamee-Jain Equation

☐ Colebrook Equation

Update

Diagram

Diagram showing the thermal entry region and fully developed region of a duct. The thermal entry region is labeled "Thermal Entry Region" and the fully developed region is labeled "Fully Developed". The temperature profile is shown as $T(r, 0) = T_s$ and $T_s > T_{in}$. The heat flux is labeled q''_s .

Comments:

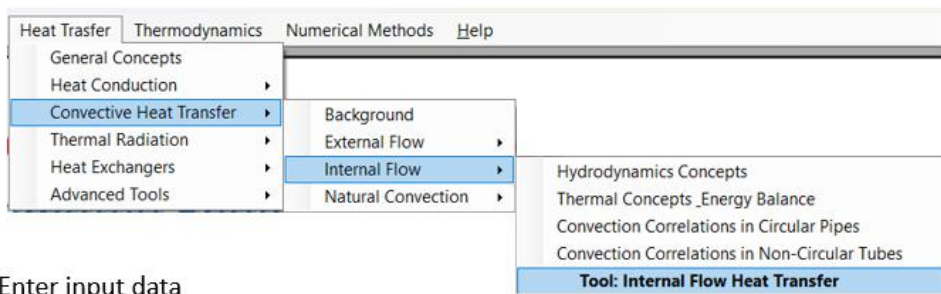
1. The Dittus-Boelter correlation is used to be consistent with the approach in the textbook.
2. The results show that roughly 900 W of heat loss from the duct in the basement.
3. The air exit temperature is about 85 °C; the duct surface temperature at the exit is 56.4 °C; the surface heat flux at the exit is 338 W/m².

Example – Flow in a rectangular duct with specified properties.

Air at 12°C enters a 2-m-long smooth rectangular duct with a cross-section of 75 mm x 150 mm. The duct is maintained at a constant surface temperature of 127 °C. The air mass flow rate is 0.10 kg/s. For air at 350 K and 1atm: $\rho=0.995 \text{ kg/m}^3$, $C_p=1009 \text{ J/kg}\cdot\text{K}$, $\mu=0.0000208 \text{ kg/m}\cdot\text{s}$, $k=0.030 \text{ W/m}\cdot\text{K}$, (Use Dittus-Boelter correlation and ignore thermal entry effects).

$Pr = 0.70$. Determine the

- (a) hydraulic diameter for the duct,
- (b) Reynolds number,
- (c) Nusselt number,
- (d) convection heat transfer coefficient,
- (e) air outlet temperature,



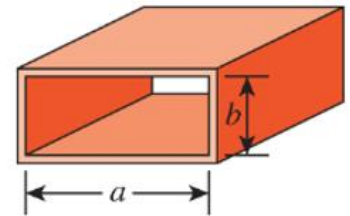
Enter input data

- Select "Enter Properties" and enter values.

The 'Fluid Properties' dialog box has two radio buttons: 'Select Fluid' and 'Enter Properties' (which is selected). Below the radio buttons is a dropdown menu for 'Air'. Under 'Pressure', the value '1.000' is entered with a unit dropdown set to 'atm'. There are two rows of property inputs: the first row has ρ (0.995 kg/m³) and C_p (1009 J/kg·K); the second row has k (0.03 W/m·K) and μ (0.0000208 Pa·s).

- Change to "Non-Circular" option.
- A form will open up to enter details

The 'Geometric Information' dialog box has two radio buttons: 'Circular' and 'Non-Circular' (which is selected). Below the radio buttons are two input fields: 'Dh' (hydraulic diameter) with a unit dropdown set to 'cm', and 'L' (duct length) with a unit dropdown set to 'm'.



- Choose rectangular cross-section and enter 15 mm and 7.5 mm for a and b .

- Click OK. The form will close, and D_h will be entered in the previous panel



Geometric Information

☐ Circular ☒ Non-Circular

D_h 10.0000 cm L 2 m

frmConvInt_Shape

☐ θ deg
☐ s cm
☒ a mm b mm
☐ $\frac{a}{b} > 8$
☐ $\frac{a}{b} = 1$
☐ $\frac{a}{b} = 1$
☐ $\frac{a}{b} = 1$
☐ $\frac{a}{b} = 1$

OK

- Enter 2 m for the length.

Inlet Flow Information

☒ $\dot{m}$ 0.1 kg/s
☐ $\dot{V}$ cfm
☐ U_{in} m/s

- Enter the 2 °C for Inlet Temperature.
- Select the "Uniform Surface Temperature" option and enter 127 °C

Thermal Conditions

T_{in} 12 C

☐ Uniform q''_s W/m²
☒ Uniform T_s 127 C
☐ Uniform T_{inf} C

h_{ext} W/m².K
 K_w W/m.K
 th_w mm

- Make sure Thermal Entry Effects is unchecked.

☐ Include Thermal Entry Effects

- In the “Correlation Corner” panel, check ON Dittus-Boelter.

Correlation Corner

Turbulent Flow - Fully Developed

☐ Gnielinsky Correlation (Smooth Wall)
 ☐ Rough wall
 ☐ Adjust Properties for Twall Effects

☒ Dittus-Boelter Correlation (Smooth Wall)

☐ Bhatti-Shah (Rough Wall)

Show Correlations

The finished form is shown below:

Fluid Properties

☐ Select Fluid
 ☒ Enter Properties

Air

Pressure 1.000 atm

ρ 0.9950 kg/m³
 C_p 1009.0000 J/kg.K

k 0.0300 W/m.K
 μ 2.080E-05 Pa.s

Geometric Information

☐ Circular
 ☒ Non-Circular

Dh 10.0000 cm L 2 m

Inlet Flow Information

☒ mdot 0.1 kg/s
 ☐ Vdot 212.95 cfm
 ☐ Uin 8.93 m/s

Thermal Conditions

Tin 12 C

☐ Uniform q''_s
☒ Uniform Ts 127 C
 ☐ Uniform Tinf

h_{ext}
 K_w
 th_w

☐ Include Thermal Entry Effects

Results

Ac 1.13E-02 m²
 Per 0.4500 m

Re 42735 Pr 0.70

Flow Regime Turbulent

Flow Conditions

Velocity Fully Developed Velocity

Temperature Fully Developed Thermal

Average

Nu_avg 100.9923
 h_{avg} 30.3 W/m².K
 qtot 2747.8 W

Exit

Nu_o 100.9923
 h_o 30.3 W/m².K
 Tmo 39.2 C
 Tso 127.0 C
 $q''_{s,o}$ 2659.1 W/m²

Local Results

x/L 0.5
 Nu 100.9923
 Tm 26.5 C
 Ts 127.0 C
 h 30.3 W/m².K
 $q''_{s,x}$ 3043.9 W/m²

Correlation Corner

Turbulent Flow - Fully Developed

☐ Gnielinsky Correlation (Smooth Wall)
 ☐ Rough wall
 ☐ Adjust Properties for Twall Effects

☒ Dittus-Boelter Correlation (Smooth Wall)

☐ Bhatti-Shah (Rough Wall)

Show Correlations

Results - Pressure Drop and Friction

Friction Factor

☒ Haaland Equation
 ☐ Swamee-Jain Equation
 ☐ Colebrook Equation

Darcy Friction Factor, f 0.0217
 Friction Head Loss, hf 1.7656 m
 Pressure Drop 0.0025 psi

DZ 0.00 m

Update

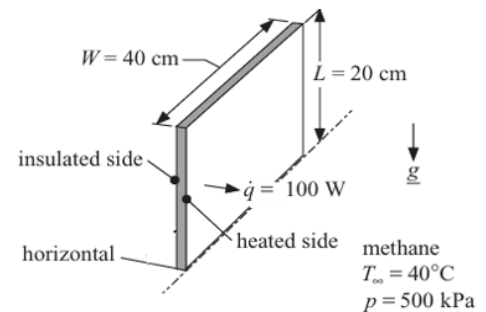
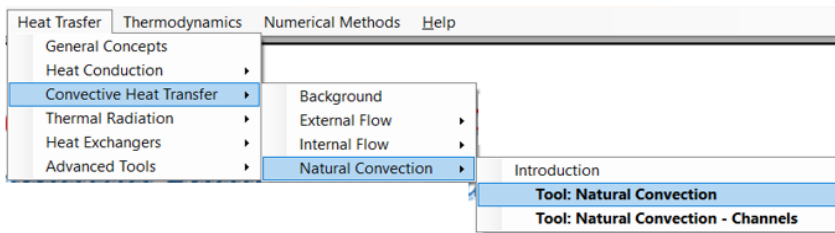
Diagram

$T_s > T_{in}$
 q''_s
 $T(r, 0) = T_s$
 $T(r, 0)$
 Thermal Entry Region
 Fully Developed

HT-16: Natural Convection over Bodies

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

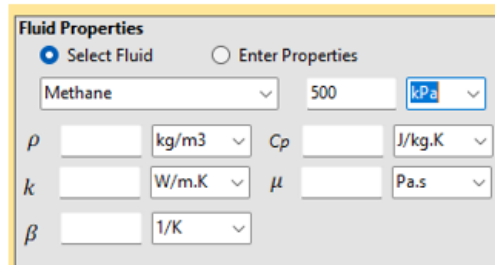
A rectangular plate heater is placed in the ullage space of a fuel tank on a military aircraft (as shown below). One side of the heater is insulated, and the other is heated. The heater is oriented vertically with respect to gravity and achieves a nearly uniform temperature. The length of the heater is $L = 20$ cm, and the width is $W = 40$ cm. The plate is exposed to fuel that has properties consistent with methane at $T_\infty = 40^\circ\text{C}$. Assuming a heater power of 100 W, determine the surface temperature of the heater for fuel at $p = 500$ kPa, with the plate tilted relative to the horizontal at $\zeta = 60^\circ$ (where 0° is horizontal, with the heated surface facing upward). a) neglect radiation, b) use emissivity = 0.75.



Enter input data

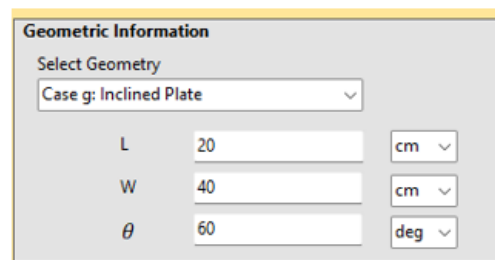
Enter Fluid Properties:

- Select “Methane” from Fluid Library.
- Change the pressure to 500 and select kPa for unit.



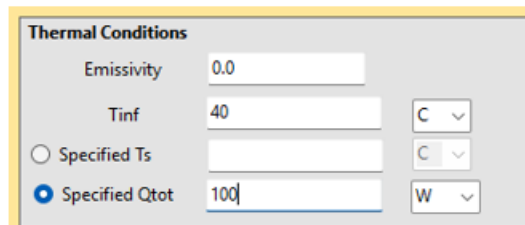
Geometric Information:

- Select “Case g” Inclined Plate.
- Enter the plate dimensions and tilt angle.



Thermal Conditions:

- Enter 40 °C for ambient temperature
- Change to Specified Q_{tot} (total heat rate)
- Enter 100 W for Plate Heat Rate



Click “Update” to Solve.

Completed Form:

Natural Convection

Fluid Properties
☒ Select Fluid ☐ Enter Properties
Methane 500 kPa
 ρ 2.7880 kg/m³ C_p 2360.5352 J/kg.K
 k 0.0411 W/m.K μ 1.260E-05 Pa.s
 β 0.0029 1/K

Parameters
As 8.0000E-02 m²
Per 1.2000 m
Grashof No N/A
Rayleigh No N/A ☐ Fix
Prandtl No 0.72 ☐ Fix

Geometric Information
Select Geometry
Case g: Inclined Plate
L 20 cm
W 40 cm
 θ 60 deg

Gravitational Acceleration
Gravity 9.81 m/s²

Thermal Conditions
Emissivity 0.0
Tinf 40 C
☐ Specified Ts 106.9 C
☒ Specified Qtot 100.0 W

Results
Nu_avg 0.000
h_avg 18.7 W/m².K
Qconv 99.95 W
Qrad 0.00 W
Qtot 99.95 W
U_char 0.62 m/s

Correlation Selector
Vertical Flat Plate (a)
☒ Eq: A1/A2-Churchill Chu
☐ Eq: A.3
☐ Eq: A.4-Nellis, Klein (EES)
Horizontal Plate Upward (b)
☒ Eq: A.5
☐ Eq: A.6-Nellis, Klein (EES)
Horizontal Plate Downward (c)
☒ Eq: A.7
☐ Eq: A.8-Nellis, Klein (EES)
Vertical Cylinder (d)
CV1
Vertical Wall with Correction
Horizontal Cylinder (e)
☒ CH11/CH12 (EES)
☐ CH11-Churchill Chu
Sphere (f)
☒ Sparrow-Gregg
Inclined Plate (g)
☒ Eq: A.13
Raithby and Holland method

Update

- Surface temperature is calculated to be 106.8 °C. This agrees very closely with the value of 106.1 °C, reported in Nelis and Kline using EES software. The small difference is due to the material properties values used.

b) Include radiation with an emissivity of 0.75.

Thermal Conditions:

- Change Emissivity to 0.75

Click "Update" to Solve.

Thermal Conditions
Emissivity 0.75
Tinf 40 C
☐ Specified Ts 91.8 C
☒ Specified Qtot 100.0 W

Natural Convection

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Methane 500 kPa

ρ 2.8526 kg/m³ C_p 2338.7798 J/kg.K

k 0.0399 W/m.K μ 1.236E-05 Pa.s

β 0.0030 1/K

Parameters

As 8.0000E-02 m²

Per 1.2000 m

Grashof No N/A

Rayleigh No N/A ☐ Fix

Prandtl No 0.72 ☐ Fix

Geometric Information

Select Geometry

Case g: Inclined Plate

L .2 m

W .4 m

θ 60 deg

Gravitational Acceleration

Gravity 9.81 m/s²

Thermal Conditions

Emissivity 0.75

Tinf 40 C

☐ Specified Ts 91.8 C

☒ Specified Qtot 100.0 W

Results

Nu_avg 0.000

h_avg 17.5 W/m².K

Qconv 72.34 W

Qrad 27.60 W

Qtot 99.94 W

U_char 0.55 m/s

Correlation Selector

Vertical Flat Plate (a)

☒ Eq: A1/A2-Churchill Chu

☐ Eq: A.3

☐ Eq: A.4-Nellis, Klein (EES)

Horizontal Plate Upward (b)

☒ Eq: A.5

☐ Eq: A.6-Nellis, Klein (EES)

Horizontal Plate Downward (c)

☒ Eq: A.7

☐ Eq: A.8-Nellis, Klein (EES)

Vertical Cylinder (d)

CV1

Vertical Wall with Correction

Horizontal Cylinder (e)

☒ CH11/CH12 (EES)

☐ CH11-Churchill Chu

Sphere (f)

☐ Sparrow-Gregg

Inclined Plate (g)

☒ Eq: A.13

Raithby and Holland method

Case a: Vertical Plate Heated or Cooled

Case b: Horizontal Plate Upward Heated or Downward Cooled

Case c: Horizontal Plate Downward Heated or Upward Cooled

Case d: Vertical Cylinder Heated or Cooled

Case e: Long Horizontal Cylinder Heated or Cooled

Case f: Long Horizontal Cylinder Heated or Cooled

Case g: Inclined Plate

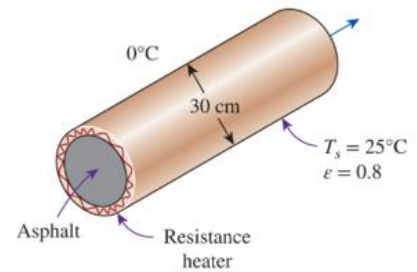
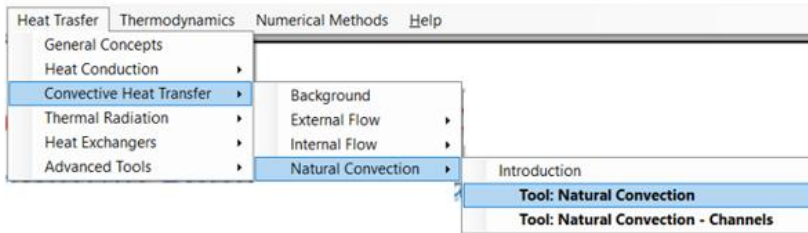
Update

Note:

- The surface temperature is reduced by 15 °C
- The rate of heat transfer by radiation is 27.6 W.

Example – Horizontal Cylinder – [Source: Cengel-Ghagar Problem 9.60]

Thick fluids, such as asphalt and waxes, and the pipes through which they flow are often heated to reduce their viscosity and thus lower pumping costs. Consider the flow of such a fluid through a 100-m-long pipe of outer diameter 30 cm in calm, ambient air at 0 °C. The pipe is electrically heated, and a thermostat maintains the pipe's outer surface temperature at 25°C. Assuming the emissivity of the pipe's outer surface is 0.8, determine the power rating of the electric resistance heater, in kW, that should be used.



Enter Fluid Properties:

- Select “Air” from Fluid Library.
- Keep pressure at 1 atm

The screenshot shows the 'Fluid Properties' dialog box. The 'Select Fluid' radio button is selected, and 'Air' is chosen from the fluid library. The pressure is set to 1 atm. Other properties like density (ρ), specific heat (C_p), thermal conductivity (k), viscosity (μ), and coefficient of thermal expansion (β) are shown with empty input fields and units.

Geometric Information:

- Select “Case e” Long Cylinder.
- Enter 100 m for length.
- Enter 30 cm for Diameter.

The screenshot shows the 'Geometric Information' dialog box. The 'Select Geometry' dropdown is set to 'Case e: Long Horizontal Cylinder'. The length (L) is entered as 100 m and the diameter (D) is entered as 30 cm.

Thermal Conditions:

- Enter 0.8 for Emissivity.
- Enter 0 °C for ambient temperature.
- Enter 25 °C for Specified surface temperature

The screenshot shows the 'Thermal Conditions' dialog box. Emissivity is entered as 0.8. T_{inf} is entered as 0 C. 'Specified T_s ' is selected and entered as 25 C. 'Specified Q_{tot} ' is shown as 20329.4 W.

Click “Update” to Solve.

Natural Convection

Fluid Properties

☒ Select Fluid ☐ Enter Properties

Air 1 atm

ρ 1.2284 kg/m³ C_p 1006.7130 J/kg.K

k 0.0252 W/m.K μ 1.774E-05 Pa.s

β 0.0035 1/K

Parameters

As 9.4248E01 m²

Per 1.2000 m

Grashof No 1.0930e08

Rayleigh No 7.7683e07 ☐ Fix

Prandtl No 0.71 ☐ Fix

Geometric Information

Select Geometry

Case e: Long Horizontal Cylinder

L 100 m

D 30 cm

Gravitational Acceleration

Gravity 9.81 m/s²

Thermal Conditions

Emissivity 0.8

Tinf 0 C

☒ Specified Ts 25 C

☐ Specified Qtot 20329.4 W

Results

Nu_avg 52.374

h_avg 4.4 W/m².K

Qconv 10346.05 W

Qrad 9983.38 W

Qtot 20329.43 W

U_char 0.51 m/s

Correlation Selector

Vertical Flat Plate (a)

☒ Eq: A1/A2-Churchill Chu

☐ Eq: A.3

☐ Eq: A.4-Nellis, Klein (EES)

Horizontal Plate Upward (b)

☒ Eq: A.5

☐ Eq: A.6-Nellis, Klein (EES)

Horizontal Plate Downward (c)

☒ Eq: A.7

☐ Eq: A.8-Nellis, Klein (EES)

Vertical Cylinder (d)

CV1

Vertical Wall with Correction

Horizontal Cylinder (e)

☒ CH11/CH12 (EES)

☐ CH11-Churchill Chu

Sphere (f)

☒ Sparrow-Gregg

Inclined Plate (g)

☒ Eq: A.13

Raithby and Holland method

Case a: Vertical Plate Heated or Cooled

Case b: Horizontal Plate Upward Heated or Downward Cooled

Case c: Horizontal Plate Downward Heated or Upward Cooled

Case d: Vertical Cylinder Heated or Cooled

Case e: Long Horizontal Cylinder Heated or Cooled

Case f: Long Horizontal Cylinder Heated or Cooled

Case g: Inclined Plate

Update

Note:

- The total heat rate is 20.33 kW.
- The average heat transfer coefficient is 4.4 W/m².K.

HT-17: Natural Convection Vertical Channels

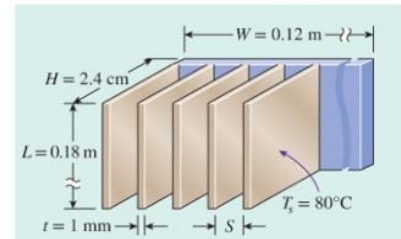
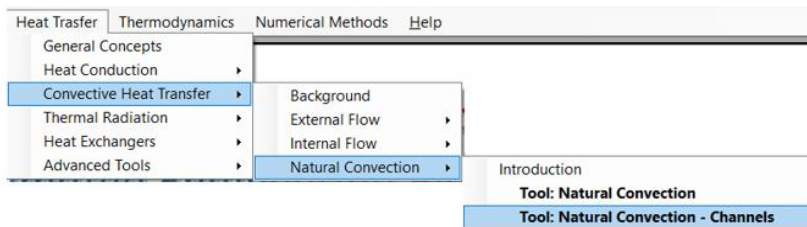
Example – Heat Sink Fin Spacing – [Source: Cengel-Ghagar Example 9.3]

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 rectangular-profile fins. The fins are 1 mm thick, 18 cm long in the vertical direction, and 2.4 cm high from the base. Assuming the base temperature of 80 °C.

- Determine the rate of heat transfer by natural convection from the heat sink using 20 fins.
- Determine the rate of heat transfer by natural convection from the heat sink using optimal fin spacing.
- 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:

Note:

- The user can choose to provide number of fins or fin spacing.
- 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. S_{max} results in max heat transfer per fin.

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:



☒ Constant Ts	80	C
Emissivity	0.25	

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.

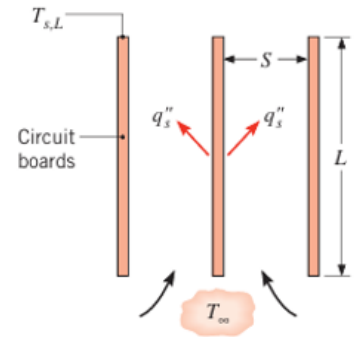
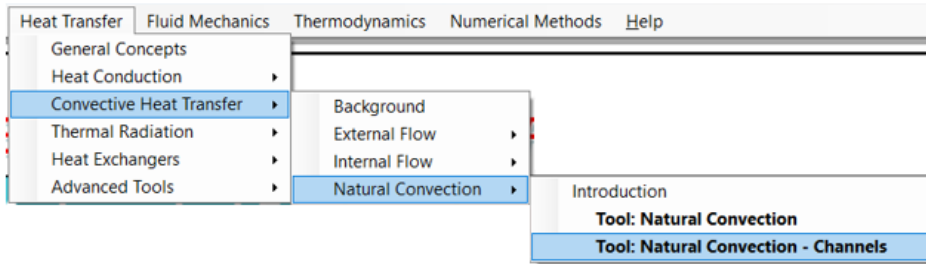
Results		
Ra	1301	
Nu	1.3066	
h	4.9	W/m2.K
Sopt	7.5056	mm
Smax	12.8346	mm

Qcnv	29.9	W
Qrad	5.9	W
Qtot	35.8	W
Tmax		C

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 $T_\infty = 17^\circ\text{C}$. 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, 1.5 mm thick, and spacing $S = 25$ mm. Assuming each board is populated on both sides and dissipates 88 W, determine the maximum board temperature and total heat dissipation of this unit.



Calculate PCB surface heat flux.

$$Q_{\text{board}} = 2 \cdot (q''_s \cdot A)$$

$$q''_s = \frac{Q_{\text{board}}}{2A} = \frac{Q_{\text{board}}}{2(L \cdot H)}$$

$$q''_s = \frac{88}{2(0.4 \times 0.4)} = 275 \text{ W/m}^2$$

- Select “Air” from Fluid Library with pressure kept at 1 atm

- Enter 60 cm for the width.
- Enter 40 cm for the length and the height of the fins.
- Enter 1.5 mm for the Thickness.
- Enter 25 mm for the spacing.
- Enter 17 °C for ambient temperature.
- Enter 275 W/m² for the heat flux.

Fluid Properties

☒ Select Fluid
 ☐ Enter Properties

Air

Pressure

1.000

atm

ρ

kg/m³

C_p

J/kg.K

k

W/m.K

μ

Pa.s

Input Parameters

W

60

cm

L

40

cm

H

40

cm

t

1.5

mm

☒ S

25

mm

☐ N

T_{inf}

17

C

☐ Constant T_s

C

Emissivity

0.0

☒ Constant q''_s

275

W/m²

☐ Insulate Back

Click Update to solve.

Natural Convection - Vertical Channels

☒ Select Fluid ☐ Enter Properties

Air Pressure 1.000 atm

ρ 1.1011 kg/m³ C_p 1007.7243 J/kg.K

k 0.0276 W/m.K μ 1.931E-05 Pa.s

Input Parameters

W 60 cm

L 40 cm

H 40 cm

t 1.5 mm

☒ S 25 mm

☐ N 23

Tinf 17 C

☐ Constant Ts

Emissivity 0.0

☒ Constant q"s 275 W/m²

☐ Insulate Back

Results

Ra 275930

Nu 4.3241

h 4.8 W/m².K

Sopt 7.5325 mm

Smax 35.9301 mm

Qcnv W

Qrad W

Qtot 2024.0 W

Tmax 74.5 C

Update

Show Tutorial

a) Constant T_s

b) Constant q_s

1. The maximum board temperature will be 74.5 °C.
2. The total heat rate will be 2,024 W.

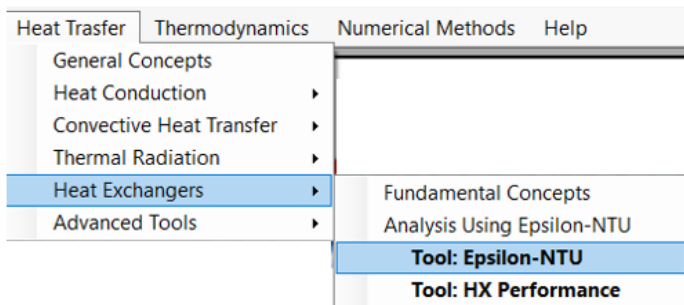
HT-18: Heat Exchanger Epsilon/NTU Calculator

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).

Input

d) Shell & Tube: n Shell pass and 2n, 4n, ... tube pass

Cr

No. of Shell passes

☒ NTU Given

☐ Epsilon Given

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	1.7885	0.65
Parallel Flow	0.75	No Solution	0.65

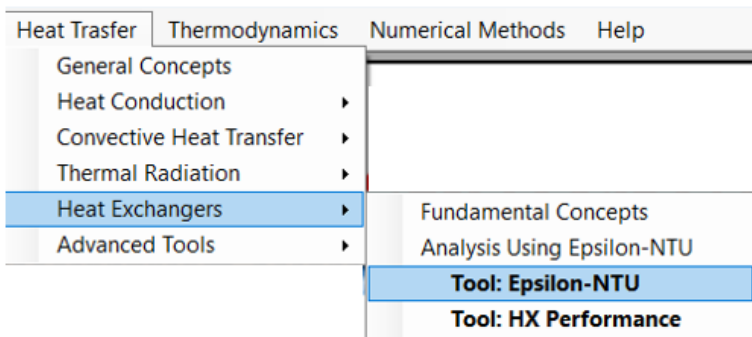
Note: Epsilon-NTU relations always provide an effectiveness value for all positive NTUs. However, as shown in the last row, not every positive epsilon yields a valid NTU.

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 flow rate of 0.06 m³/s. The heat exchanger's conductance has been calculated to be 58.4 W/K using 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).

Input

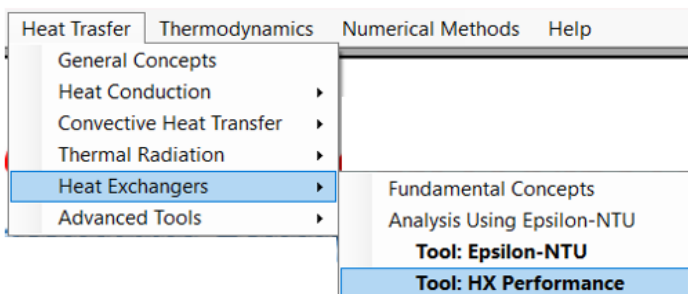
d) Shell & Tube: n Shell pass and 2n, 4n, ... tube pass

Cr

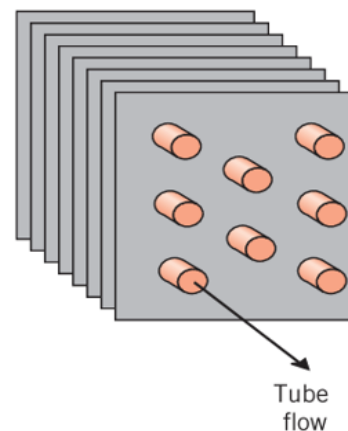
No. of Shell passes

☒ NTU Given

☐ Epsilon Given



Enter input data



Hot Side

Cold Side

Hot Fluid

Properties
☒ Select Fluid ☐ Enter Props
 Water
 ρ kg/m³
 C_p J/kg.K

Inlet Conditions
☒ mdotH 0.03 kg/s
☐ VdotH cfm
 Thot_in 60 C

Cold Fluid

Properties
☒ Select Fluid ☐ Enter Props
 Air
 ρ kg/m³
 C_p J/kg.K

Inlet Conditions
☐ mdotC kg/s
☒ VdotC 0.06 m³/s
 Tcold_in 20 C

Select HX Type from list → e) Cross Flow: both fluid unmixed

Select UA for input and enter the value →
☒ UA 58.40 W/K
☐ Thot_out C
☐ Tcold_out C
☐ Q W

Click "Solve".

Hot Fluid

Properties
☒ Select Fluid ☐ Enter Props
 Water
 ρ 979.50 kg/m³
 C_p 4187.96 J/kg.K

Inlet Conditions
☒ mdotH 0.03 kg/s
☐ VdotH cfm
 Thot_in 60 C

Cold Fluid

Properties
☒ Select Fluid ☐ Enter Props
 Air
 ρ 1.19 kg/m³
 C_p 1006.86 J/kg.K

Inlet Conditions
☐ mdotC kg/s
☒ VdotC 0.06 m³/s
 Tcold_in 20 C

HX Details

e) Cross Flow: both fluid unmixed

☒ UA 58.40 W/K
☐ Thot_out 49.1 C
☐ Tcold_out 38.9 C
☐ Q 1365.2 W

Qmax 2883.7 W
 Cr 0.57
 Epsilon 0.47
 NTU 0.81

Solve

$$C_{max} = \max(C_h, C_c)$$

$$C_{min} = \min(C_h, C_c)$$

$$C_r = \frac{C_{min}}{C_{max}} \leq 1$$

$$q_{max} = C_{min}(T_{h,i} - T_{c,i})$$

$$\epsilon = \frac{q}{q_{max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{min}(T_{h,i} - T_{c,i})}$$

$$NTU = \frac{UA}{C_{min}}$$

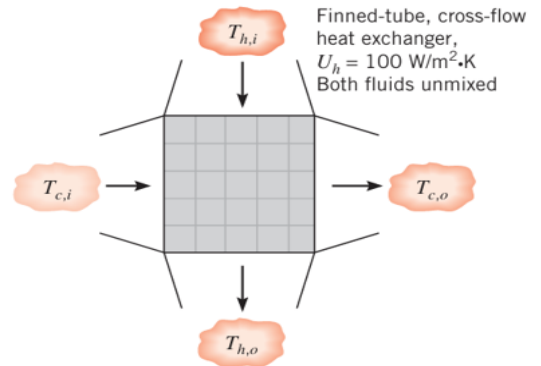
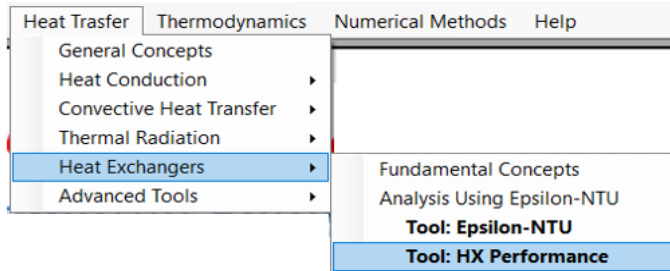
Show Tutorial

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 ($c_p = 1080 \text{ J/kg} \cdot \text{K}$), which enter a finned-tube cross-flow heat exchanger at 300°C with a flow rate of 1.75 kg/s , leave at 100°C and 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 $U_h = 100 \text{ W/m}^2\cdot\text{K}$. Determine the required gas-side surface area A_h and the outlet temperature of water.

Solution:



Enter input data

Hot Side

Cold Side

Select user-defined to enter c_p manually

Enter $\dot{m}_h$

Enter inlet temp

Select HX Type from list

Select outlet temp for the gas

Select water

Enter $\dot{m}_c$

Enter inlet temp

Hot Fluid Properties

- ☐ Select Fluid
- ☒ Enter Props
- Gas
- ρ kg/m³
- C_p 1080 J/kg.K

Inlet Conditions

- ☒ mdotH 1.75 kg/s
- ☐ VdotH cfm
- T_{hot_in} 300 C

Cold Fluid Properties

- ☒ Select Fluid
- ☐ Enter Props
- Water
- ρ kg/m³
- C_p J/kg.K

Inlet Conditions

- ☒ mdotC 1 kg/s
- ☐ VdotC cfm
- T_{cold_in} 35 C

HX Details

- e) Cross Flow: both fluid unmixed

Output Options

- ☐ UA W/K
- ☒ T_{hot_out} 100 C
- ☐ T_{cold_out} C
- ☐ Q W

Click "Solve".

Heat Exchanger Performance Analysis

Hot Fluid

Properties: ☐ Select Fluid ☒ Enter Props

Gas

ρ 0.00 kg/m³

C_p 1080.00 J/kg.K

Inlet Conditions

☒ mdotH 1.75 kg/s

☐ VdotH cfm

Thot_in 300 C

Cold Fluid

Properties: ☒ Select Fluid ☐ Enter Props

Water

ρ 993.78 kg/m³

C_p 4178.00 J/kg.K

Inlet Conditions

☒ mdotC 1 kg/s

☐ VdotC cfm

Tcold_in 35 C

HX Details

e) Cross Flow: both fluid unmixed

☐ UA 3833.87 W/K **1**

☒ Thot_out 100.0 C **2**

☐ Tcold_out 125.5 C **3**

☐ Q 378000.0 W **3**

Qmax 500850.0 W **4**

Cr 0.45 **5**

Epsilon 0.75 **6**

NTU 2.03 **7**

Solve

Diagram:

Hot In: $T_{h,i}, C_h$

Hot Out: $T_{h,o}, C_h$

Cold In: $T_{c,i}, C_c$

Cold Out: $T_{c,o}, C_c$

HX

Equations:

$$C_{max} = \max(C_h, C_c)$$

$$C_{min} = \min(C_h, C_c)$$

$$C_r = \frac{C_{min}}{C_{max}} \leq 1$$

$$q_{max} = C_{min}(T_{h,i} - T_{c,i})$$

$$\epsilon = \frac{q}{q_{max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{min}(T_{h,i} - T_{c,i})}$$

$$NTU = \frac{UA}{C_{min}}$$

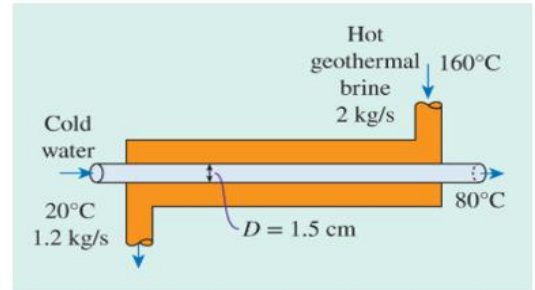
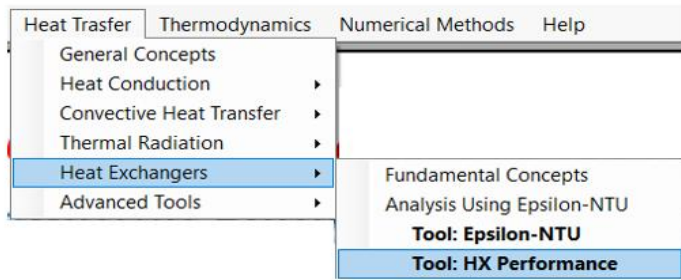
Show Tutorial

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:
$$A_h = \frac{UA}{U_h} = \frac{3833}{100} \rightarrow A_h = 38.33 \text{ m}^2$$
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 used to heat water from 20 °C to 80 °C at a rate of 1.2 kg/s. The heating is to be accomplished using geothermal water available at 160 °C with 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

Hot Side

Select user-defined to enter c_p manually

Enter $\dot{m}_h$

Enter inlet temp

Hot Fluid

Properties
☐ Select Fluid ☒ Enter Props

Geothermal H2O

ρ kg/m³

C_p 4310 J/kg.K

Inlet Conditions
☒ $\dot{m}_{dotH}$ 2 kg/s
☐ $\dot{V}_{dotH}$ cfm

T_{hot_in} 160 C

Cold Side

Select water

Enter $\dot{m}_c$

Enter inlet temp

Cold Fluid

Properties
☒ Select Fluid ☐ Enter Props

Water

ρ kg/m³

C_p J/kg.K

Inlet Conditions
☒ $\dot{m}_{dotC}$ 1.2 kg/s
☐ $\dot{V}_{dotC}$ cfm

T_{cold_in} 20 C

Select HX Type from list

Select outlet temp for water

HX Details

b) Counter Flow

☐ UA W/K

☐ T_{hot_out} C

☒ T_{cold_out} 80 C

☐ Q W

Click "Solve".

Hot Fluid

Properties: ☐ Select Fluid ☒ Enter Props
 Geothermal H2O
 ρ 0.00 kg/m3
 C_p 4310.00 J/kg.K

Inlet Conditions:
☒ mdotH 2 kg/s
☐ VdotH cfm
 T_{hot_in} 160 C

Cold Fluid

Properties: ☒ Select Fluid ☐ Enter Props
 Water
 ρ 998.37 kg/m3
 C_p 4182.11 J/kg.K

Inlet Conditions:
☒ mdotC 1.2 kg/s
☐ VdotC cfm
 T_{cold_in} 20 C

HX Details
 b) Counter Flow

☐ UA 3274.19 W/K (1)
☐ T_{hot_out} 125.1 C (2)
☒ T_{cold_out} 80.0 C (3)
☐ Q 301.1 kW (3)

Qmax 702.6 kW (4)
 Cr 0.58 (5)
 Epsilon 0.43 (6)
 NTU 0.65 (7)

Solve

Diagram and Equations:

Central oval: HX

Hot Fluid Inlet: $T_{h,i}, C_h$ (red arrow)

Hot Fluid Outlet: $T_{h,o}, C_h$ (pink arrow)

Cold Fluid Inlet: $T_{c,i}, C_c$ (blue arrow)

Cold Fluid Outlet: $T_{c,o}, C_c$ (light blue arrow)

Equations:

$$C_{max} = \max(C_h, C_c)$$

$$C_{min} = \min(C_h, C_c)$$

$$C_r = \frac{C_{min}}{C_{max}} \leq 1$$

$$q_{max} = C_{min}(T_{h,i} - T_{c,i})$$

$$\epsilon = \frac{q}{q_{max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{min}(T_{h,i} - T_{c,i})}$$

$$NTU = \frac{UA}{C_{min}}$$

Show Tutorial

1. The UA is calculated to be 3274.2 W/K.

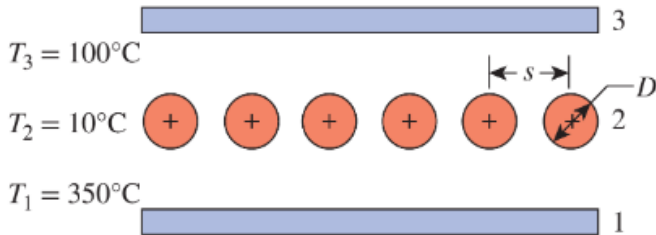
Since U is given in the problem statement to be 640 W/m².K, The length can be found:

$$A_s = \pi DL = \frac{UA}{U_h} \Rightarrow L = \frac{UA}{(U)(\pi D)} = \frac{(3274.2)}{(640)(\pi)(0.015)} \Rightarrow L = 108.6 \text{ m}$$

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.

Example – View factors from plate to cylinders – [Source: Cengel-Ghajar Problem 13-37]

A row of tubes, equally spaced at a distance of 5 cm and with a diameter of 2.5 cm, is positioned between two large parallel plates. The surface temperature of the tubes is constant at 10 °C, and the top and bottom plates are at constant temperatures of 100 °C and 350 °C, respectively. If the surfaces behave as blackbodies, determine the net radiation heat flux leaving the bottom plate.



Solution:

From energy balance, the net radiation heat flux leaving the bottom plate (surface 1) is:

$$\dot{q}_1 = \dot{q}_{12} + \dot{q}_{13} = F_{12}\sigma(T_1^4 - T_2^4) + F_{13}\sigma(T_1^4 - T_3^4)$$

With $F_{13} = 1 - F_{12}$, we have:

$$\dot{q}_1 = F_{12}\sigma(T_1^4 - T_2^4) + (1 - F_{12})\sigma(T_1^4 - T_3^4)$$

The View Factor automated tool can be used to determine F_{12}

1. Choose configuration →

2. Enter the diameter and spacing →

3. Click Update →

4. View Factor is calculated → 0.6576

$$\begin{aligned}
 \dot{q}_1 &= \sigma [F_{12}(T_1^4 - T_2^4) + (1 - F_{12})(T_1^4 - T_3^4)] \\
 &= (5.67 \times 10^{-8} \text{ W/m}^2 \cdot \text{K}^4) [(0.6576)(623^4 - 283^4) \text{ K}^4 + (1 - 0.6576)(623^4 - 373^4) \text{ K}^4] \\
 &= 7927 \text{ W/m}^2
 \end{aligned}$$