

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 Plane Wall Cylinder Sphere
No. of Layers

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*.

Right BC (x=0)
 Convection h W/m2.K T_{amb} C
 Fixed Temp T_s C
 Heat FLux q”_s W/m2

Right BC (x=L)
 Convection h W/m2.K T_{amb} C
 Fixed Temp T_s C
 Heat FLux q”_s W/m2

3. Geometric Parameters

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

Cylinder: Inner Radius; Cylinder Length

Sphere: Inner Radius

X0 cm L m
W 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
Layer_1/Layer_2 Value C.in2/W

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 Thickness 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:

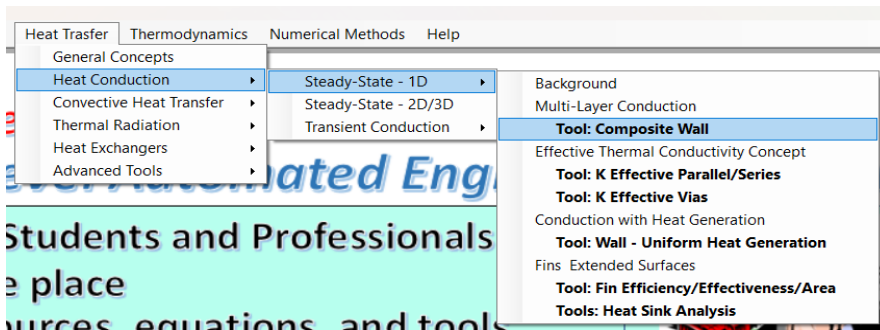
A 0.8 m by 1.5 m glass window (thermal conductivity of 0.78 W/m.K) with a thickness of 8 mm. Assume the interior of the room is maintained at 20 °C with natural convection heat transfer coefficient of 10 W/m².K, and the external ambient at -10 °C with heat transfer coefficient of 40 W/m².K (which include thermal radiation effects). Determine the steady rate of heat transfer through this window and the temperature of its inner surface for the following two conditions:

- Single pane with a thickness of 8 mm.
- Double-pane consisting of two 4-mm thick glass layers separated by 10-mm thick stagnant airspace.

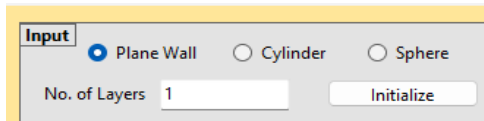
Solution:

- Single pane

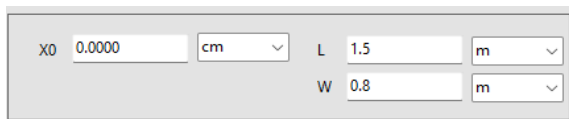
Open “Heat Conduction Composite Systems” Panel:



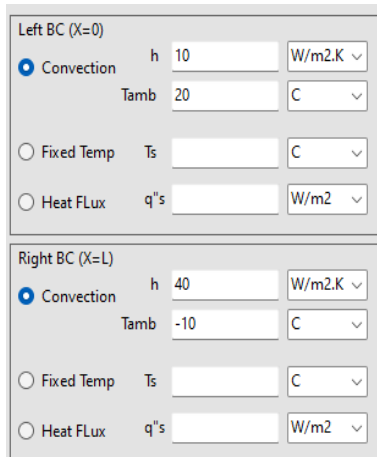
Select *Plane Wall* (default) and set number of layers to “1” (for Case-a) and click *Initialize*.



Set Width and Length to 0.8 m and 1.5 m



Set values for *Left BC* ($h=10$ W/m².K, $T_{amb}=20$ °C), and *Right BC* ($h=40$ W/m².K, $T_{amb}=-10$ °C)



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

Heat Conduction - Composit Systems

Input

Plane Wall Cylinder Sphere

No. of Layers: 1

Right BC (x=0):

Convection h: 10 W/m².K Tamb: 20 C

Fixed Temp Ts: C

Heat FLux q"s: W/m²

Right BC (x=L):

Convection h: 40 W/m².K Tamb: -10 C

Fixed Temp Ts: C

Heat FLux q"s: W/m²

X0: 0.0000 cm L: 1.00 m W: 1.00 m

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

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

Layer Name	Therm. Cond.	Thickness
Layer_1	0.78	8

Results

q"tot: 221.8009 W/m²

qtot: 221.8009 W

R"tot: 0.1353 C.m²/W

U: 7.3934 W/m².K

Layers: X: mm T: C Rth: C.m²/W

X Left	T Left	Rth	X Right	T Right
0.000	-2.18	0.010	8.000	-4.45

The diagram shows a cross-section of a wall with multiple layers (L1, L2, L3, ..., LN). On the left, there is convection with heat transfer coefficient h1 and ambient temperature T∞,1. On the right, there is convection with heat transfer coefficient h2 and ambient temperature T∞,2. The wall surface temperatures are T0 and TN. Thermal resistances are shown as Rcnv1, R1, RN, and Rcnv2.

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:

Heat Conduction - Composit Systems

Input

Plane Wall Cylinder Sphere

No. of Layers: 3 Initialize

Right BC (x=0): Convection h: 10 W/m².K Tamb: 20 C

Fixed Temp Ts: C

Heat FLux q''s: W/m²

Right BC (x=L): Convection h: 40 W/m².K Tamb: -10 C

Fixed Temp Ts: C

Heat FLux q''s: W/m²

X0: 0.0000 cm L: 1.5 m W: 0.8 m

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	4
Layer_2	0.026	10
Layer_3	0.78	4

Results

q''tot: 57.7065 W/m²

qtot: 69.2478 W

R''tot: 0.5199 C.m²/W

U: 1.9236 W/m².K

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

Update

Show Tutorial

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, considerably better performance with the double-pane window.

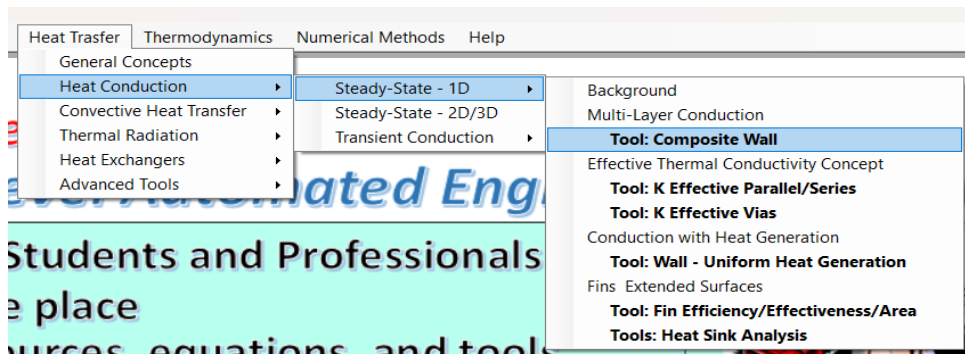
<<End-of-Tutorial>>

Example – Composite Cylinder:

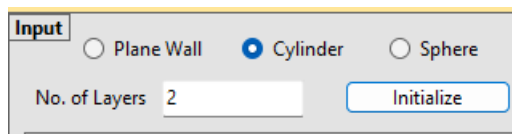
Steam at 320 °C flows in a 5-cm diameter cast iron pipe ($k=80$ W/m.K) with wall thickness of 0.25 cm. The pipe is covered with 3-cm-thick glass wool insulation ($k=0.05$ W/m.K). Assuming internal heat transfer coefficient of 60 W/m²°C and external ambient at 5 °C with heat transfer coefficient of 18 W/m²°C, determine the rate of heat loss from the pipe per unit length and temperature drops across the pipe shell and the insulation.

Solution:

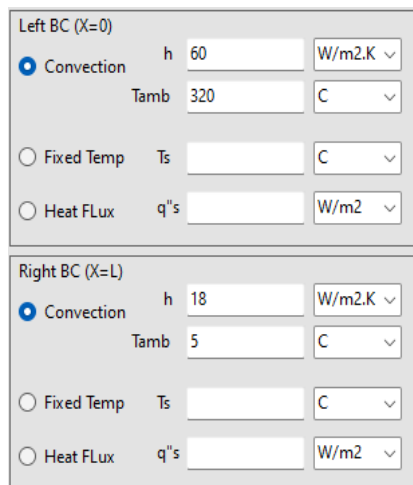
Open “Heat Conduction Composite Systems” Panel:



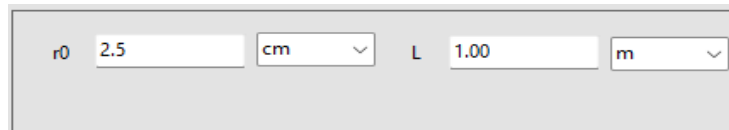
Select *Cylinder* and set number of layers to “2” and click *Initialize*.



Set values for *Left BC* ($h=60$ W/m².K, $T_{amb}=320$ °C), and *Right BC* ($h=18$ W/m².K, $T_{amb}=5$ °C)



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



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

Layers			
	Thickness	cm	k
	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
— □ ×

Input

Plane Wall
 Cylinder
 Sphere

No. of Layers: Initialize

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

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

r0: cm L: m

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

Layers

	Thickness	cm	k	W/m.K
Layer_1	80	0.25		
Layer_2	0.05	3		

Results

qr: W
R"tot: C.m2/W
R"tot: C.m2/W
U: W/m2.K

Layers					
X	cm	T	C	Rth	C.m2/W
r Left	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 $307.16 - 23.17 = 283.99 \text{ }^\circ\text{C}$.

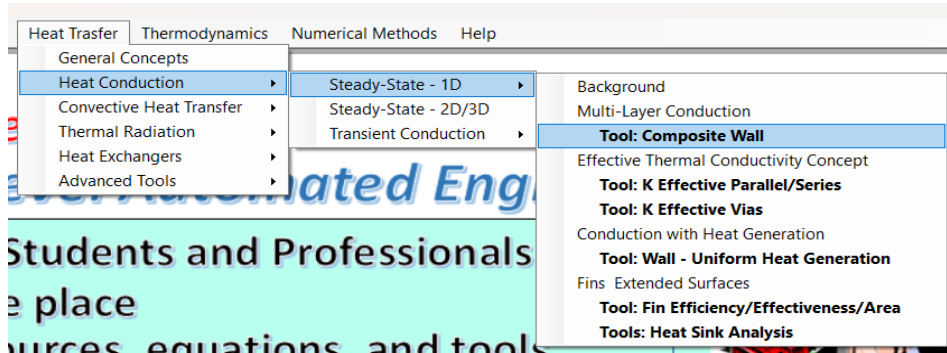
<<End-of-Tutorial>>

Example – Composite Sphere:

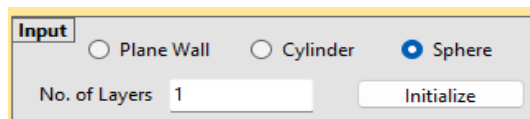
A stainless-steel spherical tank ($k=15 \text{ W/m.K}$), with 3 m internal diameter and 2 cm thickness, is used to store ice water at 0°C . If the environment is at 22°C , and assuming $80 \text{ W/m}^2\text{C}$ and $16 \text{ W/m}^2\text{C}$ for internal and external heat transfer coefficients, determine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

Solution:

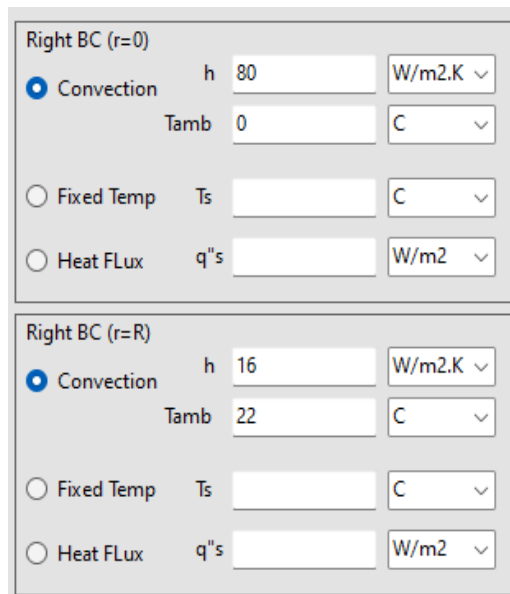
Open “Heat Conduction Composite Systems” Panel:



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



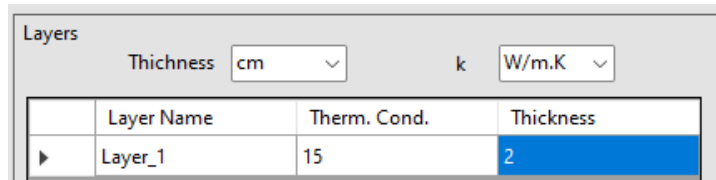
Set values for *Left BC* ($h=80 \text{ W/m}^2.K$, $T_{amb}=0^\circ\text{C}$), and *Right BC* ($h=16 \text{ W/m}^2.K$, $T_{amb}=22^\circ\text{C}$)



Set $r_0=1.5 \text{ cm}$.



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



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

Heat Conduction - Composit Systems

Input

Plane Wall Cylinder Sphere
 No. of Layers: 1 Initialize

Right BC (r=0):
 Convection h: 80 W/m2.K Tamb: 0 C
 Fixed Temp Ts: C
 Heat FLux q's: W/m2

Right BC (r=R):
 Convection h: 16 W/m2.K Tamb: 22 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	15	2

Results
 qr: -8329.1342 W
 R'tot: 0.0026 C.m2/W
 R'tot: C.m2/W
 U: 378.5970 W/m2.K

Layers:
 X: m T: C Rth: C.m2/W

	r Left	T Left	Rth	r Right	T Right
	1.50000	3.68	0.00005	1.52000	4.07

[Update](#)

[Show Tutorial](#)

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

<<End-of-Tutorial>>

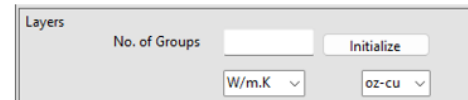
HT-02: K effective Parallel/Series

This tool 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:

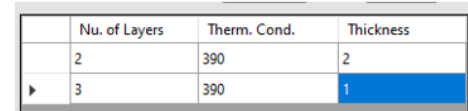
1. Initialization

- 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.



2. Definition of Layers

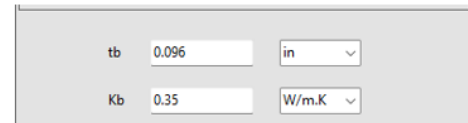
- Enter values for
 - Number of layers for each group.
 - Thermal conductivity for these layers (Choose unit from the Combo-Box above).
 - Thickness for each layer (Choose unit from the Combo-Box above).



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

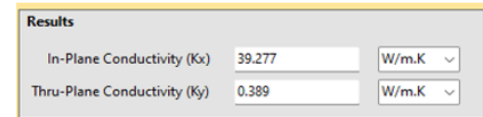
3. Background (Dielectric) Definition

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



Results:

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



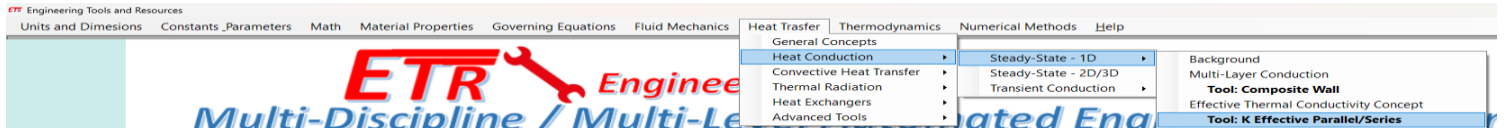
Results		
In-Plane Conductivity (Kx)	39.277	W/m.K
Thru-Plane Conductivity (Ky)	0.389	W/m.K

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 ($k=0.35$ W/m.K) and copper conductivity is 390 W/m.K.

Solution:

Open “Effective Thermal Conductivity” Panel:



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

Layers

No. of Groups: 2

W/m.K

oz-cu

Initialize

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

W/m.K

oz-cu

Initialize

	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 (K_x): 39.277 W/m.K

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

Update

Diagram: A cross-section of a multi-layer PCB is shown with layers labeled $N1, t1, k1$, $N2, t2, k1$, and $N3, t3, k3$. The total thickness is t_b . The X and Y axes are indicated.

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

<<End-of-Tutorial>>

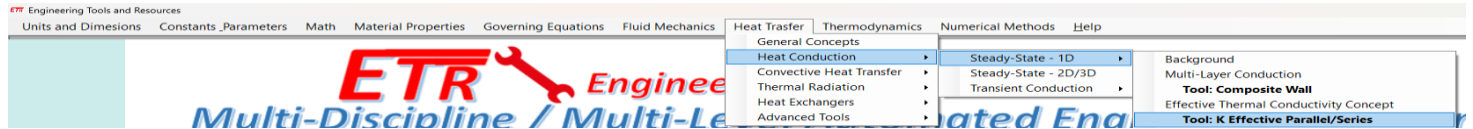
HT-03: K effective Vias

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

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

Solution:

Open “Effective Conductivity - Vias” Panel:



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

Via Inner Diameter = **0.3 mm**

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

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

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

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

L = **0.65 in**

W = **0.4 in**

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

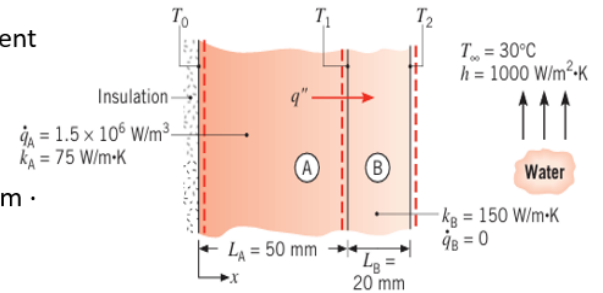
The calculated thru-board conductivity is 14.107 W/m.K. Note that this analysis assumes unfilled vias (air conductivity of 0.03 W/m.K is used by default).

<<End-of-Tutorial>>

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

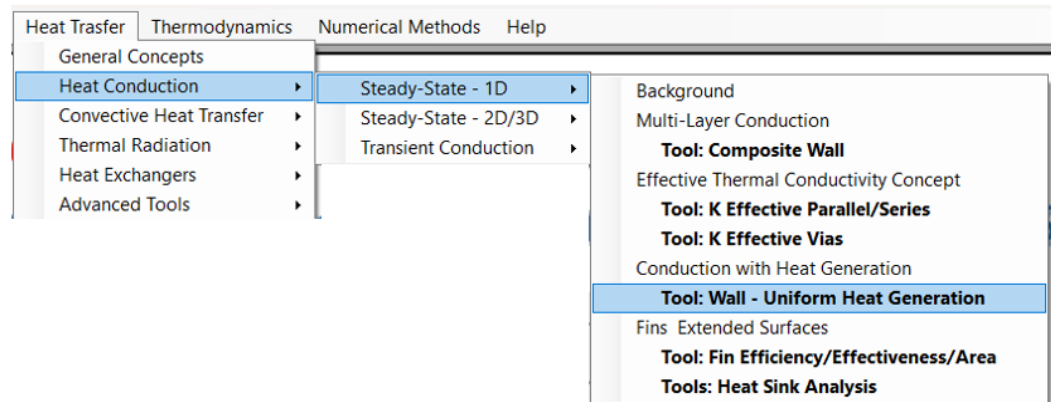
A plane wall of material A has uniform heat generation 1.50 MW/m^3 , $k_A = 75 \text{ W/m} \cdot \text{K}$, and thickness $L_A = 50 \text{ mm}$. The outer surface of the wall (larger x value) is cooled by a water stream with a temperature of $30 \text{ }^\circ\text{C}$ and heat transfer coefficient of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

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



Solution:

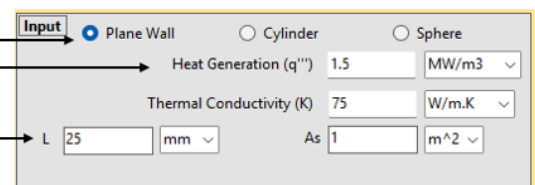
a) Open "Solids with Uniform Heat Generation" Panel:



Enter input data

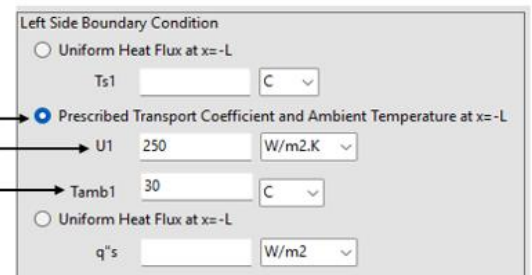
Geometric Parameters:

Geometry: Plane Wall
 Heat Generation: Note can select 1.5 MW/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.



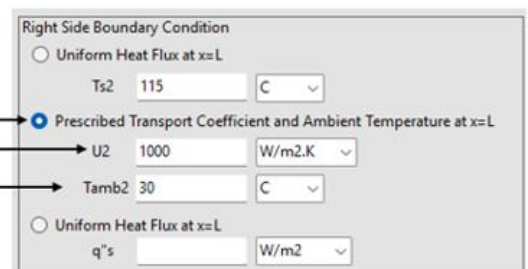
Left side (x=-L) Boundary Conditions:

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



Right side (x=L) Boundary Conditions:

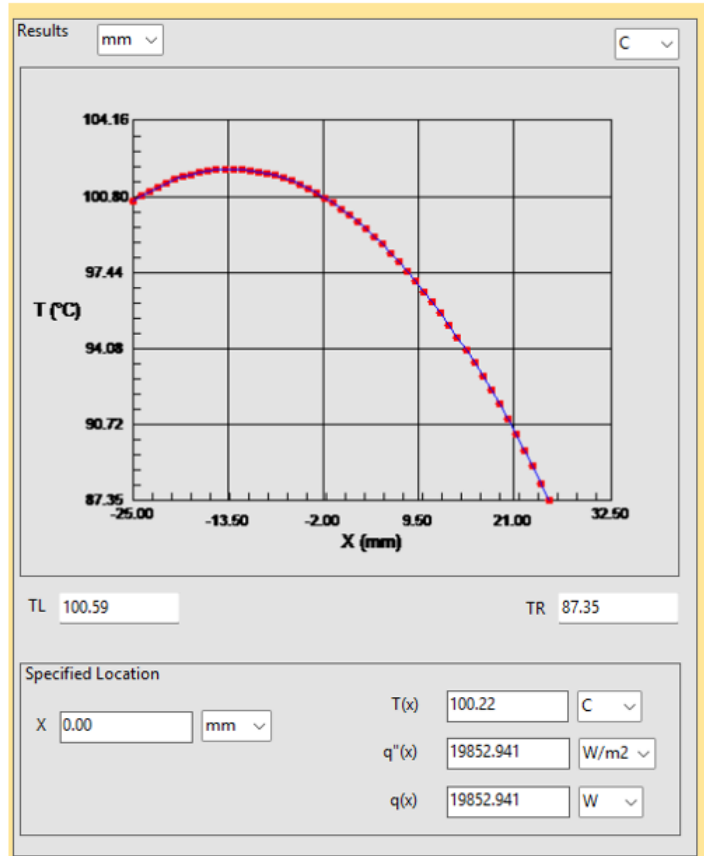
Select Convective BC.
 Enter the value for convection coefficient on the right side.
 Enter the value for the ambient temperature on the right side.



Results

Click Update to solve.

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



b) The Inner surface is insulated

The Only change to be made is to make the left side insulated. This can be done by assigning a value of zero for the overall heat transfer coefficient U_1 .

Left Side Boundary Condition

Uniform Heat Flux at $x=-L$

T_{s1} C

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

U_1 0 W/m².K

T_{amb1} 30 C

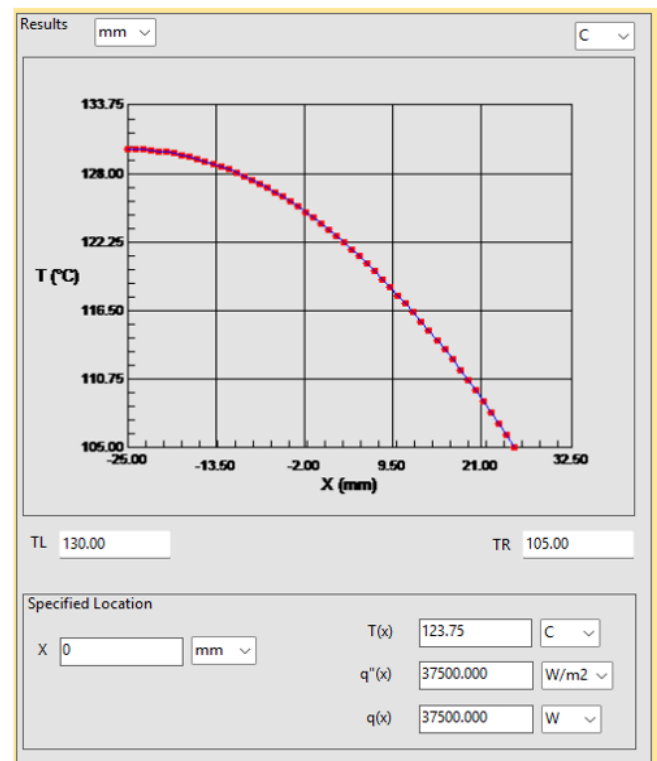
Uniform Heat Flux at $x=-L$

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.

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

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

Right Side Boundary Condition

Uniform Heat Flux at x=L

Ts2 115 C

Prescribed Transport Coefficient and Ambient Temperature at x=L

U2 882.353 W/m2.K

Tamb2 30 C

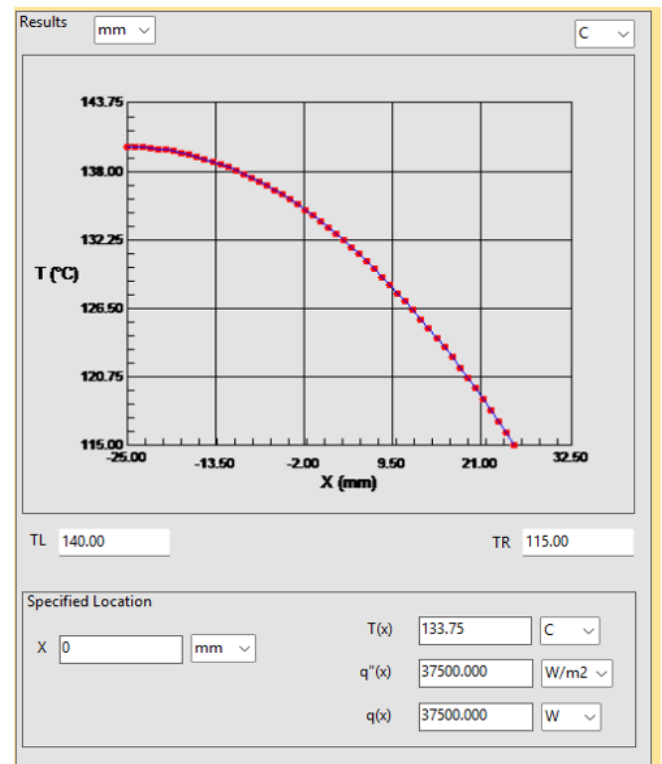
Uniform Heat Flux at x=L

q''s W/m2

Results

Click Update to solve.

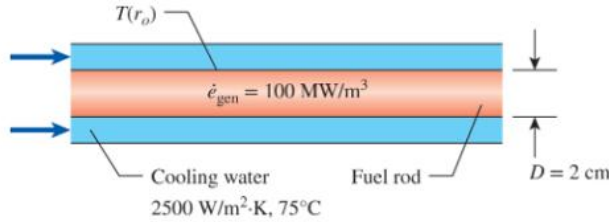
Plot of temperature distribution through the slab indicates insulated BC on the left face and updated temperature and heat flux values.



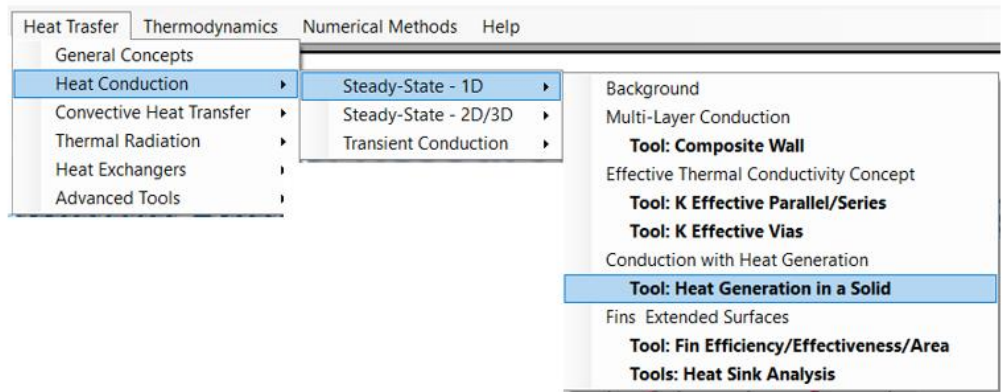
Example: Water-Cooled Fuel Rod [Source: Cengel-Ghagar Problem 2.104]

A cylindrical fuel rod ($k = 30 \text{ W/m.K}$) 2 cm in diameter is encased in a concentric tube and cooled by water. The fuel rod generates heat uniformly at a rate of 100 MW/m^3 , and the average temperature of the cooling water is 75°C with a convection heat transfer coefficient of $2500 \text{ W/m}^2.\text{K}$. The operating pressure of the cooling water is such that the surface temperature of the fuel rod must be kept below 200°C to prevent the cooling water from reaching the critical heat flux (CHF). The critical heat flux is a thermal limit at which a boiling crisis can occur that causes overheating on the fuel rod surface and leads to damage. Determine the variation of temperature in the fuel rod and the temperature of the fuel rod surface. Is the surface of the fuel rod adequately cooled?

Solution:



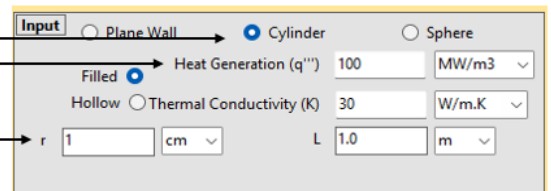
Open "Heat Generation in a Solid" Panel



Enter input data

Geometric Parameters:

- Geometry: Cylinder and select "Filled" option
- Heat Generation: Enter 100 and change unit to MW/m³
- Enter thermal conductivity 30 W/m.K
- Radius: Enter 1 cm (diameter is given as 2 cm).
- Length 1.0 m is assumed since the value is not given.



Left side (r=r1) Boundary Conditions:

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

Left Side Boundary Condition

Uniform Heat Flux at r=r1
Ts1 C

Prescribed Transport Coefficient and Ambient Temperature at r=r1
U1 W/m2.K
Tamb1 C

Uniform Heat Flux at r=r1
q"s W/m2

Right side (r=r2) Boundary Conditions:

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

Right Side Boundary Condition

Specified Temperature at r=r2
Ts2 C

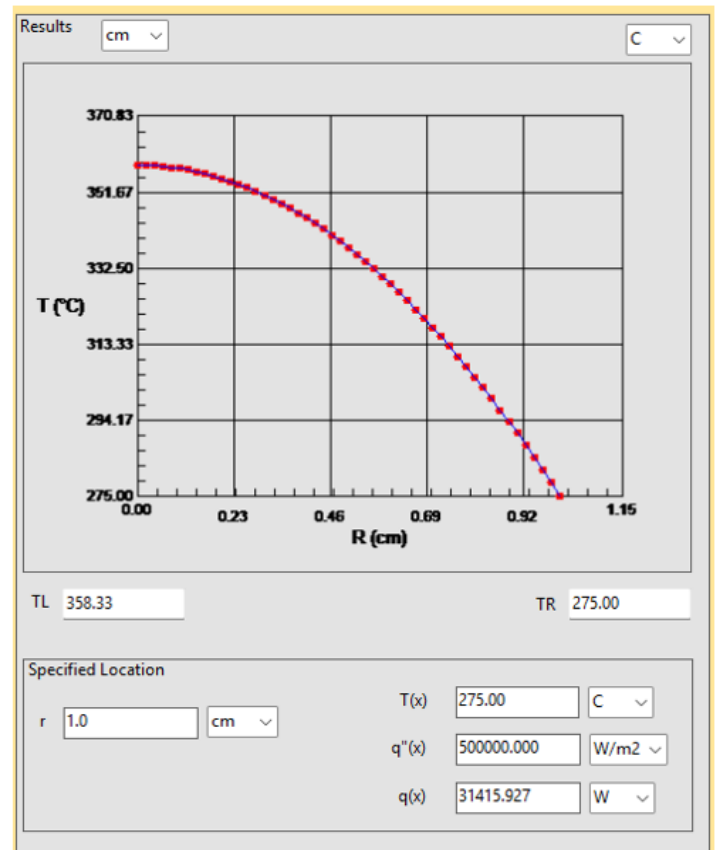
Prescribed Transport Coefficient and Ambient Temperature at r=r2
U2 W/m2.K
Tamb2 C

Uniform Heat Flux at r=r2
q"s W/m2

Results

Click Update to solve.

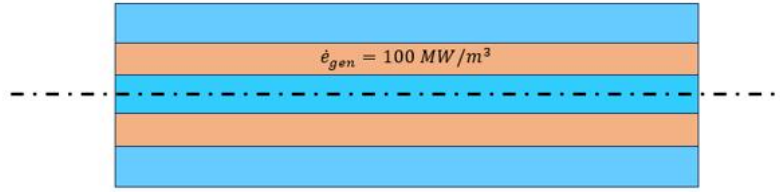
1. Plot of temperature distribution through the slab is provided.
2. Left-side (centerline) and Right-Side (r=1 cm) temperatures are shown.
3. The user can select the units for temperature and length for the plot are from unit boxes at the top.
4. The surface temperature of 275 °C is 75°C higher than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



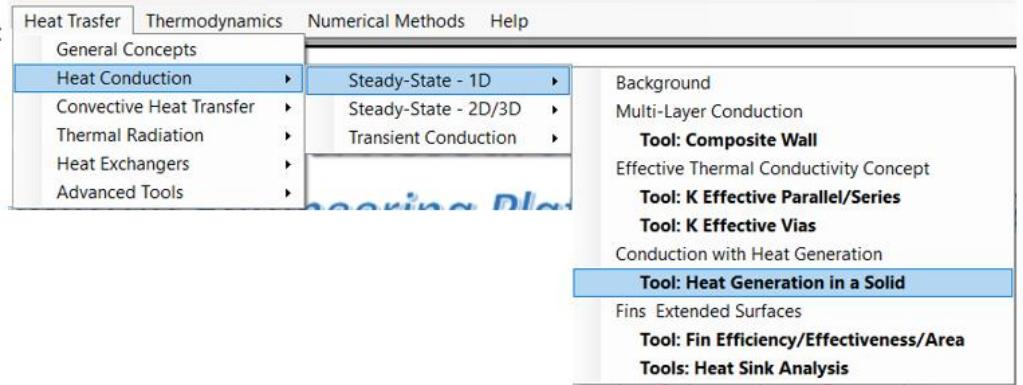
Example: Water-Cooled Fuel Rod [Source: Cengel-Ghagar Problem 2.104 - Modified]

Repeat the above example with 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.

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

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

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=r1) Boundary Conditions:

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

Left Side Boundary Condition

Specified Temperature at r=r1
Ts1 C

Prescribed Transport Coefficient and Ambient Temperature at r=r1
U1 W/m2.K
Tamb1 C

Uniform Heat Flux at r=r1
q"s W/m2

Right side (r=r2) Boundary Conditions:

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

Right Side Boundary Condition

Specified Temperature at r=r2
Ts2 C

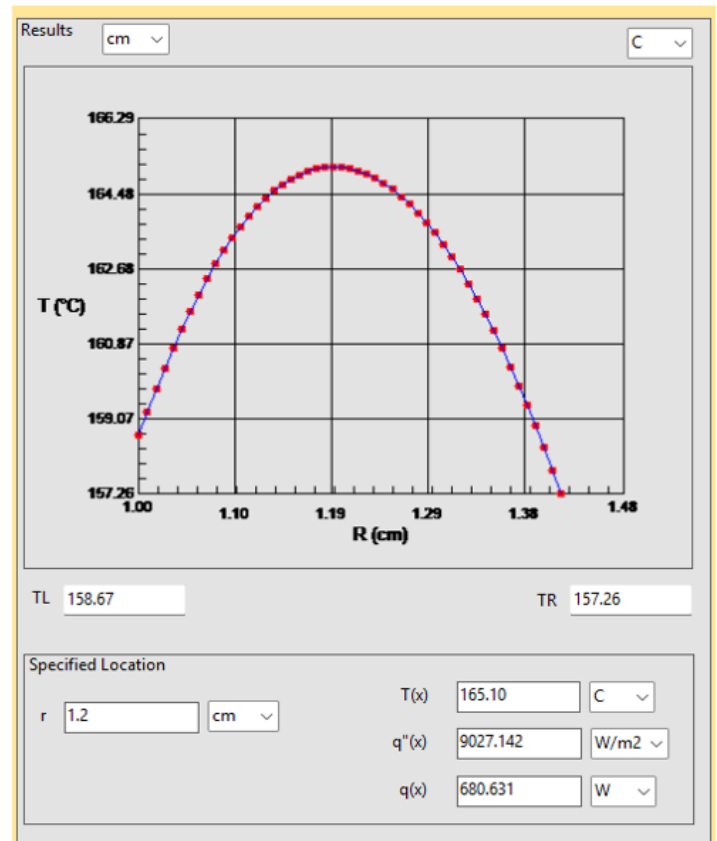
Prescribed Transport Coefficient and Ambient Temperature at r=r2
U2 W/m2.K
Tamb2 C

Uniform Heat Flux at r=r2
q"s W/m2

Results

Click Update to solve.

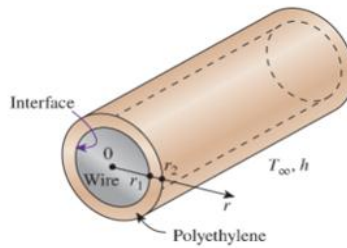
1. Plot of temperature distribution through the slab is provided.
2. Left-side (centerline) and Right-Side (r=1 cm) temperatures are shown.
3. The user can select the units for temperature and length for the plot are from unit boxes at the top.
4. The surface temperature of 158.57 °C and 157.26 °C are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



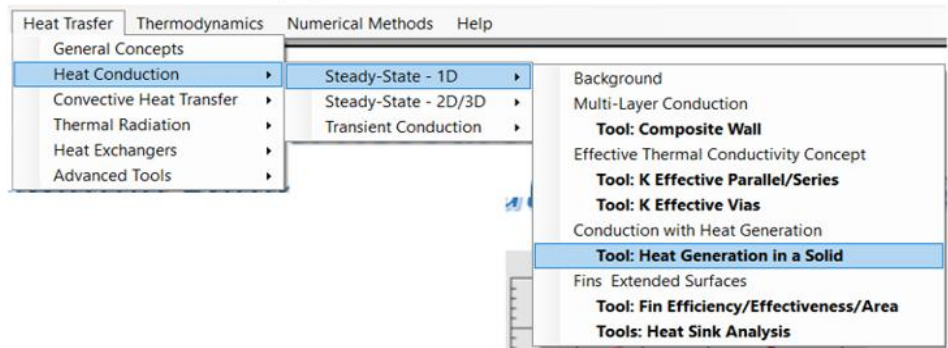
Example: Electrical Resistance wire with Insulation [Source: Cengel-Ghagar Problem 2.105 - Modified]

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:

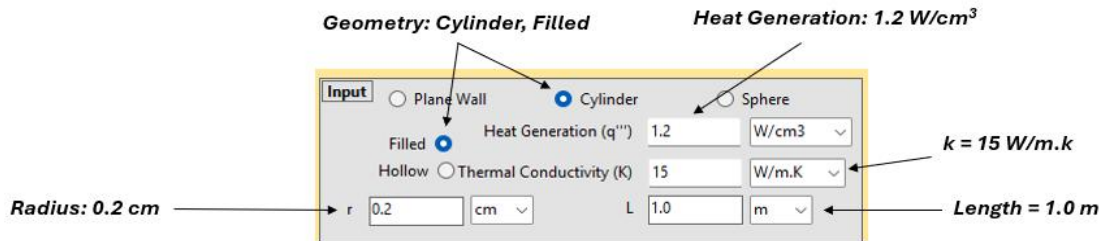


Open "Heat Generation in a Solid" Panel:



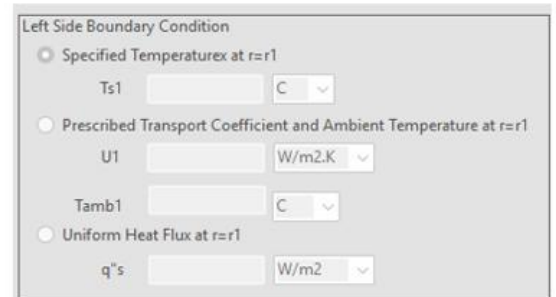
Enter input data

Geometric Parameters:



Left side (r=r1) Boundary Conditions:

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



Right side (r=r2) Boundary Conditions:

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

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

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.

The screenshot shows the Composite Solid tool interface. The 'Input' section is set to 'Cylinder' with 1 layer. The left boundary condition (r=0) is set to 'Fixed Temp' with $T_s = 30$ °C. The right boundary condition (r=R) is set to 'Convection' with $h = 7$ W/m².K and $T_{amb} = 20$ °C. The 'Layers' table shows one layer with a thickness of 0.5 cm and thermal conductivity of 0.4 W/m.K. The 'Results' section shows $q_r = 2.6691$ W, $R'_{tot} = 3.7465$ C/W, $UA = 0.2669$ W/C, $U_{Rmin} = 21.2404$ W/m².K, and $U_{Rmax} = 6.0687$ W/m².K. The 'Layers' table in the results section shows the calculated R_{th} for the layer as 0.498 C.m²/W.

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

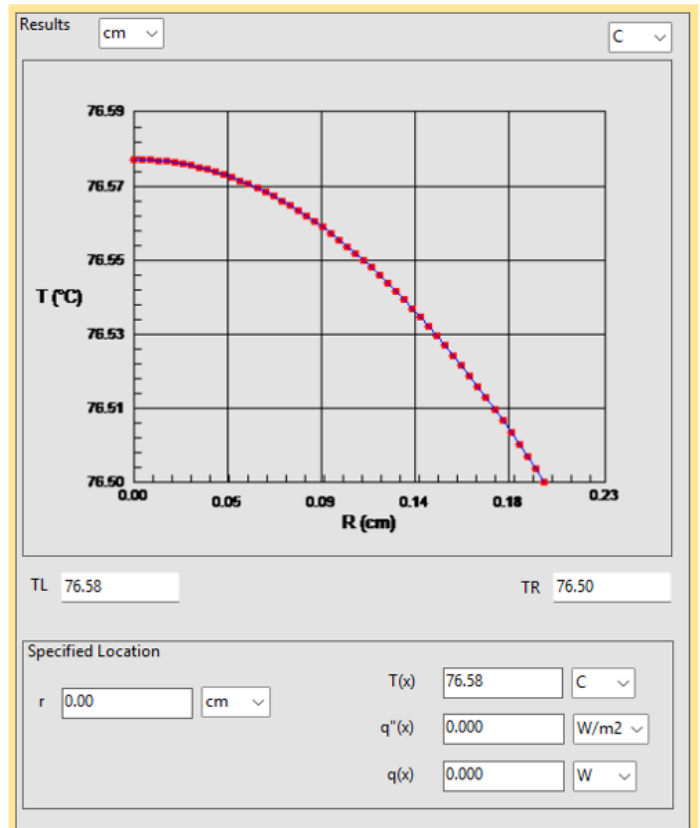
Select Convective BC.
Enter the value for convection coefficient on the outer surface.
Enter the value for the ambient temperature on the outer surface.

The screenshot shows the 'Right Side Boundary Condition' panel. The 'Prescribed Transport Coefficient and Ambient Temperature at r=r2' option is selected. The U_2 value is set to 21.2404 W/m².K and the ambient temperature T_{amb2} is set to 20 °C. Arrows point from the text above to these two input fields.

Results

Click Update to solve.

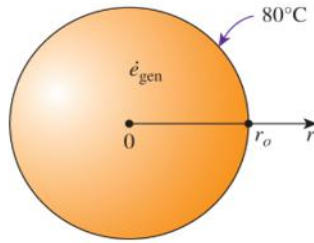
1. Plot of temperature distribution through the slab is provided.
2. Left-side (centerline) and Right-Side ($r=1$ cm) temperatures are shown.
3. The user can select the units for temperature and length for the plot are from unit boxes at the top.
4. The surface temperature of 76.5 °C is 1.5 °C higher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



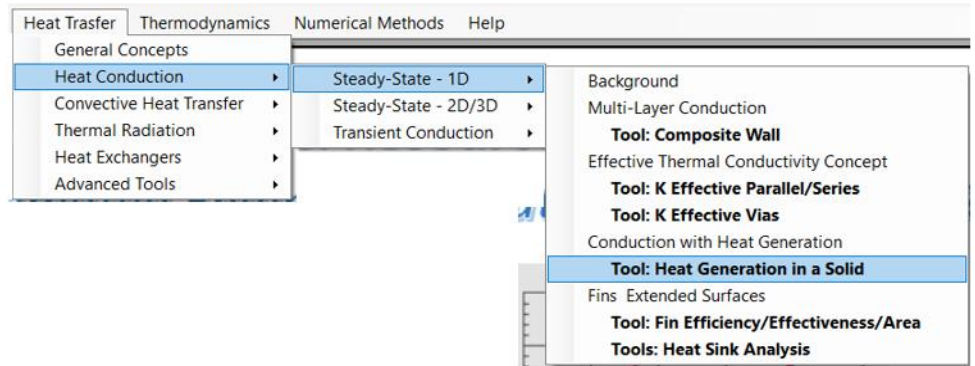
Example:

Consider a homogeneous spherical piece of radioactive material of radius $r_o = 0.04$ m that is generating heat at a constant rate of $e_{gen} = 4 \times 10^7$ W/m³. The heat generated is dissipated to the environment steadily. The outer surface of the sphere is maintained at a uniform temperature of 80C, and the thermal conductivity of the sphere is $k = 15$ W/m.K. Assuming steady one-dimensional heat transfer, determine the temperature at the center of the sphere.

Solution:



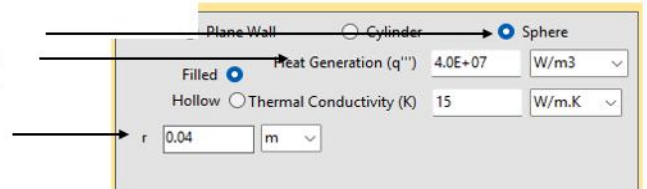
Open "Heat Generation in a Solid" Panel:



Enter input data

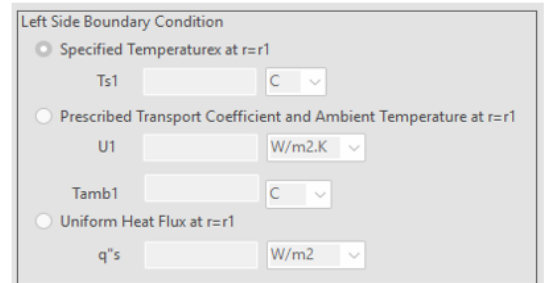
Geometric Parameters:

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



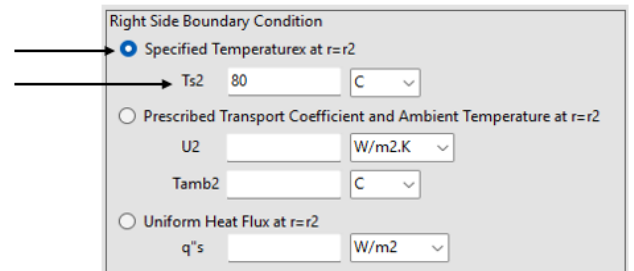
Left side (r=r1) Boundary Conditions:

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



Right side (r=r2) Boundary Conditions:

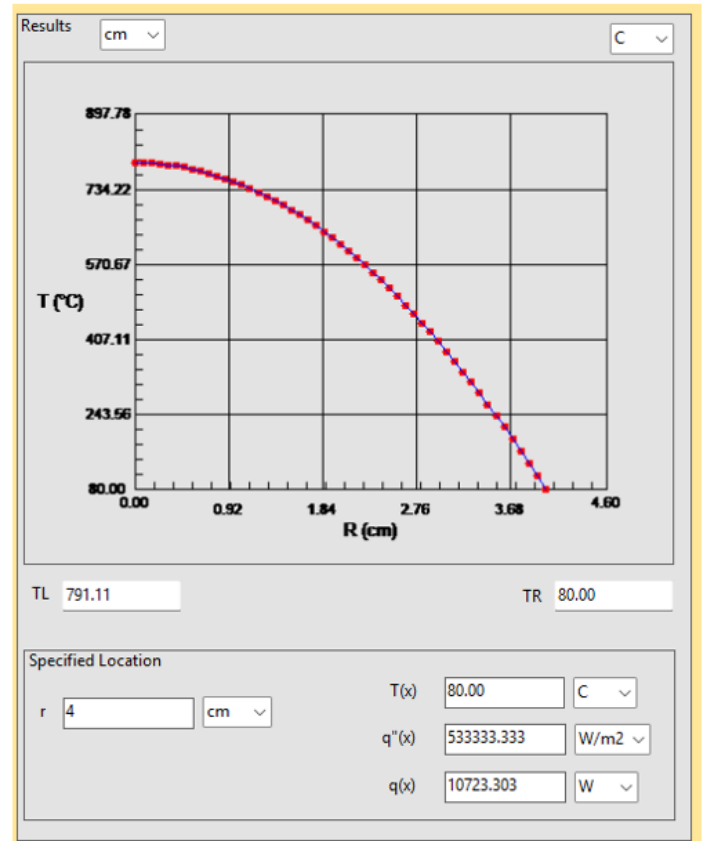
Select Specified Temperature.
Enter 80 °C.



Results

Click Update to solve.

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

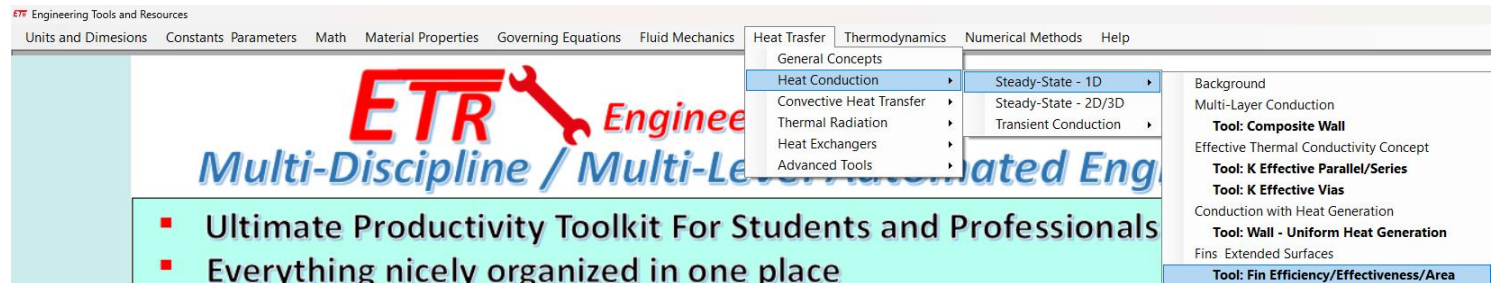


HT-05: Fin Efficiency/Effectiveness

Example: A heat sink utilizes aluminum 2024-T6 ($k = 186 \text{ W/m} \cdot \text{K}$) 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:



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

f) Cone with blunt tip

k 186 W/m.K

h 20 W/m².K

Fin Dimensions

L 20 mm

D 5 mm

Fin Area 2.11E-04 m²

Base Area 1.96E-05 m²

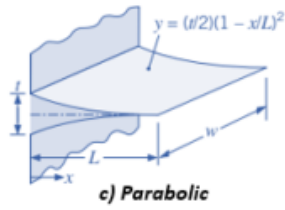
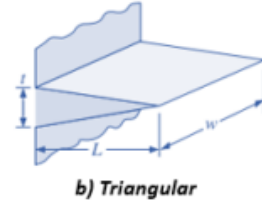
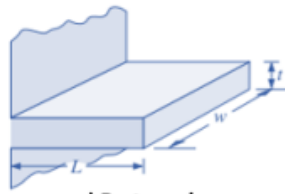
Fin Volume 5.39E-04 m³

Fin Efficiency 0.9668

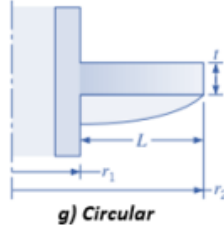
Fin Effectiveness 10.3703

Update

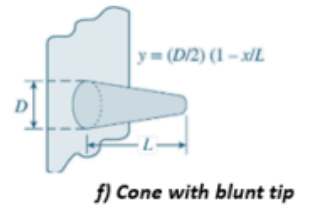
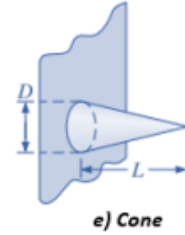
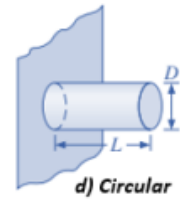
Straight Fins



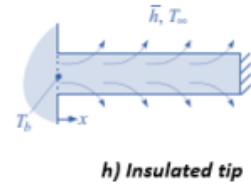
Annular Fin



Pin Fins



Constant Cross-Section Fin



Show Tutorial

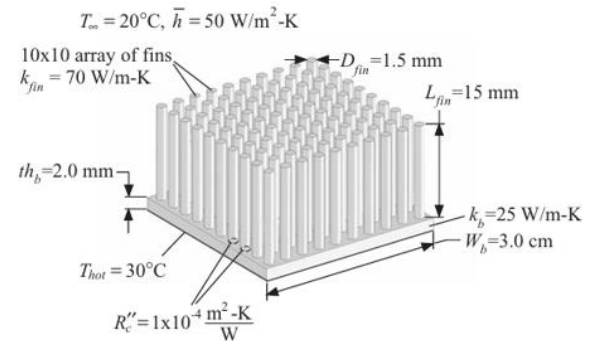
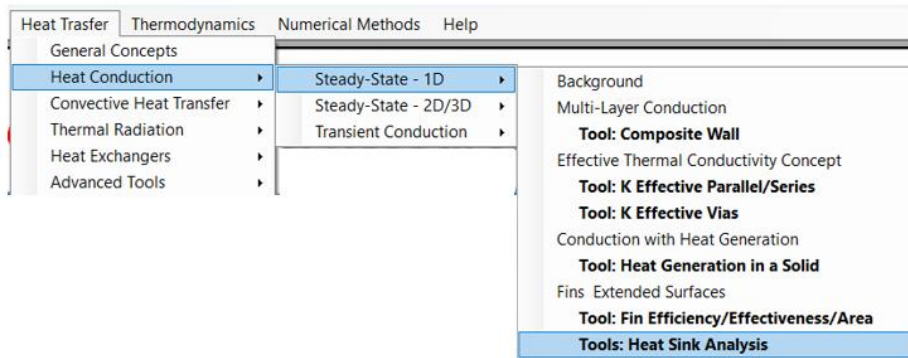
The fin efficiency is 96.67% and the effectiveness is calculated to be 10.37.

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

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

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

Solution:



Enter input data

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 (N_{fin}) 100

Contact Base-Fin (R''_c, fin) 0.0001 C.m²/W

Avg Heat Tran Coeff (h) 50 W/m².K

Ambient Temp. (T_{amb}) 20 C

Fin Details

Fin surface area [] m²

Fin base area [] m²

Fin Efficiency []

Use Fin Tool

d) Circular

k 70 W/m.K

h 50 W/m².K

Fin Dimensions

L 15 mm

D 1.5 mm

Fin Area [] m²

Base Area [] m²

Fin Volume [] m³

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 m²

Fin base area 1.77E-06 m²

Fin Efficiency 0.8780

Use Fin Tool

1-Heat Sink Input

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..

frmHeatSink

Heat Sink Input

Rectangular Heat Sink Circular Heat Sink

Base Plate Length (L) cm

Base Plate Width (w) cm

Base Plate Thickness (t) mm

Base Plate Conductivity (kb) W/m.K

Contact base-source (R^{c,b}) C.m²/W

Number of Fins (Nfin)

Contact Base-Fin (R^{c,fin}) C.m²/W

Avg Heat Tran Coeff (h) W/m².K

Ambient Temp. (Tamb) C

Fin Details

Fin surface area m²

Fin base area m²

Fin Efficiency

$R_{fin} = \frac{1}{\eta_{fin} h A_{s,fin}}$ (thermal resistance – single fin)
 $R_{unfinned} = \frac{1}{h(A_b - N_{fin} A_{c,fin})}$ (thermal resistance – unfinned section)
 $R_{cond,b} = \frac{t_b}{k_b A_b}$ (thermal resistance – baseplate)
 $R_{c,b}, R_{c,fin}$ (contact resistances – baseplate, fin)
 $A_{s,fin} \equiv$ Fin total surface area
 $A_{c,fin} \equiv$ Fin cross-section area
 $A_b \equiv$ Baseplate Area

Results

Total Thermal Resistance (Rtot) C.m²/W Source Temperature (Thot) C

Overall Surface Efficiency (η_o) Heat Transfer Rate (Qs) W

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

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

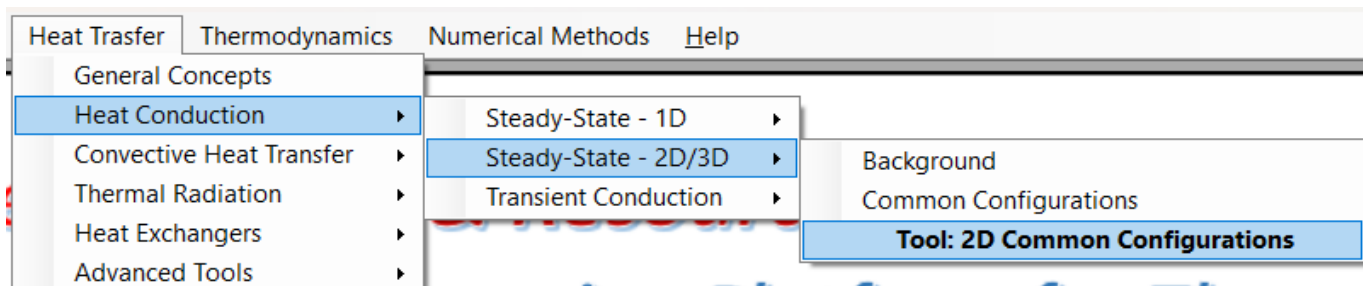
Note: The above results are identical to those in Nellis-Klein as obtained using EES program.

HT-07: Multidimensional Heat Transfer in Common Configurations

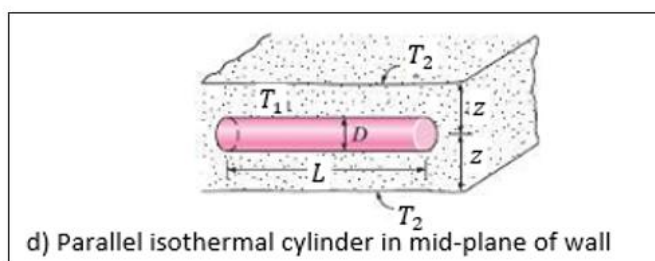
Example: Hot water at $53\text{ }^{\circ}\text{C}$ flows through a 5-m long section of a thin-walled hot water pipe at an average velocity of 5 m/s. The pipe passes through the center of a 14-cm thick wall filled with fiberglass ($k = 0.035\text{ W/m}\cdot\text{K}$) insulation. Assuming the surfaces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

Solution:

Open “Common Configuration/Shape Factor” Panel:



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



$$\begin{aligned}T_1 &= 53\text{ }^{\circ}\text{C} \\T_2 &= 18\text{ }^{\circ}\text{C} \\D &= 2.5\text{ cm} \\L &= 5\text{ m} \\z &= 7\text{ cm}\end{aligned}$$

Enter input in given units:

Dimensions		
D	<input type="text" value="2.5"/>	<input type="text" value="cm"/>
L	<input type="text" value="5"/>	<input type="text" value="m"/>
z	<input type="text" value="7"/>	<input type="text" value="cm"/>

Thermal Input		
k	<input type="text" value="0.035"/>	<input type="text" value="W/m.K"/>
T1	<input type="text" value="53"/>	<input type="text" value="C"/>
T2	<input type="text" value="18"/>	<input type="text" value="C"/>

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

Results		
S	<input type="text" value="15.993"/>	<input type="text" value="m"/>
Rth	<input type="text" value="1.786"/>	<input type="text" value="K/W"/>
Q12	<input type="text" value="19.592"/>	<input type="text" value="W"/>
<input type="button" value="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:

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

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

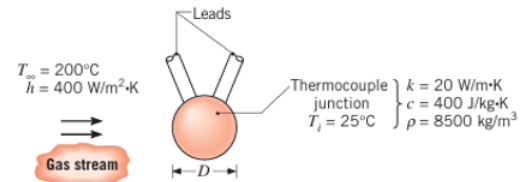
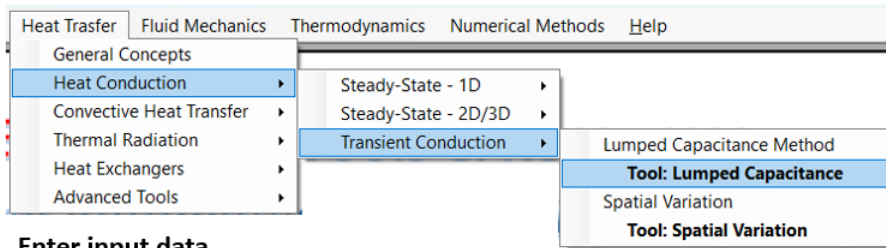
<<End-of-Tutorial>>

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

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

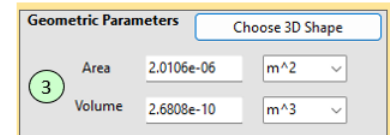
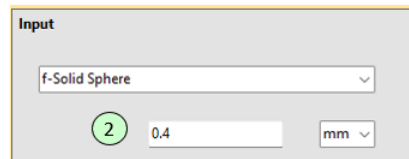
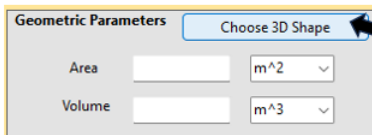
- temperature versus time for eight seconds.
- Determine the junction temperature at time $t = 1 \text{ s}$.
- Determine how long it will take for the junction to reach 198°C .

Solution:



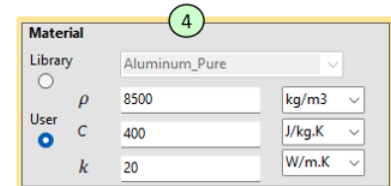
Enter input data

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

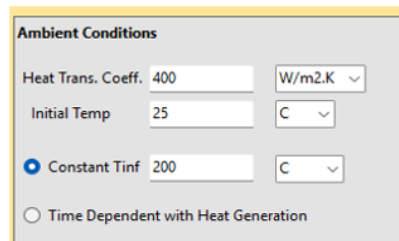


- Choose option “f” for Solid Sphere and enter the radius.
- Click update to automatically enter these parameters into “Lumped Capacity Form”.

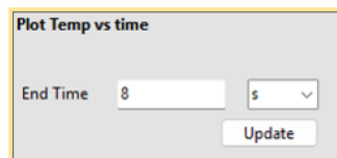
- Enter solid thermal properties.



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



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



Part b):

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

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.

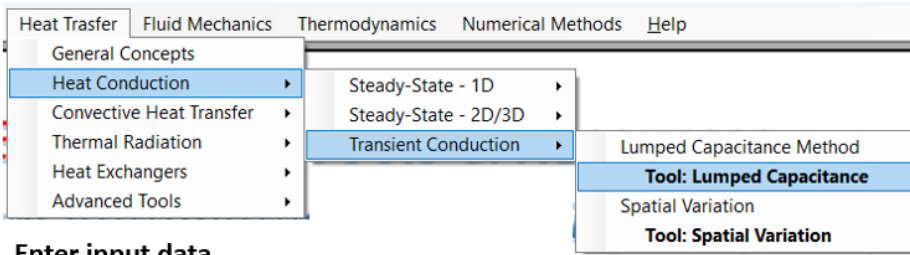
The screenshot shows the 'Lumped Capacitance' software interface. On the left, there are three main sections: 'Geometric Parameters' (Area: 2.0106e-06 m², Volume: 2.6808e-10 m³), 'Material' (Library: Aluminum_Pure, ρ: 8500 kg/m³, C: 400 J/kg.K, k: 20 W/m.K), and 'Ambient Conditions' (Heat Trans. Coeff: 400 W/m².K, Initial Temp: 25 C, Constant Tinf: 200 C). Below these is a table for time, Tinf, and Q, and buttons for 'Add Row' and 'Remove Row'. The central plot, titled 'Plot of Temp vs Time', shows temperature (C) on the y-axis (ranging from 25.00 to 210.00) and time (s) on the x-axis (ranging from 0.00 to 9.00). A blue curve with red markers shows the temperature rising from 25.00 C at 0.00 s and asymptotically approaching a steady state of approximately 200 C. A green circle with the number '6' is placed on the plot. Below the plot, the Biot number (Bi) is 0.0027. To the right of the plot are controls for 'Temperature Unit' (Celsius selected) and 'Time Unit' (Second selected). At the bottom, there are three summary panels: 'Plot Temp vs time' (End Time: 8 s), 'Temperature at Specified Time' (Time: 1 s, Temperature: 127.584 C), and 'Time to Reach Temperature' (Temperature: 198 C, Time: 5.068 s). Green circles with numbers '7' and '8' are placed on these panels. A 'Show Tutorial' button is at the bottom right.

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

Example – Thermal Response of Thermocouple in a Thermal Chamber

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

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



Enter input data

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

2. Choose option “f” for Solid Sphere and enter the radius.
3. Click update to automatically enter these parameters into “Lumped Capacity Form”.
4. Enter solid thermal properties. Choose Al_6061-T6 from list of materials in the library.

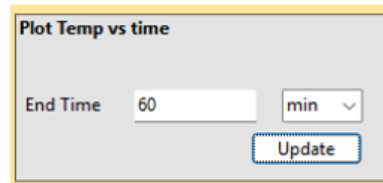
5) In “Ambient Conditions” section:

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

time	Tinf	Q
0	85	
10	-50	
30	-50	
40	85	

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.

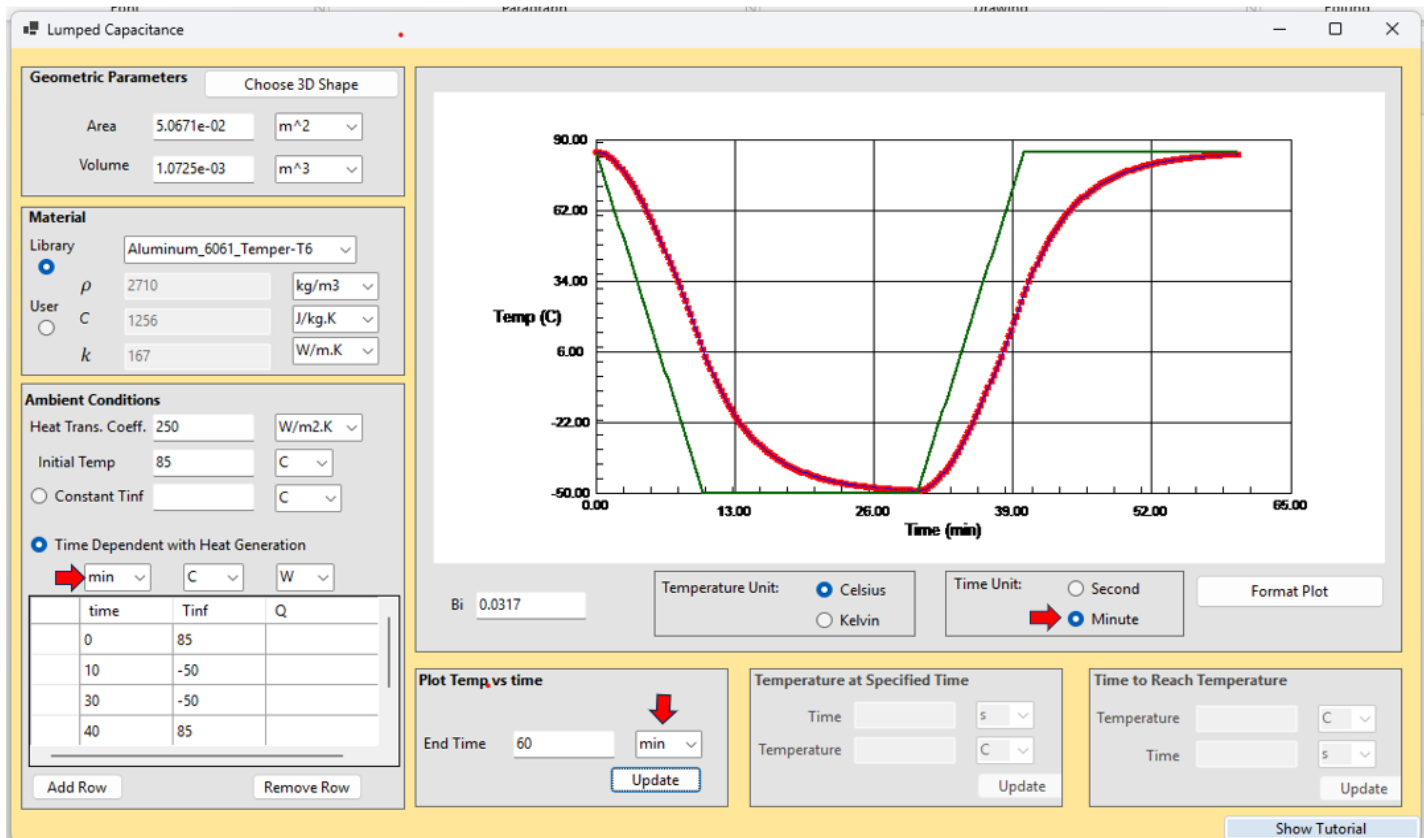


Plot Temp vs time

End Time 60 min

Update

Finished form is shown below.



Lumped Capacitance

Geometric Parameters Choose 3D Shape

Area 5.0671e-02 m²

Volume 1.0725e-03 m³

Material

Library Aluminum_6061_Temper-T6

User ρ 2710 kg/m³

C 1256 J/kg.K

k 167 W/m.K

Ambient Conditions

Heat Trans. Coeff. 250 W/m².K

Initial Temp 85 C

Constant Tinf C

Time Dependent with Heat Generation

min C W

time	Tinf	Q
0	85	
10	-50	
30	-50	
40	85	

Add Row Remove Row

Bi 0.0317

Temperature Unit: Celsius Kelvin

Time Unit: Second Minute

Format Plot

Plot Temp vs time

End Time 60 min

Update

Temperature at Specified Time

Time s

Temperature C

Update

Time to Reach Temperature

Temperature C

Time s

Update

Show Tutorial

Temp (C) vs Time (min) Plot:

Time (min)	Temp (C)
0.00	85.00
13.00	-50.00
26.00	-50.00
39.00	85.00
52.00	85.00
65.00	85.00

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

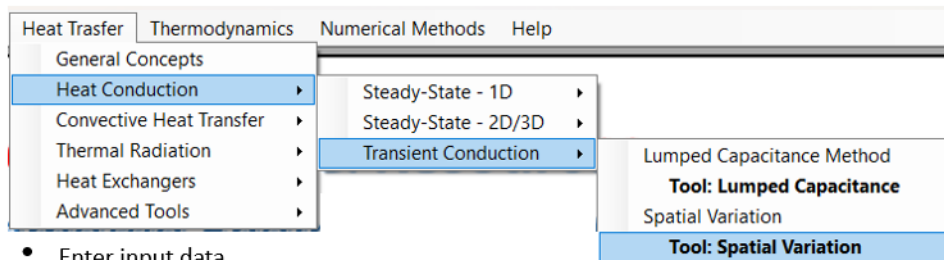
Example – Plate: Heating of a Steel Pipeline [Source: Bergman-Lavine Example 7.7]

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 . Upon initiation of flow, hot oil at 60°C is pumped through the pipe, creating a convective condition with $h = 500 \text{ W/m}^2 \cdot \text{K}$ at the inner surface of the pipe.

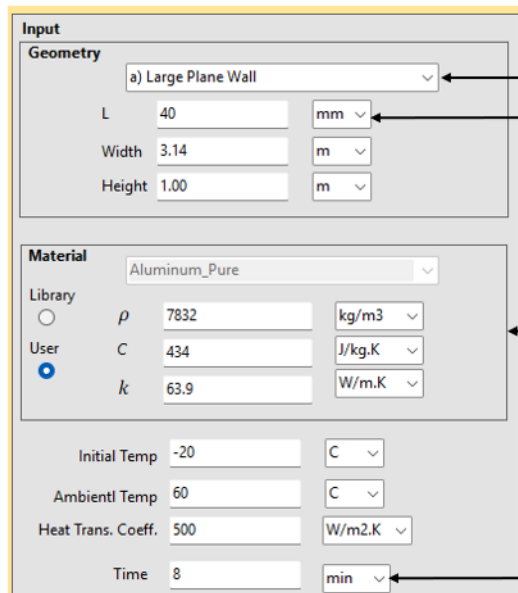
- Determine the temperature of the exterior pipe surface covered by the insulation at $t = 8 \text{ min}$
- How much energy per meter of pipe length has been transferred from the oil to the pipe at $t = 8 \text{ min}$?

Solution:

- Open “Transient Conduction” Tool



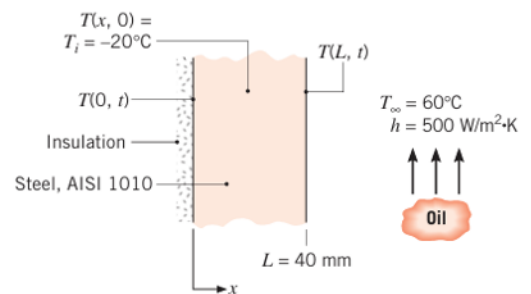
- Enter input data



Geometry: 1-D Plane Wall
 Insulated side – Width=L

User-defined material properties

End time = 8 minutes



- Click Update and Solve

Transient Conduction

Input

Geometry

a) Large Plane Wall

L: 40 mm

Width: 3.14 m

Height: 1.00 m

Material

Aluminum_Pure

Library: ρ 7832 kg/m³

User: C 434 J/kg.K

k 63.9 W/m.K

Initial Temp: -20 C

Ambient Temp: 60 C

Heat Trans. Coeff.: 500 W/m².K

Time: 8 min

Results

Bi: 0.313 Center Temp: 43.13 C

Fo: 5.640 Total Energy Gain: -5.457E+07 J

Internal Temp

x/L: 1.0

Temp: 45.46 C

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

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

Update

Show Tutorial

Comments:

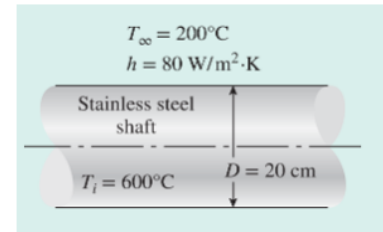
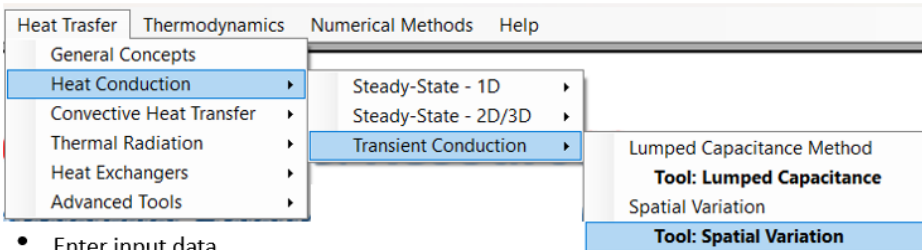
1. The Biot number, $Bi=0.313$ and the Fourier number $Fo=5.64$
2. The "Center Temperature" refers to the temperature at the center of a wall with thickness $2L$. Here, it indicates the temperature at the insulated side of the wall.
3. The "Total Energy Gain" of $-5.457E+07$ J is the energy gained by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

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

A long 20-cm diameter cylindrical shaft made of stainless-steel 304 is out of an oven at a uniform temperature of 600 °C. The shaft is then allowed to cool in an environmental chamber at 200 °C and a heat transfer coefficient of 80 W/m².K. Determine a) the temperature at the center of the shaft after 45 minutes and b) total amount of heat transferred from the shaft to the air during the cooling process per unit length.

Solution:

- Open “Transient Conduction” Tool



- Enter input data

The input data form is divided into several sections:

- Geometry:** Long Cylinder (selected), Radius $r_0 = 10\text{ cm}$, Height 1.00 m .
- Material:** Steel_Stainless_316 (selected). User-defined properties: $\rho = 7900\text{ kg/m}^3$, $C = 477\text{ J/kg}\cdot\text{K}$, $k = 14.9\text{ W/m}\cdot\text{K}$.
- Initial Temp:** 600 °C
- Ambient Temp:** 200 °C
- Heat Trans. Coeff.:** 80 W/m².K
- Time:** 45 min

Annotations point to the following fields:

- Geometry: Long Cylinder
- Radius, r_0
- User-defined material properties
- End time = 45 minutes

Properties of Stainless Steel as given:

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

- Click Update and Solve

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

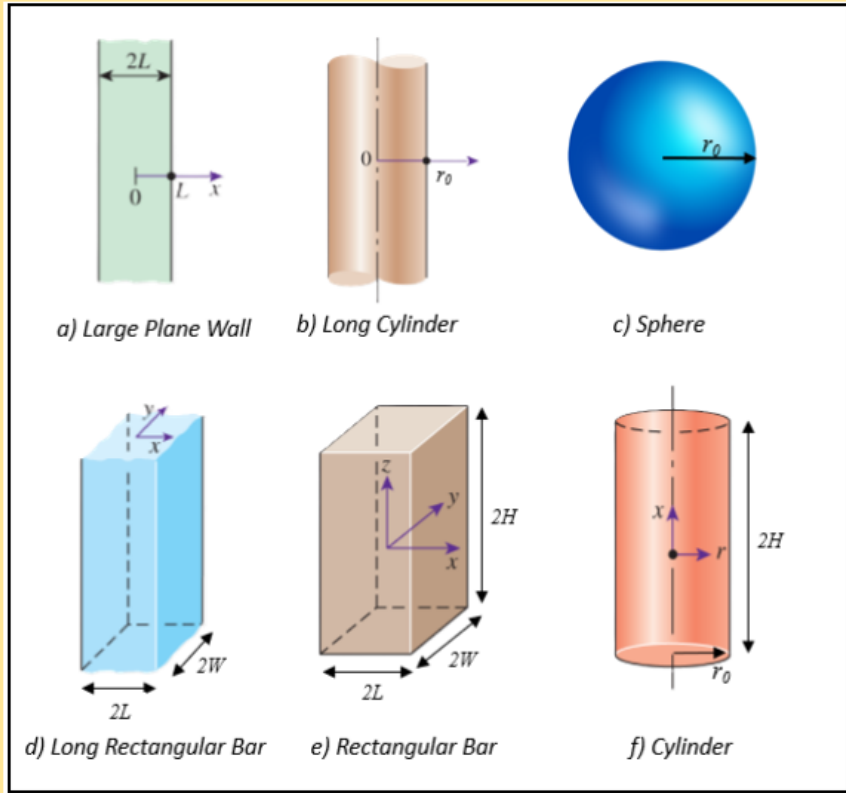
Bi 0.537 Center Temp 362.83 C

Fo 1.068 Total Energy Gain 3.028E+07 J

Internal Temp

r/r0 1.0

Temp 326.45 C



Update

Show Tutorial

Comments:

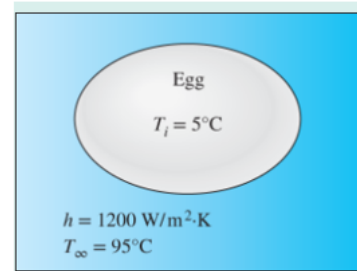
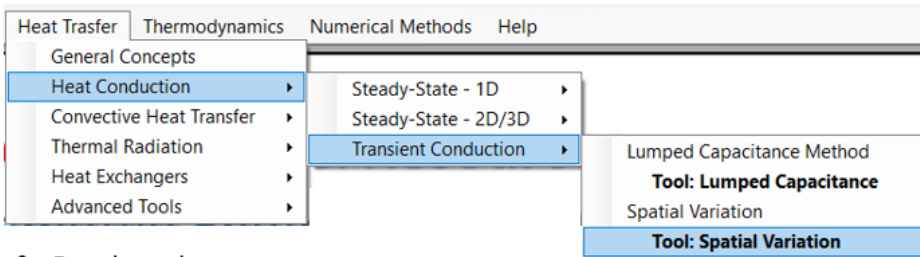
1. The Biot number, $Bi=0.537$ and the Fourier number $Fo=1.068$
2. The "Center Temperature" refers to the temperature at the center of the cylinder is $362.83\text{ }^\circ\text{C}$.
3. The "Total Energy Gain"; $Q_{tot}=3.028E+07\text{ J}$ per meter of the pipe.
4. The surface temperature ($r/r_0=1$)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

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

Geometry: Long Cylinder

Radius, r_0

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

c) Sphere

r_0 2.5 cm

Material

Aluminum_Pure

Library ρ 994 kg/m³

User 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

Results

Bi 48.154 Center Temp 71.71 C

Fo 0.202 Total Energy Gain -2.244E+04 J

Internal Temp r/r_0 1.0

Temp 93.88 C

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

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

Update

Show Tutorial

Comments:

1. The Biot number, $Bi=48.15$ and the Fourier number $Fo=0.202$
2. The "Center Temperature" is $71.7\text{ }^{\circ}\text{C}$.
3. The surface temperature of the egg is calculated to be $93.9\text{ }^{\circ}\text{C}$.
4. The "Total Energy Gain"; $Q_{tot}=2.244\text{E}+04\text{ J}$ into the egg.

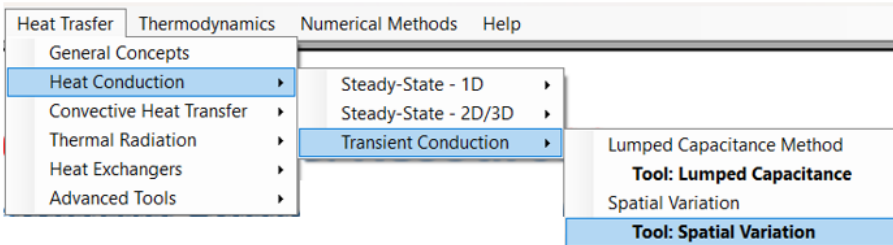
Example – Short Cylinder: Cooling of a Brass Bar

A short brass cylinder ($\rho = 8530 \text{ kg/m}^3$, $C_p = 0.389 \text{ kJ/kg}\cdot\text{K}$, $k = 110 \text{ W/m}\cdot\text{K}$) 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 $40 \text{ W/m}^2\cdot\text{K}$. 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:

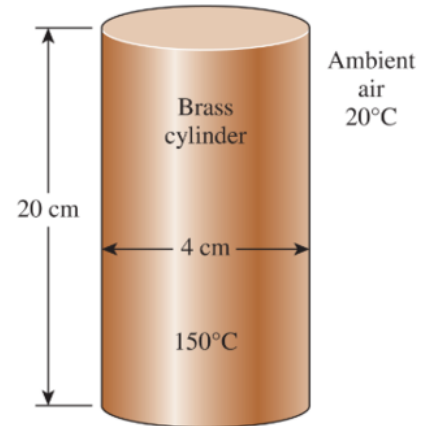


- Enter input data

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

Material	
Brass	
Library	
<input type="radio"/> ρ 8530 kg/m ³	← Since the material properties are specified, choose User option and enter the properties.
User <input checked="" type="radio"/> C 389 J/kg.K	
k 110 W/m.K	

Initial Temp 150 C	← Enter the initial temperature, ambient temperature, heat transfer coefficient, and cooling time.
Ambientl Temp 20 C	
Heat Trans. Coeff. 40 W/m ² .K	
Time 15 min	



- Click update to solve.

The finished form is shown.

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

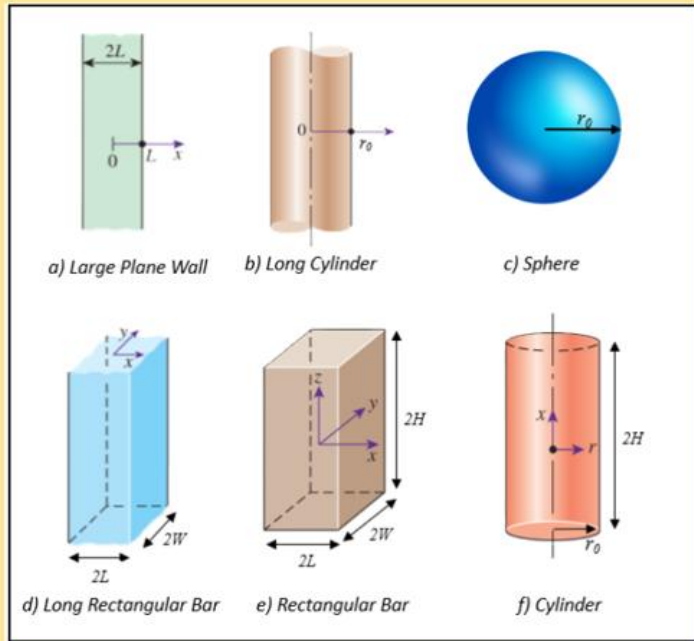
Bi 0.007 Center Temp 60.45 C

Fo 74.589 Total Energy Gain 7.498E-04 J

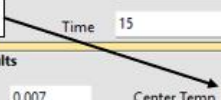
Internal Temp r/r0 z/H

1.0 1.0

Temp 59.57 C



A) Center Temp



Location of center of the top surface

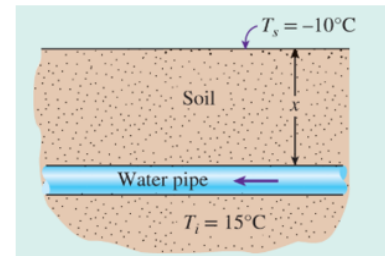
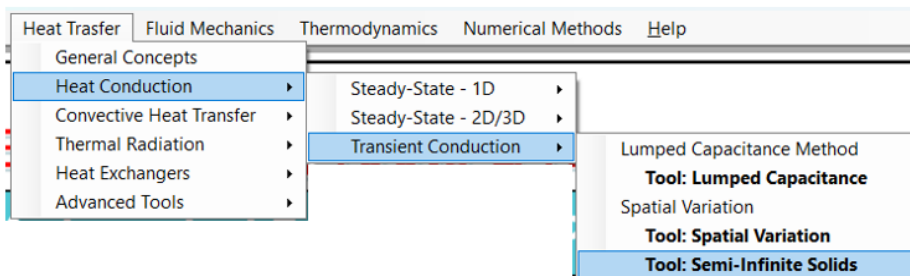
B) Temperature at center of the top surface

Update

Show Tutorial

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 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/m³
- C : 320 J/kg.K
- k : 0.4 W/m.K

Boundary Condition

Specified Temperature

T_s : -10 C

Heat Transfer Coef

h : W/m².K

T_{amb} : C

Uniform Heat Flux

q''_s : W/m²

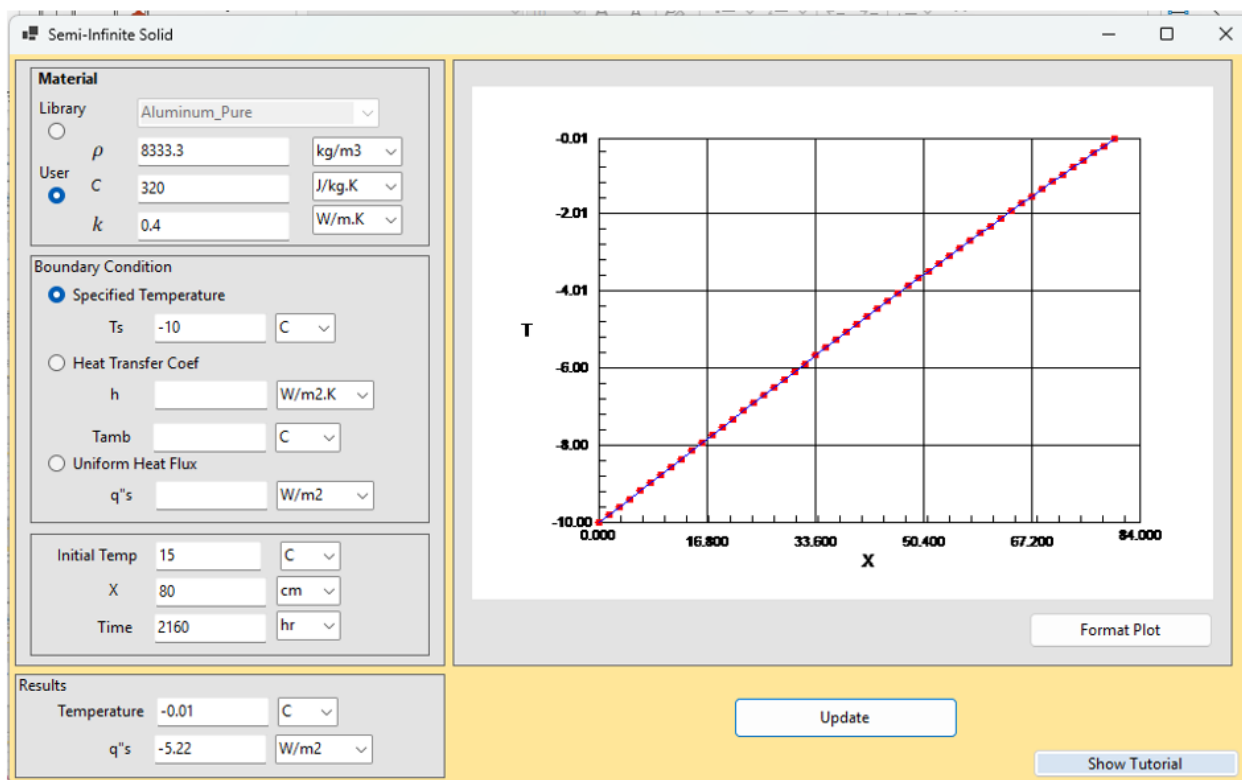
Initial Temp: 15 C

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 $x=500 \text{ cm}$, we can see the decay.

Material

Library

ρ 8333.3 kg/m³

User C 320 J/kg.K

k 0.4 W/m.K

Boundary Condition

Specified Temperature

T_s -10 C

Heat Transfer Coef

h W/m².K

T_{amb} C

Uniform Heat Flux

q''_s W/m²

Initial Temp 15 C

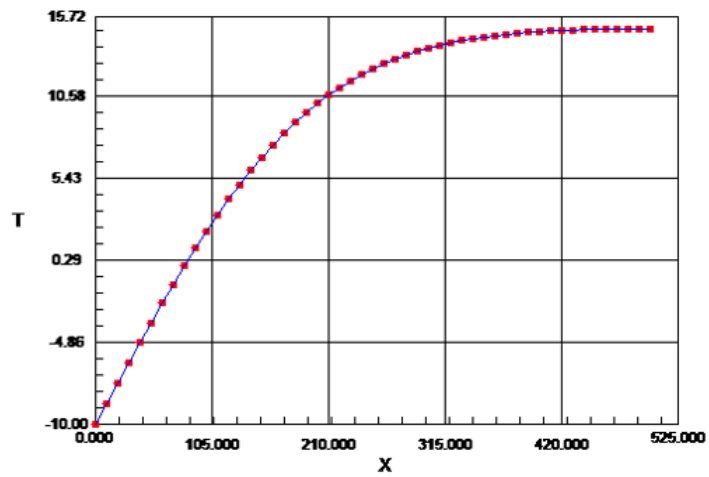
X 500 cm

Time 2160 hr

Results

Temperature 14.97 C

q''_s -5.22 W/m²



Format Plot

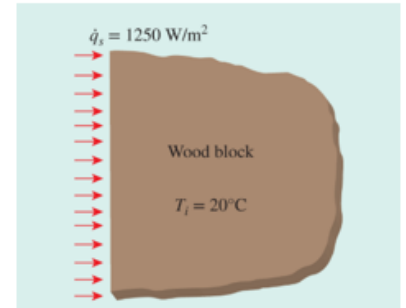
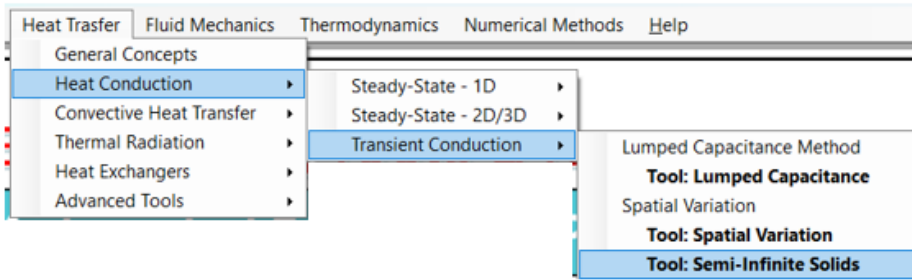
Update

Show Tutorial

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 block made of pure aluminum, and Plot results to 100 cm into the block.



Enter input data

Material			
Library	Aluminum_Pure		
<input type="radio"/>	ρ	721	kg/m ³
<input type="radio"/>	User	c	1260
<input checked="" type="radio"/>	k	0.159	W/m.K

Boundary Condition	
<input type="radio"/>	Specified Temperature
<input type="radio"/>	Heat Transfer Coef
<input checked="" type="radio"/>	Uniform Heat Flux

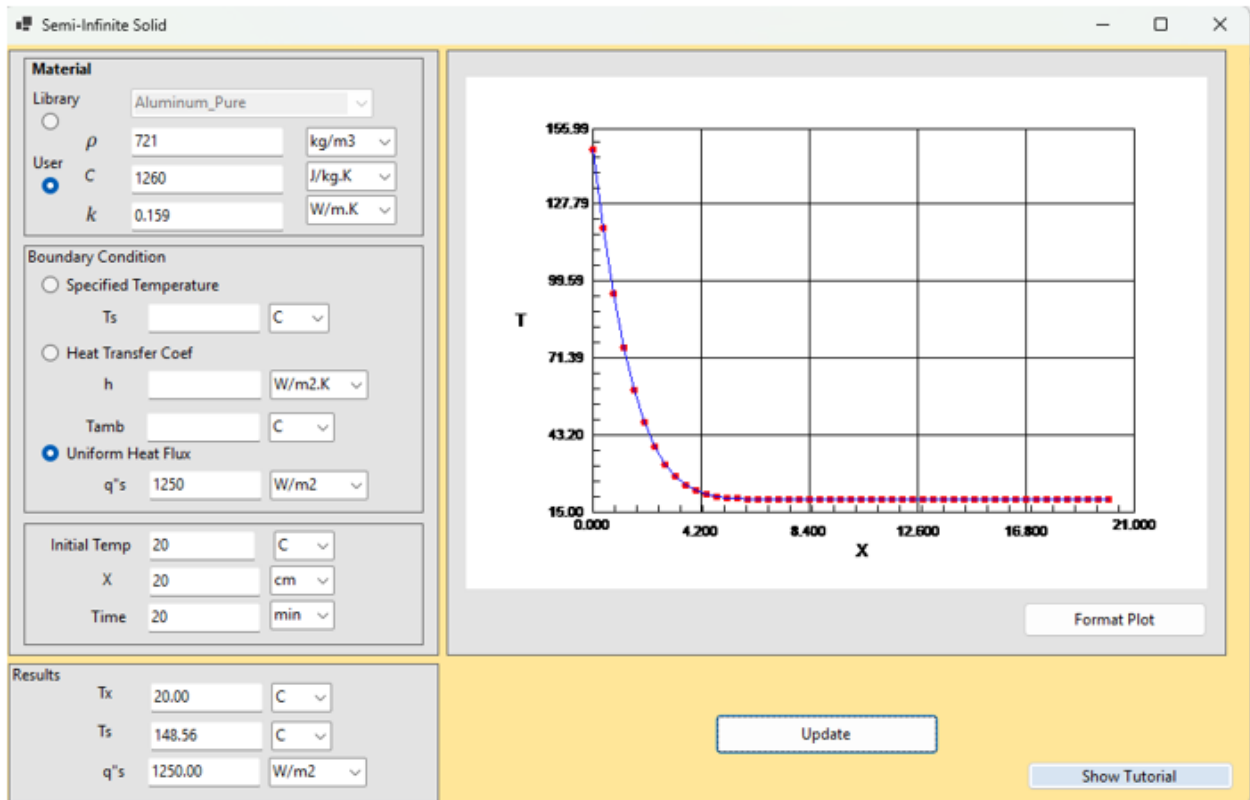
Initial Temp	20	C
X	0	cm
Time	20	min

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

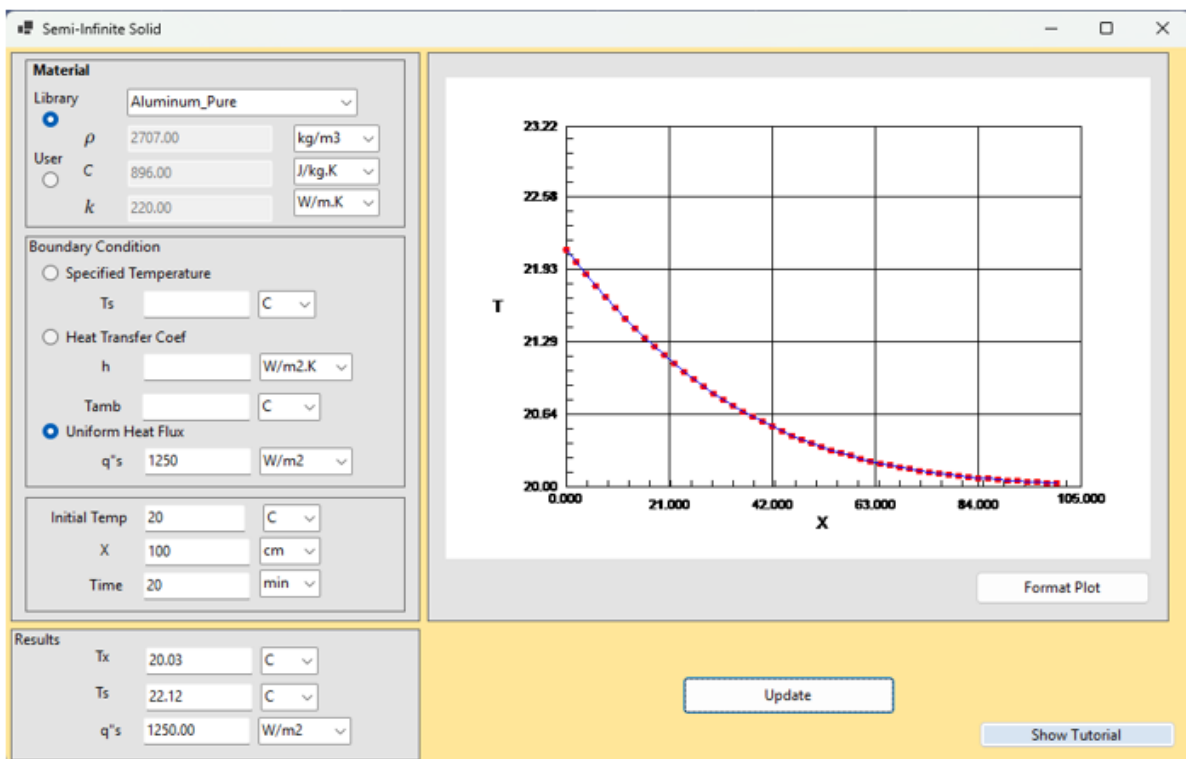
3. Choose "Uniform Heat Flux" BC and enter the given value.

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

Finished form is shown below.



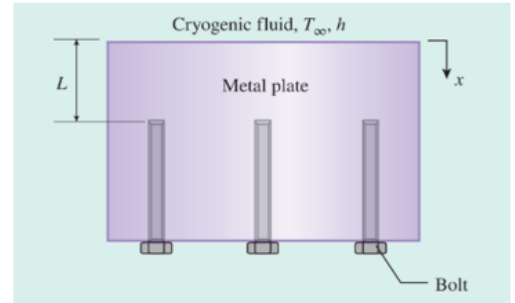
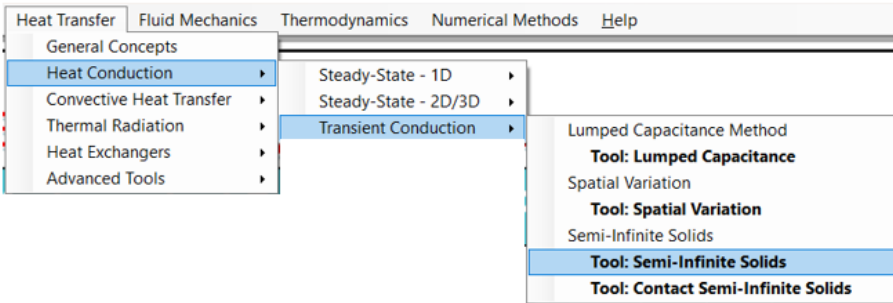
- Wood Block:**
1. The surface temperature is $148.6^\circ C$.
 2. The plot shows thermal penetration within the wooden block to the depth of 4.2 cm.



- 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/cm^3 . The upper surface of the plate is occasionally exposed to cryogenic fluid at -70°C with a convection heat transfer coefficient of $300 \text{ W/m}^2\text{.K}$. 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 $L = 1 \text{ cm}$. 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.

Material			
Library	User-Mat		
<input type="radio"/>	ρ	8	g/cm ³
<input checked="" type="radio"/>	User	C	500 J/kg.K
	k	16.3	W/m.K

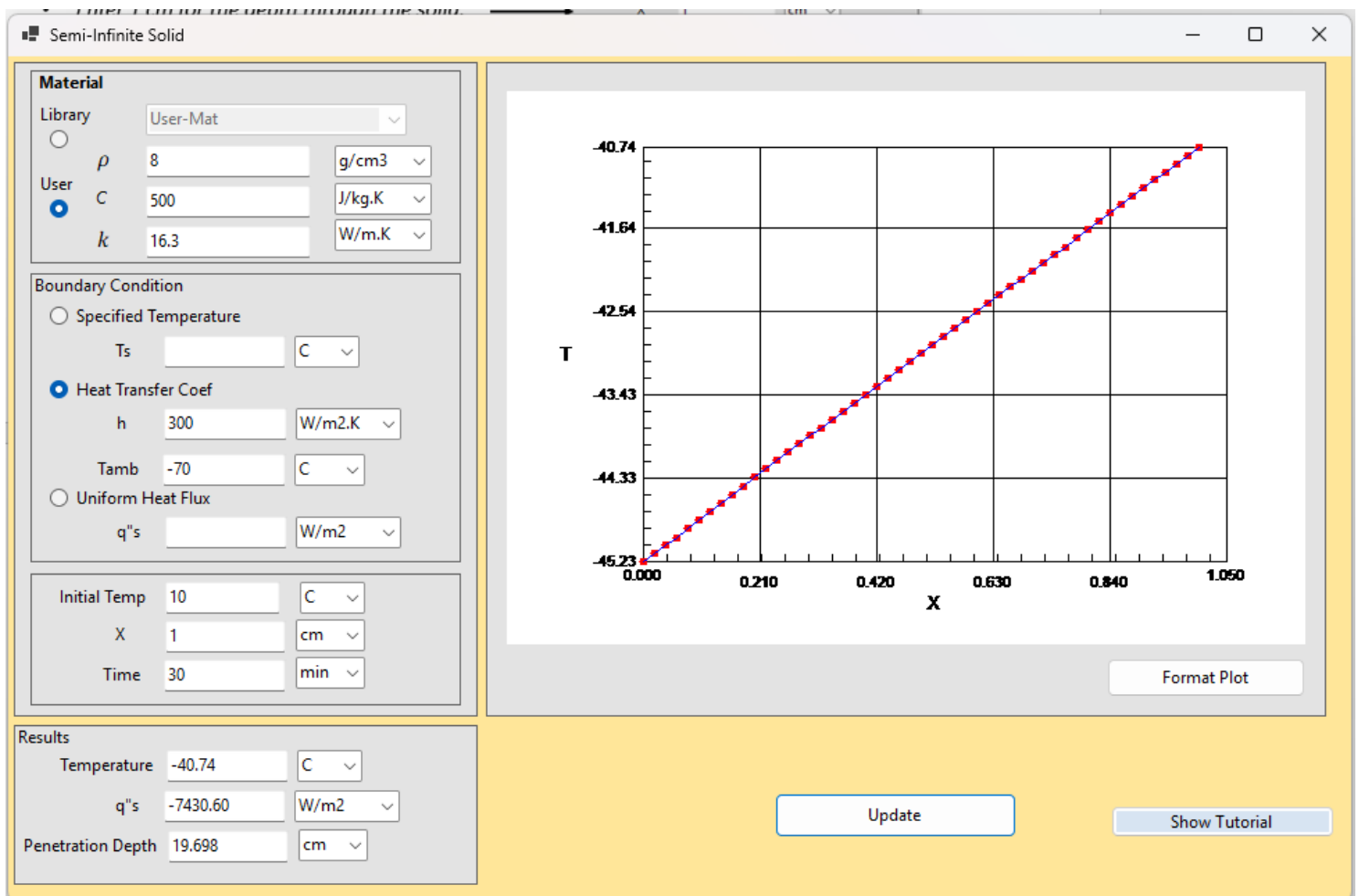
- Choose the 2nd type of boundary conditions and enter $300 \text{ W/m}^2\text{.K}$ for the heat transfer coefficient and -70° for the ambient temperature.

Boundary Condition			
<input type="radio"/>	Specified Temperature		
	T_s		C
<input checked="" type="radio"/>	Heat Transfer Coef		
	h	300	W/m ² .K
	T_{amb}	-70	C
<input type="radio"/>	Uniform Heat Flux		
	q''_s		W/m ²

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

	Initial Temp	10	C
	X	1	cm
	Time	30	min

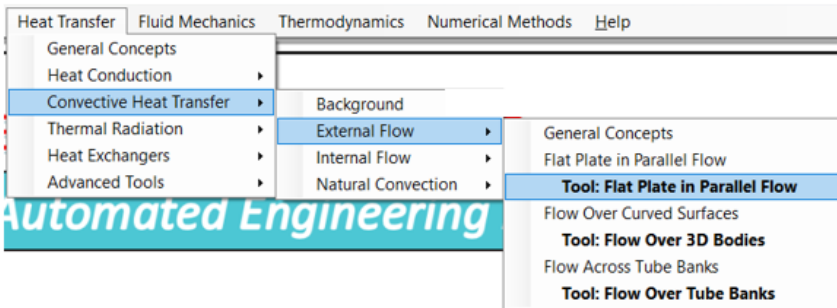
Click update to solve.



- 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.

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.

Fluid Properties

Select Fluid Enter Properties

Engine Oil

Pressure 1.000 atm

ρ kg/m³

C_p J/kg.K

k W/m.K

μ 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

Input Data

L 5 m

T_{inf} 60 C

U_{inf} 2 m/s

Uniform T_s 20 C

Uniform q''_s W/m²

Forced Convection - Flat Plate in Parallel Flow

Fluid Properties

Select Fluid Enter Properties

Engine Oil

Pressure 1.000 atm

ρ 875.98 kg/m³

C_p 1964.23 J/kg.K

k 0.144 W/m.K

μ 2.18E-01 Pa.s

Avg. Results

Flow Regime Laminar

Nu_avg 1910.4666

C_f _avg 0.0066

δ 124.5881 mm

h _avg 55.1628 W/m².K

Input Data

L 5 m

T_{inf} 60 C

U_{inf} 2 m/s

Uniform T_s 20 C

Uniform q''_s W/m²

Total Heat and Drag

Width 1.000 m

Q_{conv} -11032.5626 W

Drag 57.9733 N

Parameters

Re 4.0E04

Pr 2947.975

Pe 118700051.084

Local Results

X m

C_{fx}

Nu_x

h W/m².K

Plot Options

X-Axis

X

Re

Y-Axis

Nu Local

Nu Avg

h Local

h Avg

C_f Local

C_f Avg

δ

Update

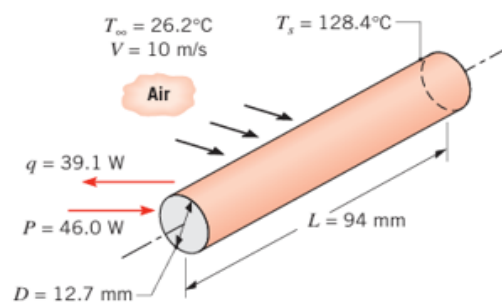
- The drag force is calculated to be 58 N.
- The rate of heat removal from the oil (heat addition to the plate) is roughly 11 kW.
- The plot shows the variation of the average heat transfer coefficient versus the distance to the leading edge (other properties may be selected).

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 by an electrical heater and is subjected to a cross flow of air in a low-speed wind tunnel. Under a specific set of operating conditions, which the upstream air velocity and temperature were maintained at $V = 10 \text{ m/s}$ and 26.2°C , respectively, the heater power dissipation was measured to be $P = 46 \text{ W}$, 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:

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

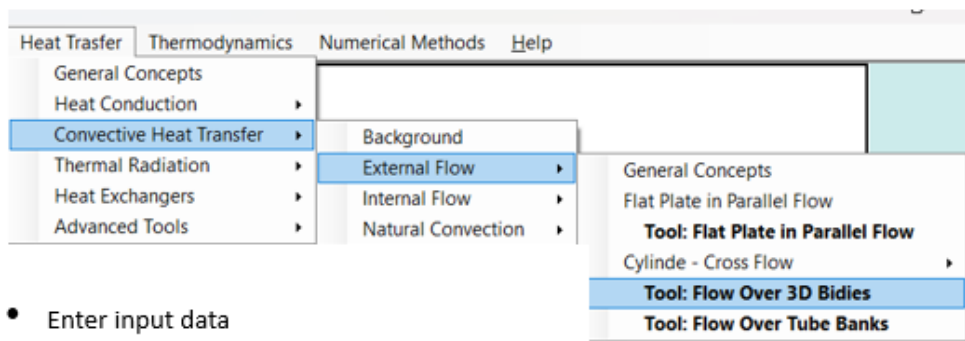
a) The convection coefficient can be evaluated from the experimental data, assuming 85% of the total heat dissipation dissipated from the surface of the rod:



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

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

- 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

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

Select Air from materials library

Enter Flow and Thermal Information

Enter Geometric Information

- 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

Noncircular Cylinders

c) Square

e) Hexagon

d) Tilted Square

f) Tilted Hexagon

g) Plate

Update

Comments:

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

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
 - Background
 - External Flow
 - 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
 - Thermal Radiation
 - Heat Exchangers
 - Advanced Tools
 - Natural Convection

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 4.5 m/s

T_i 20 C

T_s 120 C

D 1.5 cm

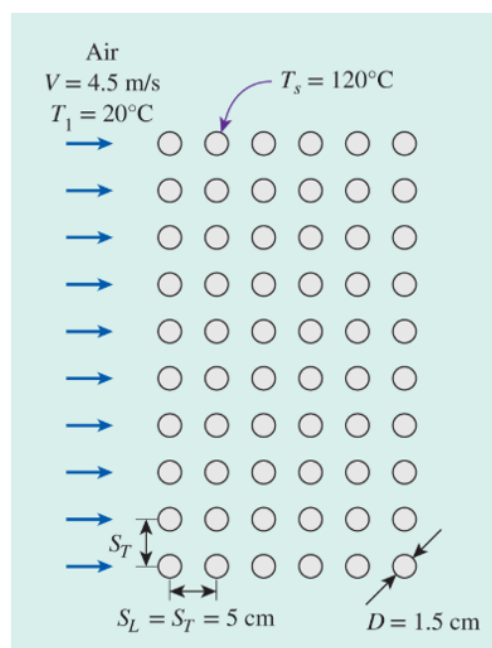
S_T 5 cm

S_L 5 cm

L 1.0 m

No of Columns - Longitudinal Dir, NL 6

No of Rows - Transverse Dir, NT 10



$$N_L = 6$$
$$N_T = 10$$
$$L = 1 \text{ m}$$

- 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_∞)

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

Diagram b) Staggered Arrangement

(V, T_∞)

Update Show Tutorial

Comments:

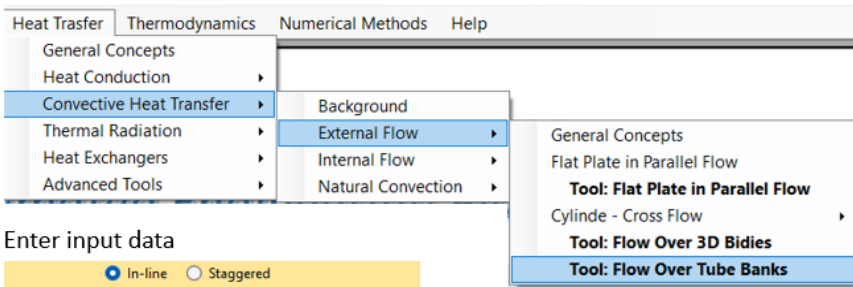
- The Reynolds number, $Re=6150$; it is based on V_{max}
- Average heat transfer coefficient is 96.2 W/m.K
- Air exit temperature is 29.6 °C
- Heat transfer into air is 25888.1 W (25.8 kW)
- Pressure drop across the bank can be calculated to be 23.2 Pa by multiplying the Dp/xf value of 145.66 Pa by the 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 is used to determine the mass flow rate through the bank.

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

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



Enter input data

In-line Staggered

Fluid Properties

Select Fluid Enter Properties

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-04 Pa.s

Pr 0.727 Prs 0.711

Input Parameters

V 5.2 m/s

T_i 20 C

T_s 100 C

D 1.6 cm

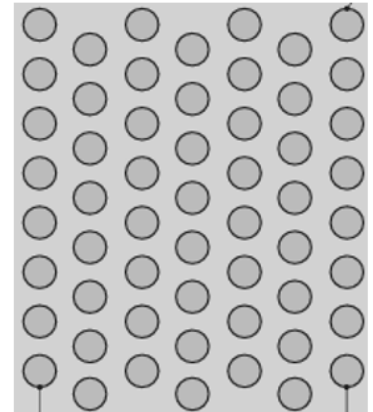
S_T 4 cm

S_L 4 m

L 3.0 m

No of Columns - Longitudinal Dir, NL 20

No of Rows - Transverse Dir, NT 10



Staggered Arrangement

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

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

$$N_T = 10; N_L = 20$$

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

$$T_i = 20 \text{ }^\circ\text{C}$$

$$T_s = 100 \text{ }^\circ\text{C}$$

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

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

- Click Update and Solve

In-line
 Staggered

Fluid Properties

 Select Fluid
 Enter Properties

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-01 Pa.s

Pr 0.727 Prs 0.711

Input Parameters

V 5.2 m/s

Ti 20 C

Ts 100 C

D 1.6 cm

ST 4 cm

SL 4 m

L 3.0 m

No of Columns - Longitudinal Dir, NL 20

No of Rows - Transverse Dir, NT 10

Results

Re 8379

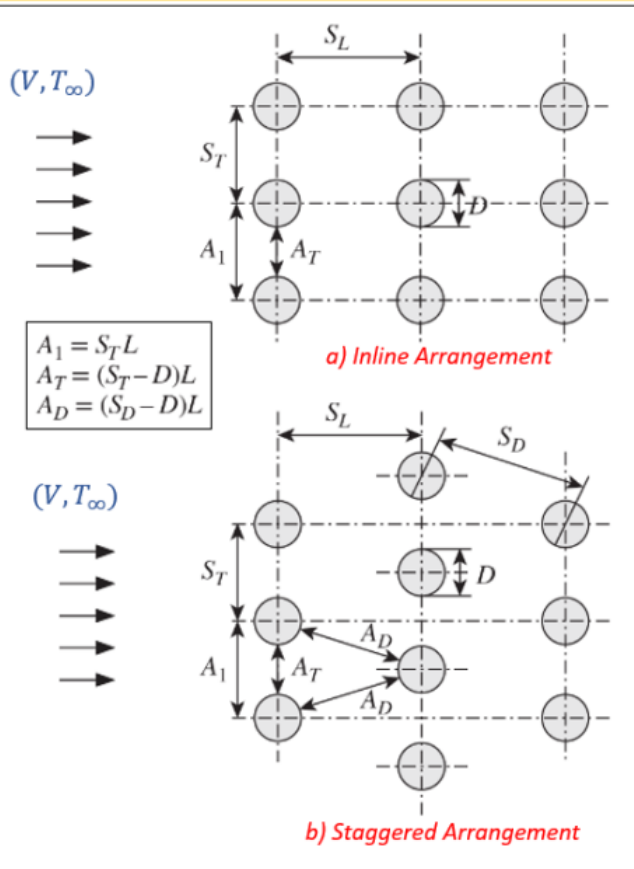
Nu_avg 71.7015

h_avg 117.6 W/m².K

Te 49.9 C

Q 226564.4 W

Dp/xf 860.022 Pa



Update

Show Tutorial

Comments:

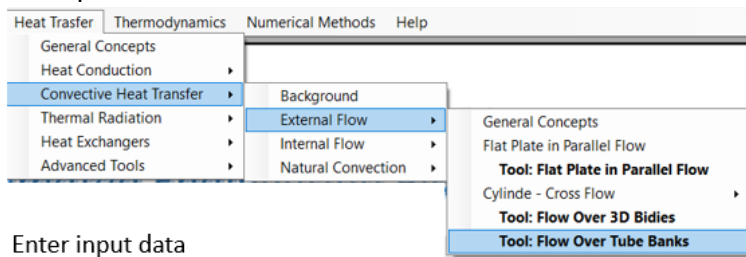
1. The Reynolds number, $Re=8379$ is based on V_{max}
2. Average heat transfer coefficient is 117.6 W/m.K
3. Air exit temperature is 49.9 °C
4. Heat transfer into air is 226564.4 W (22.7 kW)
5. Pressure drop across the bank can be calculated to be 283.8 Pa, by multiplying the Dp/xf value of 860.022 Pa by the product of the friction factor and the correction factor (roughly 0.33 and 1.0 from the chart).

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

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

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 Prs

Input Parameters

V 6 m/s

T_i 15 C

T_s 70 C

D 16.4 mm

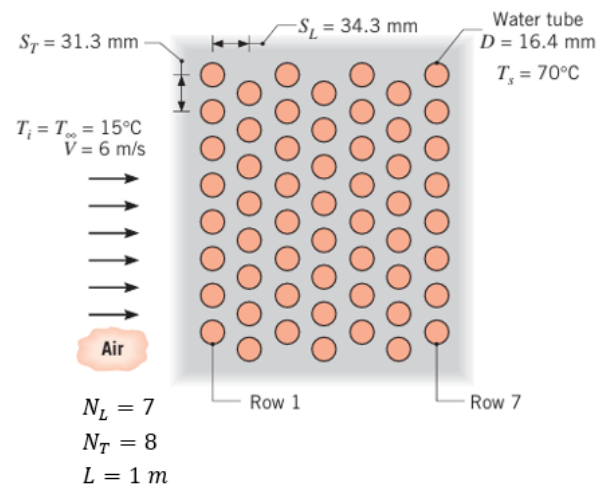
ST 31.3 mm

SL 34.3 mm

L 1.0 m

No of Columns - Longitudinal Dir, NL 7

No of Rows - Transverse Dir, NT 8



- 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 C_p 1006.87 J/kg.K

k 0.02578 W/m.K μ 1.8138E-01 Pa.s

Pr 0.709 Prs 0.701

Input Parameters

V 6 m/s

Ti 15 C

Ts 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

Update Show Tutorial

(V, T_∞)

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

a) Inline Arrangement

(V, T_∞)

b) Staggered Arrangement

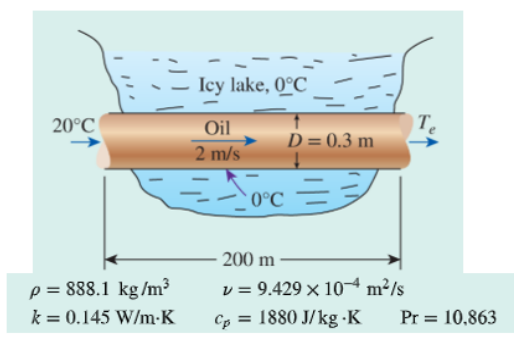
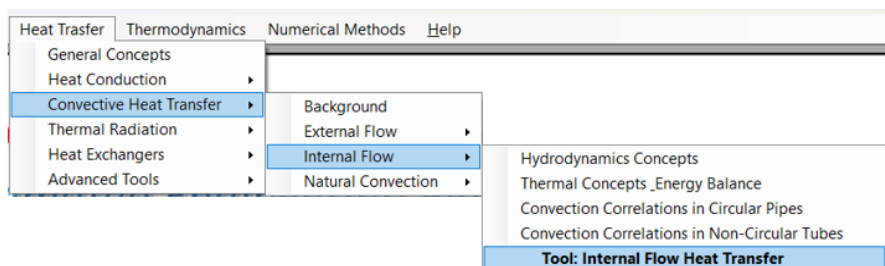
Comments:

1. The Reynolds number, Re , is 13,579, and it is based on V_{max}
2. Average heat transfer coefficient is 138.7 W/m.K
3. Air exit temperature is 25.7 °C
4. Heat transfer into air is 19,777 W (19.8 kW)
5. Pressure-drop across the bank can be calculated to be 241.15 Pa by multiplying 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.

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

m-dot V-dot U-in

kg/s cfm m/s

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

The screenshot displays a software interface for calculating heat transfer and fluid flow in a pipe. The interface is divided into several sections:

- Fluid Properties:** Select Fluid (Oil), Enter Properties, 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, 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_{in} , h_{ext} , K_w , th_w .
- Results:** Ac (7.07E-02 m²), Per (0.9425 m), Re (637), Pr (10852.14), Flow Regime (Laminar), 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), Rough wall, ϵ (mm), Adjust Properties for TWall Effects, Bhatti-Shah (Rough Wall), 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.

A schematic diagram at the top right shows a pipe with a thermal entry region where $T_s > T_{in}$ and a fully developed region where $T(r, 0) = T_s$. The diagram also shows velocity profiles $u(r)$ and q''_s at different axial positions.

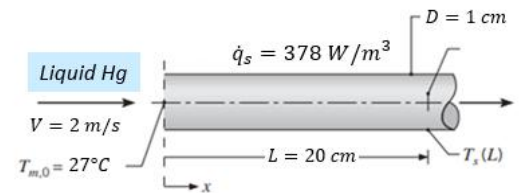
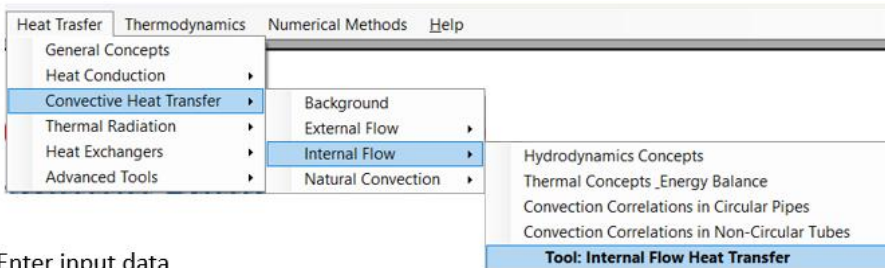
Comments:

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 m_{dot} .

Inlet Flow Information

\dot{m} kg/s

\dot{V} cfm

U_{in} 2 m/s

- Enter the 27 °C for Inlet Temperature.
- Select the “Uniform Heat Flux” option and enter 378 W/m²

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 “Include Thermal Entry Effects”.

Include Thermal Entry Effects

- Click Update to Solve.

Fluid Properties

Select Fluid Enter Properties

Air

Pressure: 1.000 atm

ρ : 1.1394 kg/m³ C_p : 1007.2647 J/kg.K

k : 0.0268 W/m.K μ : 1.877E-05 Pa.s

Geometric Information

Circular Non-Circular

D: 1 cm L: 20 cm

Inlet Flow Information

mdot: 1.790E-04 kg/s

Vdot: 0.33 cfm

Uin: 2 m/s

Thermal Conditions

Tin: 27 C

Uniform q''s: 378 W/m²

Uniform Ts

Uniform Tinf

h_ext: W/m².K

K_w: W/m.K

th_w: mm

Include Thermal Entry Effects

Results

Ac: 7.85E-05 m² Per: 0.0314 m

Re: 1214 Pr: 0.71

Flow Regime: Laminar

Flow Conditions: Velocity: Entry Region Velocity, Temperature: Entry Region Thermal

Average: Nu_avg: 8.7210, h_avg: 23.4 W/m².K, qtot: 2.4 W

Exit: Nu_o: 5.0225, h_o: 13.5 W/m².K, Tmo: 39.9 C, Tso: 68.0 C, q''s_o: 378.0 W/m²

Local Results: x/L: 0.5, Nu: 5.8960, Tm: 33.5 C, Ts: 57.4 C, h: 15.8 W/m².K, q''s_x: 378.0 W/m²

Correlation Corner

Turbulent Flow - Fully Developed

Gnielinsky Correlation (Smooth Wall) Rough wall

Dittus-Boelter Correlation (Smooth Wall) Adjust Properties for Twall Effects

Bhatti-Shah (Rough Wall)

Show Correlations

Results - Pressure Drop and Friction

Darcy Friction Factor, f: 0.0527

Friction Head Loss, hf: 0.2150 m

Pressure Drop: 0.0003 psi

Friction Factor: Haaland Equation, Swamee-Jain Equation, Colebrook Equation

DZ: 0.00 m

Update

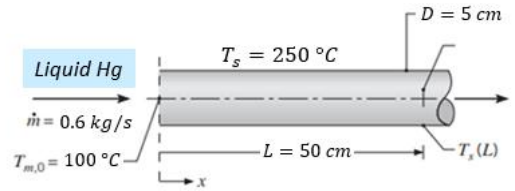
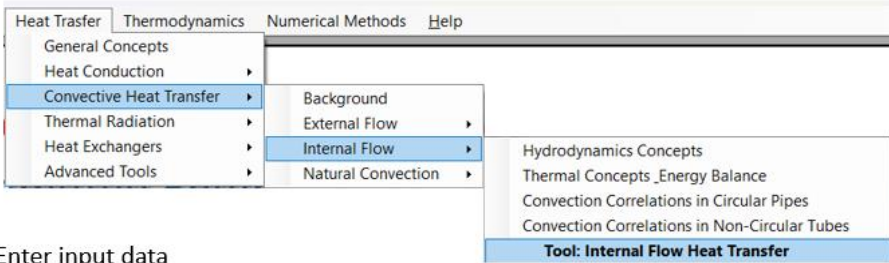
Comments:

1. The results indicate 40 °C for the exit mean temperature and 68 °C for the exit surface temperature..
2. The calculated Nu is 8.72, which is much larger than the fully developed value of 3.66. This indicates that the thermal entry effects were, in fact, important to include.

Example – Low Prandtl Number Flow – Liquid Mercury Flow in a Pipe [Source: Cengel-Ghajar 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.

- a) Determine the outlet mean temperature at $x=50$ cm.
- b) 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.
- Select the “Uniform Temperature” option and enter 250 °C

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

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: 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

\dot{m} : 0.6 kg/s

\dot{V} : 0.10 cfm

U_{in} : 0.02 m/s

Thermal Conditions

T_{in} : 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

T_{mo} : 201.6 C

T_{so} : 250.0 C

$q''_{s,o}$: 56259.1 W/m²

Local Results x/L : 0.5

Nu: 5.8168

T_m : 164.6 C

T_s : 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) Adjust Properties for Twall Effects

Bhatti-Shah (Rough Wall)

ϵ : mm

Show Correlations

Results - Pressure Drop and Friction

Friction Factor

Haaland Equation Swamee-Jain Equation Colebrook Equation

Darcy Friction Factor, f : 0.0000

Friction Head Loss, h_f : 0.0000 m

Pressure Drop: 0.0000 psi

DZ: 0.00 m

Update

Comments:

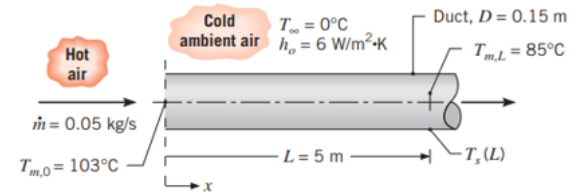
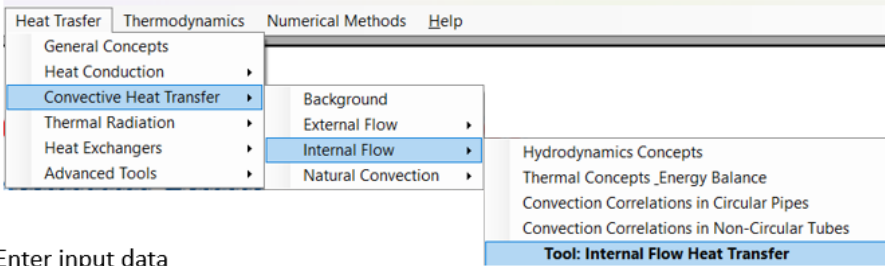
The uniform surface temperature liquid metal correlation is used.

- 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.
- 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 $D = 0.15$ m and $L = 5$ m, which is in the crawlspace of a house. The heat transfer coefficient between the duct outer surface and the ambient air at $T_\infty = 0^\circ\text{C}$ is known to be $h_{\text{ext}} = 6$ W/(m² · 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.

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 $D=0.15$ m and $L=5$ m

Geometric Information

Circular Non-Circular

D 0.15 m L 5 m

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

Inlet Flow Information

\dot{m} 0.05 kg/s

\dot{V} cfm

U_{in} m/s

- Enter the 103 °C for Inlet Temperature.

Thermal Conditions

T_{in} 103 C

Uniform q''_s W/m²

Uniform T_s C

Uniform T_{inf} 0 C

h_{ext} 6.0 W/m².K

K_w 400 W/m.K

th_w 1 mm

- Select the “Uniform Ambient Temperature” option and enter 0 °C
- Enter 6.0 W/m².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: 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

mdot: 0.05 kg/s Vdot: 111.27 cfm Uin: 2.97 m/s

Thermal Conditions

Tin: 103 C

Uniform q''_s Uniform Ts Uniform Tinf: 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 $qtot$: -894.6 W

Exit

Nu_o: 56.0550 h_o : 11.7 W/m2.K Tm_o : 85.3 C Ts_o : 56.4 C q''_s_o : -338.3 W/m2

Local Results x/L : 0.5

Nu: 54.2765 Tm : 93.7 C Ts : 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) Rough wall ϵ : mm

Dittus-Boelter Correlation (Smooth Wall) Adjust Properties for Twall Effects

Bhatti-Shah (Rough Wall)

Results - Pressure Drop and Friction

Darcy Friction Factor, f: 0.0262

Friction Head Loss, hf: 0.3935 m

Pressure Drop: 0.0005 psi

Friction Factor: Haaland Equation Swamee-Jain Equation Colebrook Equation

DZ: 0.00 m

Comments:

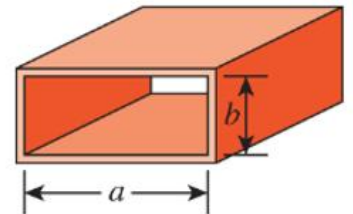
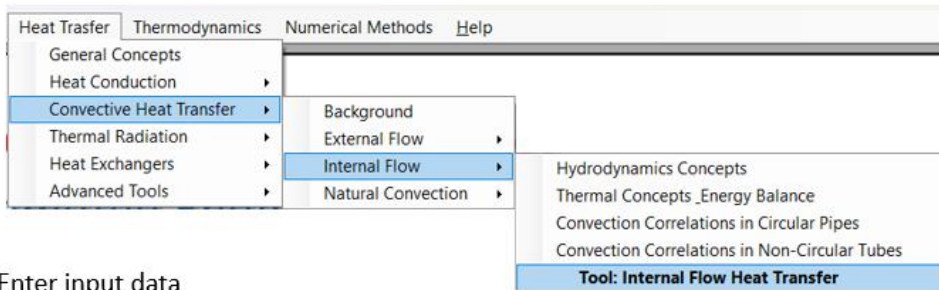
The Dittus-Boelter correlation is used to be consistent with the approach in the textbook.

5. The results show that roughly 900 W of heat loss from the duct in the basement.
6. 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

- hydraulic diameter for the duct,
- Reynolds number,
- Nusselt number,
- convection heat transfer coefficient,
- air outlet temperature,



Enter input data

- Select "Enter Properties" and enter values.

The 'Fluid Properties' dialog box is shown. It has two radio buttons: 'Select Fluid' (unselected) and 'Enter Properties' (selected). The fluid is set to 'Air'. The pressure is 1.000 atm. The density ρ is 0.995 kg/m³, the specific heat C_p is 1009 J/kg·K, the thermal conductivity k is 0.03 W/m·K, and the dynamic viscosity μ is 0.0000208 Pa·s.

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

The 'Geometric Information' dialog box is shown. It has two radio buttons: 'Circular' (unselected) and 'Non-Circular' (selected). The hydraulic diameter D_h is set to a value in cm, and the length L is set to a value in m.

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

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



Geometric Information

Circular Non-Circular

Dh 10.0000 cm L 2 m

frmConvInt_Shapes

θ [] deg
 s [] cm
 a [15] mm
 $\frac{a}{b} > 8$
 $\frac{a}{b} = 1$
 $\frac{a}{b} = 1$
 $\frac{a}{b} = 1$
 $\frac{a}{b} = 1$
 $\frac{a}{b} = 1$

OK

- Enter 2 m for the length.

- Enter 0.1 kg/s for mdot.

Inlet Flow Information

mdot 0.1 kg/s
 Vdot [] cfm
 Uin [] m/s

- Enter the 2 °C for Inlet Temperature.

- Select the "Uniform Surface Temperature" option and enter 127 °C

Thermal Conditions

Tin 12 C
 Uniform q"s [] W/m2
 Uniform Ts 127 C
 Uniform Tinf [] 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)

Dittus-Boelter Correlation (Smooth Wall)

Bhatti-Shah (Rough Wall)

Rough wall

ϵ mm

Adjust Properties for TWall Effects

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

\dot{m} 0.1 kg/s

\dot{V} 212.95 cfm

U_{in} 8.93 m/s

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

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

q_{tot} 2747.8 W

Exit

Nu_o 100.9923

h_o 30.3 W/m².K

T_{mo} 39.2 C

T_{so} 127.0 C

$q''_{s,o}$ 2659.1 W/m²

Local Results x/L 0.5

Nu 100.9923

T_m 26.5 C

T_s 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)

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 0.0217

Friction Head Loss, h_f 1.7656 m

Pressure Drop 0.0025 psi

Friction Factor

Haaland Equation

Swamee-Jain Equation

Colebrook Equation

DZ 0.00 m

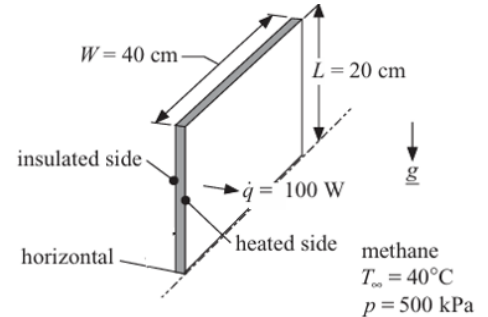
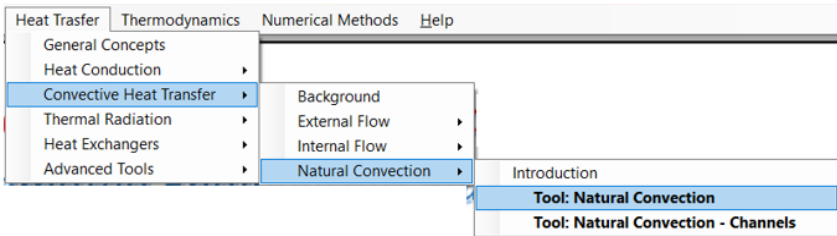
Update

Diagram: A schematic of a pipe showing the thermal entry region and the fully developed region. The inlet temperature is $T_s > T_{in}$. The wall heat flux is q''_s . The inlet temperature profile is $T(r, 0)$ and the outlet temperature profile is $T(r, 0) = T_s$. The thermal entry region is indicated by a blue arrow, and the fully developed region is indicated by a red arrow.

HT-16: Natural Convection over Bodies

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

A rectangular plate heater is placed in the ullage space of a fuel tank on a military aircraft (as shown below). One side of the heater is insulated, and the other is heated. The heater is oriented vertically with respect to gravity and achieves a nearly uniform temperature. The length of the heater is $L = 20\text{ cm}$, and the width is $W = 40\text{ cm}$. The plate is exposed to fuel that has properties consistent with methane at $T_\infty = 40\text{ }^\circ\text{C}$. Assuming a heater power of 100 W , determine the surface temperature of the heater for fuel at $p = 500\text{ kPa}$, 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 <input checked="" type="radio"/> Select Fluid <input type="radio"/> Enter Properties Methane 500 kPa ρ 2.7880 kg/m ³ 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 m ² Per 1.2000 m Grashof No N/A Rayleigh No N/A <input type="checkbox"/> Fix Prandtl No 0.72 <input type="checkbox"/> Fix									
Geometric Information Select Geometry Case g: Inclined Plate L 20 cm W 40 cm θ 60 deg	Results Nu_avg 0.000 h_avg 18.7 W/m ² .K Q_conv 99.95 W Q_rad 0.00 W Q_tot 99.95 W U_char 0.62 m/s									
Gravitational Acceleration Gravity 9.81 m/s ²	Correlation Selector <table border="1" style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 25%;"> Vertical Flat Plate (a) <input checked="" type="radio"/> Eq: A1/A2-Churchill Chu <input type="radio"/> Eq: A.3 <input type="radio"/> Eq: A.4-Nellis, Klein (EES) </td> <td style="width: 25%;"> Horizontal Plate Upward (b) <input type="radio"/> Eq: A.5 <input type="radio"/> Eq: A.6-Nellis, Klein (EES) </td> <td style="width: 25%;"> Vertical Cylinder (d) CV1 Vertical Wall with Correction </td> <td style="width: 25%;"> Sphere (f) <input type="radio"/> Sparrow-Gregg </td> </tr> <tr> <td> Horizontal Plate Downward (c) <input type="radio"/> Eq: A.7 <input type="radio"/> Eq: A.8-Nellis, Klein (EES) </td> <td> Horizontal Cylinder (e) <input type="radio"/> CH11/CH12 (EES) <input type="radio"/> CH11-Churchill Chu </td> <td colspan="2"> Inclined Plate (g) <input checked="" type="radio"/> Eq: A.13 Raithby and Holland method </td> </tr> </table>		Vertical Flat Plate (a) <input checked="" type="radio"/> Eq: A1/A2-Churchill Chu <input type="radio"/> Eq: A.3 <input type="radio"/> Eq: A.4-Nellis, Klein (EES)	Horizontal Plate Upward (b) <input type="radio"/> Eq: A.5 <input type="radio"/> Eq: A.6-Nellis, Klein (EES)	Vertical Cylinder (d) CV1 Vertical Wall with Correction	Sphere (f) <input type="radio"/> Sparrow-Gregg	Horizontal Plate Downward (c) <input type="radio"/> Eq: A.7 <input type="radio"/> Eq: A.8-Nellis, Klein (EES)	Horizontal Cylinder (e) <input type="radio"/> CH11/CH12 (EES) <input type="radio"/> CH11-Churchill Chu	Inclined Plate (g) <input checked="" type="radio"/> Eq: A.13 Raithby and Holland method	
Vertical Flat Plate (a) <input checked="" type="radio"/> Eq: A1/A2-Churchill Chu <input type="radio"/> Eq: A.3 <input type="radio"/> Eq: A.4-Nellis, Klein (EES)	Horizontal Plate Upward (b) <input type="radio"/> Eq: A.5 <input type="radio"/> Eq: A.6-Nellis, Klein (EES)	Vertical Cylinder (d) CV1 Vertical Wall with Correction	Sphere (f) <input type="radio"/> Sparrow-Gregg							
Horizontal Plate Downward (c) <input type="radio"/> Eq: A.7 <input type="radio"/> Eq: A.8-Nellis, Klein (EES)	Horizontal Cylinder (e) <input type="radio"/> CH11/CH12 (EES) <input type="radio"/> CH11-Churchill Chu	Inclined Plate (g) <input checked="" type="radio"/> Eq: A.13 Raithby and Holland method								
Thermal Conditions Emissivity 0.0 T_inf 40 C <input type="radio"/> Specified Ts 106.9 C <input checked="" type="radio"/> Specified Q_tot 100.0 W	<input type="button" value="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
 T_inf 40 C
 Specified Ts 91.8 C
 Specified Q_tot 100.0 W

Natural Convection

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

Fluid Properties

Geometric Information

Select Geometry

Case g: Inclined Plate

L .2 m

W .4 m

θ 60 deg

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

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

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

Sphere (f)

Sparrow-Gregg

Horizontal Cylinder (e)

CH11/CH12 (EES)

CH11-Churchill Chu

Inclined Plate (g)

Eq: A.13

Raithby and Holland method

Case Diagrams:

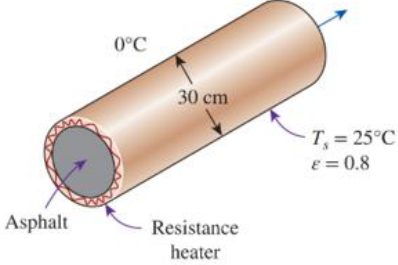
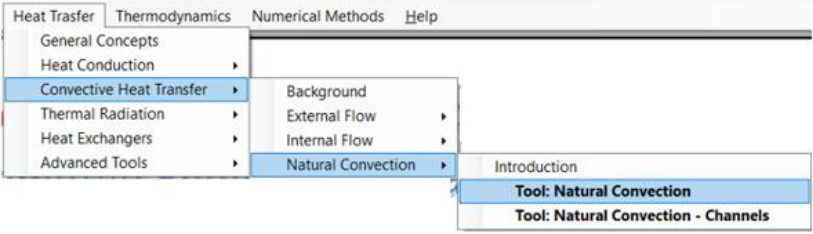
- Case a: Vertical Plate Heated or Cooled
- Case b: Horizontal Plate Upward Heated or Downward Cooled
- Case c: Horizontal Plate Downward Heated or Upward Cooled
- Case d: Vertical Cylinder Heated or Cooled
- Case e: Long Horizontal Cylinder Heated or Cooled
- Case f: Long Horizontal Cylinder Heated or Cooled (Sphere)
- Case g: Inclined Plate

Note:

- The surface temperature 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

Fluid Properties

Select Fluid Enter Properties

Air 1 atm

ρ kg/m³ C_p J/kg.K

k W/m.K μ Pa.s

β 1/K

Geometric Information:

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

Geometric Information

Select Geometry

Case e: Long Horizontal Cylinder

L 100 m

D 30 cm

Thermal Conditions:

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

Thermal Conditions

Emissivity 0.8

Tinf 0 C

Specified Ts 25 C

Specified Qtot 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
 A_s 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

Results
 Nu_{avg} 52.374
 h_{avg} 4.4 W/m².K
 Q_{conv} 10346.05 W
 Q_{rad} 9983.38 W
 Q_{tot} 20329.43 W
 U_{char} 0.51 m/s

Gravitational Acceleration
 g 9.81 m/s²

Thermal Conditions
 ϵ 0.8
 T_{inf} 0 C
 Specified T_s 25 C
 Specified Q_{tot} 20329.4 W

Correlation Selector

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

Case Diagrams:
 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 (Sphere)
 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.

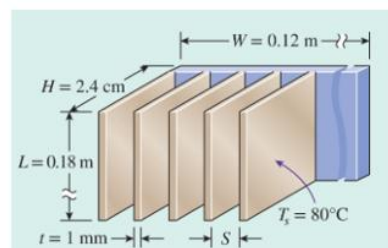
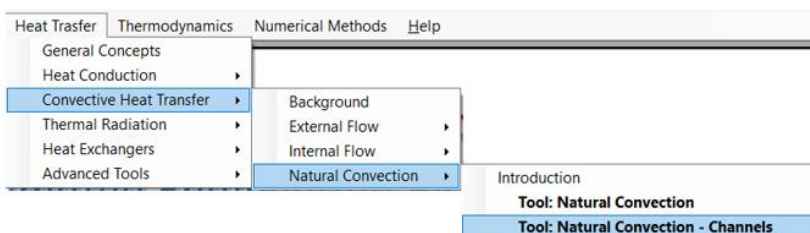
Example – Heat Sink Fin Spacing – [Source: Cengel-Ghagar Example 9.3]

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.

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

Solution:

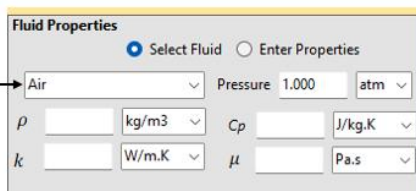
a)



Enter input data

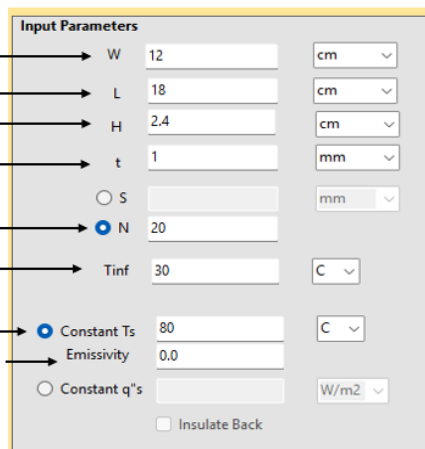
Enter Fluid Properties:

- Select "Air" from Fluid Library.



Enter Input:

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



Note:

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

Click "Update" to Solve.

Completed Form:

Fluid properties at T_{film} are displayed.

Fin spacing is calculated and displayed.

Rayleigh number, Nusselt number and the heat transfer coefficient.

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

Calculated Heat Transfer rate.

b)

To use optimal fin spacing:

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

Leave all other parameters unchanged.

Click **“Update”** to Solve.

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

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

c)

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

→ Constant Ts 80 C
Emissivity 0.25

Click "Update" to Solve.

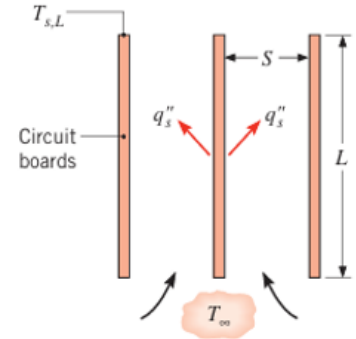
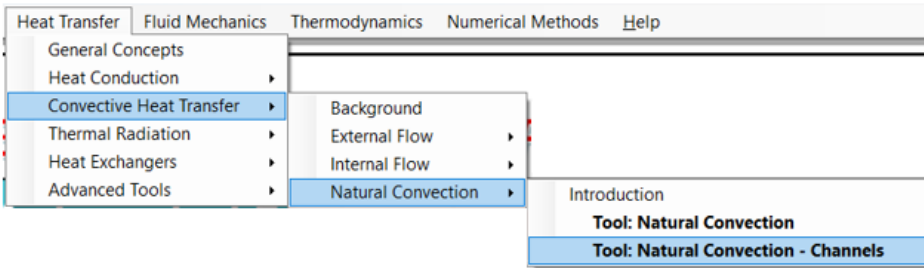
- This will result in an additional 5.9 W heat loss due to radiation, bringing the total heat transfer rate to ambient to 35.8 W.

Results	
Ra	1301
Nu	1.3066
h	4.9 W/m ² .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 than 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

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

- 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.

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 T_s Emissivity 0.0 C

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

Qcncv W

Qrad W

Qtot 2024.0 W

Tmax 74.5 C

Update

Show Tutorial

a) Constant T_s

b) Constant q_s

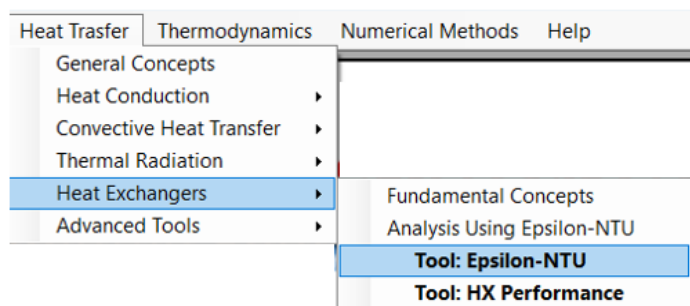
- The maximum board temperature will be 74.5 °C.
- The total heat rate will be 2,024 W.

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).

The 'Input' panel contains the following fields and options:

- A dropdown menu for flow arrangement, currently showing 'd) Shell & Tube: n Shell pass and 2n, 4n, ... tube pass'.
- A text input field for 'Cr' (Capacity Ratio).
- A text input field for 'No. of Shell passes'.
- Two radio button options: 'NTU Given' (selected) and 'Epsilon Given'.
- Two text input fields corresponding to the radio button options.

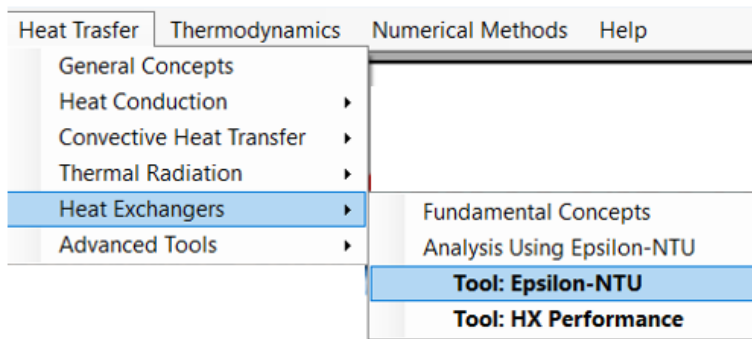
Heat Exchanger Flow Arrangement	Capacity Ratio, C_r	NTU	Epsilon
Parallel Flow	0.5	2.5	0.651
Counter Flow	0.5	2.5	0.8328
Shell-Tube 2 Tube Passes; Single Shell Pass	0.5	2.5	0.7237
Shell-Tube 2 Tube Passes; 2 Shell Pass	0.5	2.5	0.802
Cross-Flow Both Fluids Unmixed	0.5	2.5	0.7911
Cross-Flow Both Fluids 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.

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

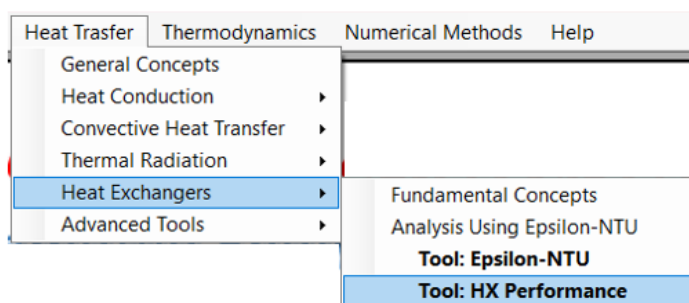
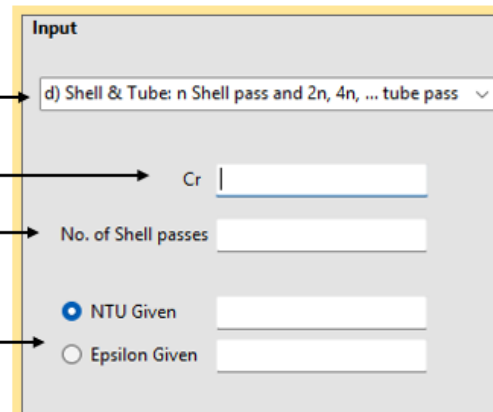
The cross-flow heat exchanger with both fluids unmixed is used to heat air with hot water. Water enters the heat exchanger tubing with a mass flow rate of 0.03 kg/s and 60 °C. Air at 20 °C and atmospheric pressure is blown across the heat exchanger with a volumetric flow rate of 0.06 m³/s. The heat exchanger's 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:

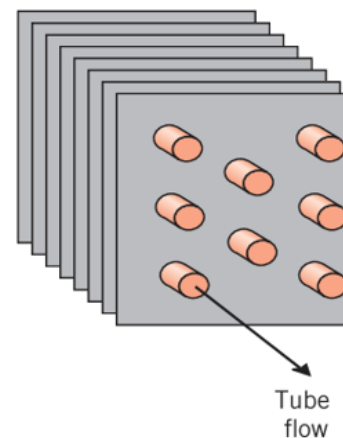


Tool utilizes a very simple user input panel shown below:

- 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).



Enter input data



Hot Side

Cold Side

Select Water →

Enter \dot{m}_h →

Enter inlet temp →

Select Air →

Enter \dot{V}_c →

Enter inlet temp →

Select HX Type from list →

Select UA for input and enter the value →

Click "Solve".

Heat Exchanger Performance Analysis

Hot Fluid

Properties: Select Fluid Enter Props
Water

ρ 979.50 kg/m³
 C_p 4187.96 J/kg.K

Inlet Conditions: \dot{m}_{dotH} 0.03 kg/s
 \dot{V}_{dotH} cfm
 T_{hot_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: \dot{m}_{dotC} kg/s
 \dot{V}_{dotC} 0.06 m³/s
 T_{cold_in} 20 C

HX Details

e) Cross Flow: both fluid unmixed

UA 58.40 W/K
 T_{hot_out} 49.1 C
 T_{cold_out} 38.9 C
 Q 1365.2 W

Qmax 2883.7 W
Cr 0.57
Epsilon 0.47
NTU 0.81

Solve

Show Tutorial

$C_{max} = \max(C_h, C_c)$
 $C_{min} = \min(C_h, C_c)$
 $C_r = \frac{C_{min}}{C_{max}} \leq 1$

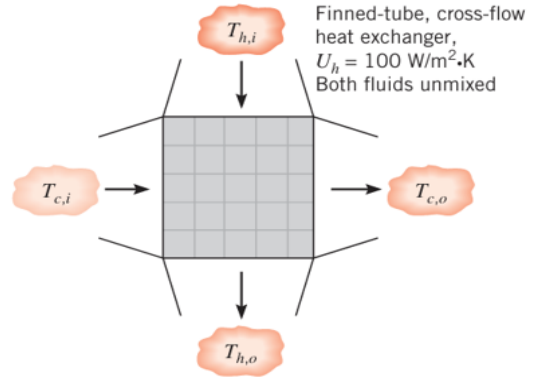
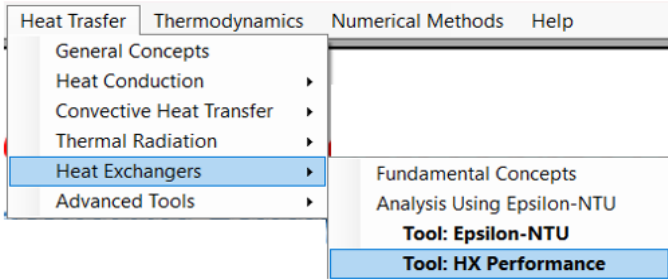
$q_{max} = C_{min}(T_{h,i} - T_{c,i})$
 $\epsilon = \frac{q}{q_{max}} = \frac{C_h(T_{h,i} - T_{h,o})}{C_{min}(T_{h,i} - T_{c,i})}$
 $NTU = \frac{UA}{C_{min}}$

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

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

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

mdotC 1 kg/s

VdotC cfm

T_{cold_in} 35 C

Select HX Type from list

HX Details

e) Cross Flow: both fluid unmixed

Select outlet temp for the gas

UA W/K

T_{hot_out} 100 C

T_{cold_out} C

Q W

Click "Solve".

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
 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
 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 and 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

- 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:

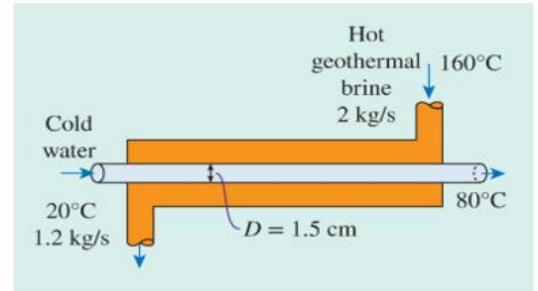
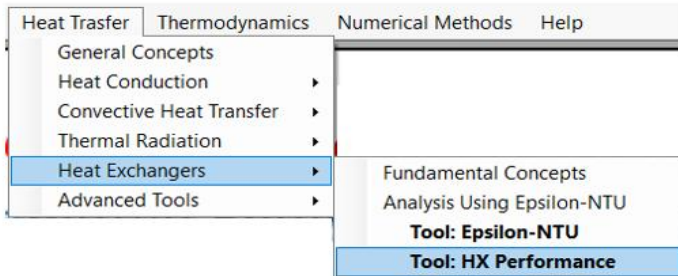
$$A_h = \frac{UA}{U_h} = \frac{3833}{100} \rightarrow A_h = 38.33 \text{ m}^2$$

- The outlet temperature of water is 125.5 °C.
- The total rate of heat transfer between hot gas and water is 378 kW.
- The maximum heat transfer rate (Q_{max}) is 500850 W.
- The capacity ratio is calculated to be 0.45.
- The heat exchanger effectiveness is calculated to be 0.75.
- 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 $c_p = 4310 \text{ J/kg} \cdot \text{K}$). The inner tube is thin-walled and has a diameter of 1.5 cm. The overall heat transfer coefficient of the heat exchanger is $640 \text{ W/m}^2 \cdot \text{K}$, determine the length of the heat exchanger required to achieve the desired heating.

Solution:



Enter input data

Hot Side

Cold Side

Hot Fluid

Properties: Select Fluid Enter Props

Fluid: Geothermal H2O

ρ : kg/m³

C_p : 4310 J/kg.K

Inlet Conditions:

\dot{m} (kg/s): 2

\dot{V} (cfm)

T_{hot_in} : 160 C

Cold Fluid

Properties: Select Fluid Enter Props

Fluid: Water

ρ : kg/m³

C_p : J/kg.K

Inlet Conditions:

\dot{m} (kg/s): 1.2

\dot{V} (cfm)

T_{cold_in} : 20 C

HX Details

Select HX Type from list: b) Counter Flow

Select outlet temp for water: T_{cold_out} : 80 C

UA: W/K

T_{hot_out} : C

Q: W

Click "Solve".

Heat Exchanger Performance Analysis

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
 Thot_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
 Tcold_in 20 C

HX Details
 b) Counter Flow

UA 3274.19 W/K (1)
 Thot_out 125.1 C (2)
 Tcold_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

$C_{max} = \max(C_h, C_c)$
 $C_{min} = \min(C_h, C_c)$
 $C_r = \frac{C_{min}}{C_{max}} \leq 1$

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

Show Tutorial

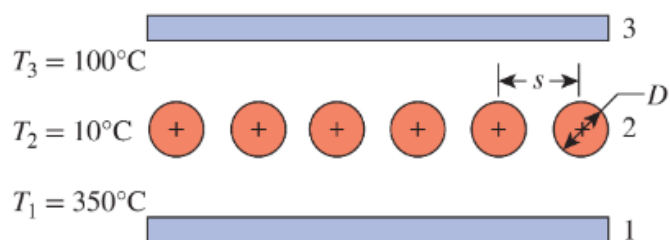
1. The UA is calculated to be 3274.2 W/K.
 Since U is given in the problem statement to be 640 W/m².K, The length can be found:

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

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 →

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