Problem #1. Simple wall:

Consider a 3 m high, 5 m wide, and L = 0.3 m thick wall whose thermal conductivity is k = 0.9 W/m·K (Figure 1). On a certain day, the temperatures of the inner and the outer surfaces of the wall are measured to be T_1 = 16°C and T_2 = 2°C, respectively. Determine the rate of heat loss through the wall on that day.

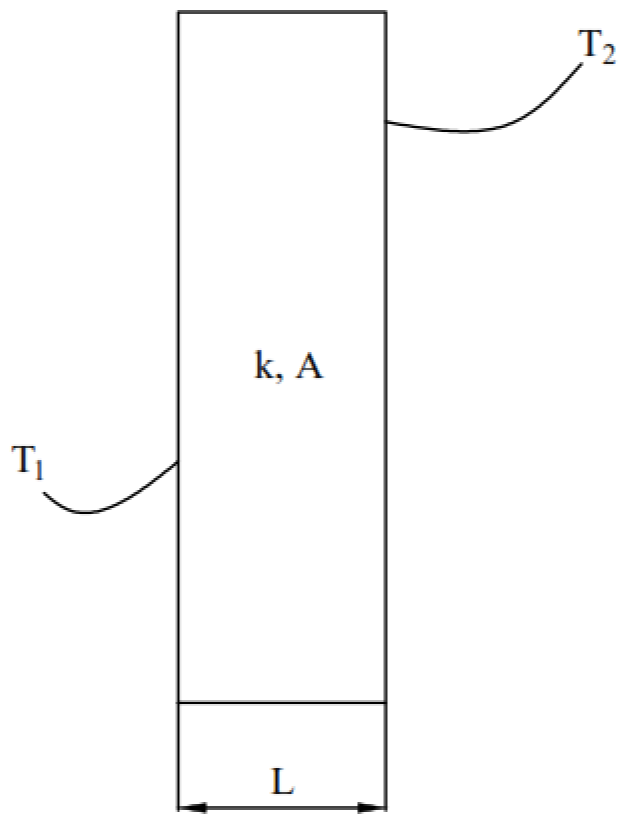

Figure 1. Simple wall with conduction heat transfer, and prescribed temperature on both sides

Analytical solution:

Applying Fourier Law:

$$\dot{Q} = \frac{kA}{L} \cdot (T_1 - T_2)$$

Substituting the problem data:

$$\dot{Q} = \frac{0.9\ \mathrm{W}/_{\mathrm{m}\cdot{}^{\circ}\mathrm{C}} \cdot 15\ \mathrm{m}^2}{0.3\ \mathrm{m}} \cdot (16-2)^{\circ}\mathrm{C} = 630\ \mathrm{W}$$

Now, we can calculate the temperature at x = L/2:

$$\dot{Q} = \frac{kA}{L/2} \cdot (T_1 - T_m)$$

$$T_m = T_1 - \frac{\dot{Q} \cdot L/2}{kA}$$

Substituting the problem data:

$$T_m = 16 - \frac{630\ \mathrm{W} \cdot 0.15\ \mathrm{m}}{0.9\ \mathrm{W}/_{\mathrm{m}\cdot{}^{\circ}\mathrm{C}} \cdot 15\ \mathrm{m}^2} = 9^{\circ}\mathrm{C}$$

Finite Element Model

We'll use two elements to model the problem #1, so we can check the rate of heat loss and the middle temperature of the wall. The model is shown on figure 2.

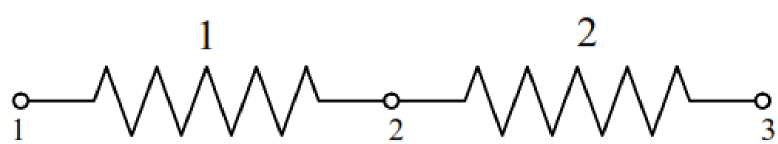

Figure 2. Finite element model proposed

The length of the elements it's *L/2*, where L = 0.30 m. The surface area is A = 15 m^2 and the thermal conductivity is k = 0.9 W/m·ºC.

The boundary conditions are going to be prescribed temperature on nodes 1 and 3, which are the outer surface temperature and inner surface temperature of the wall, as described in problem description.

Python code: Input data

1. Import the required libraries:

```
import matrix_assembly as m_a
import fem_solution as f_s
import boundary_conditions_1d_elements as bc
import numpy as np
```

2. Specify problem type and Finite Element Model properties:

```
problem_type = 1
Number_of_elements = 2
Number_of_nodes = 3
```

3. Specify coordinates of the model

```
coordinates = np.array([[0, 0], [0.15, 0], [0.30, 0]])
```

4. Specify Geometric properties and Material Properties of every element

```
geometric_properties = np.array([[15], [15]])
material_properties = np.array([[0.9], [0.9]])
```

5. Specify Boundary Conditions

```
bound_cond = np.array([[1, 1, 16, 0, 0], [1, 3, 2, 0, 0]])
```

6. Specify type of element and element nodes

```
elements = np.array([[1, 1, 2], [1, 2, 3]])
```

7. Call for assembly of matrix function, boundary conditions application and solution of the Finite Element problem (this lines must be placed for every problem).

```
KG,FG=m_a.stiffness_matrix_assembly(problem_type,coordinates,geometric_proper
ties,material_properties,elements,Number_of_elements,Number_of_nodes)
KG, FG = bc.thermal_boundary_conditions(bound_cond,elements,KG, FG)
Temps = f_s.steady_state_solution(KG, FG)
print(Temps)
```

The nodal temperatures results are:

$$\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix} = \begin{Bmatrix} 16 \\ 9 \\ 2 \end{Bmatrix} ^\circ\mathrm{C}$$

This file is attached as **problem1.py**

Problem #2. Composite wall:

A composite wall consists of three materials, as shown in figure 3. The outer temperature is T_w = 20 ºC. Convection heat transfer takes place on the inner surface of the wall with T_o = 800 ºC and h = 25 W/m^2·ºC. Determine the temperature distribution in the wall.

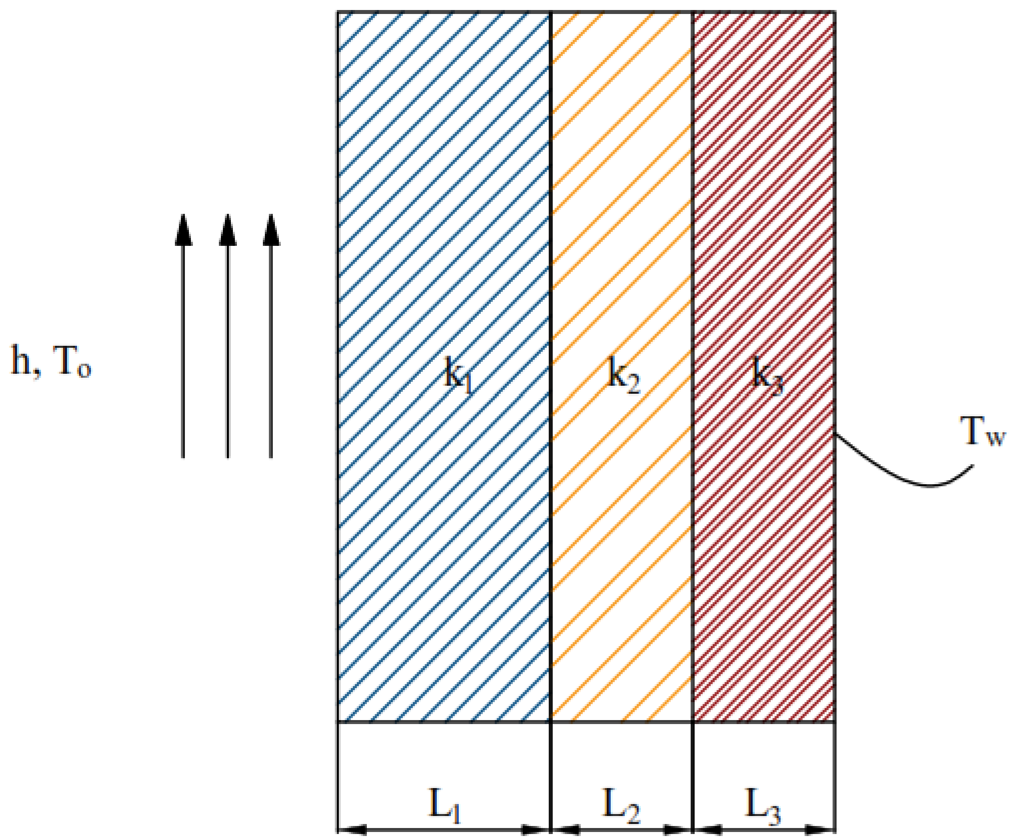

Figure 3. Composite wall with convection heat transfer on left side and prescribed temperature on right side

Table 1. Data for problem #2

k_1 (W/m·ºC)	k_2 (W/m·ºC)	k_3 (W/m·ºC)	L_1 (m)	L_2 (m)	L_3 (m)
20	30	50	0.30	0.15	0.15

Analytical Solution:

Results from [1] are:

$$\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} 304.6 \\ 119.0 \\ 57.1 \\ 20.0 \end{Bmatrix} \text{°C}$$

Finite Element Model:

Figure 4. Finite element model proposed

The Finite Element model is going to have 3 elements, each one of length L_1, L_2, L_3, respectively, with thermal conductivity of k_1, k_2, k_3, respectively. Since there's no given information about the area, we'll take as the unity ($A = 1\ m^2$).

The boundary conditions are going to be: Convection heat transfer on left side, and prescribed temperature on right side.

Python code: Input data

1. Import the required libraries:

```
import matrix_assembly as m_a
import fem_solution as f_s
import boundary_conditions_1d_elements as bc
import numpy as np
```

2. Specify problem type and Finite Element Model properties:

```
problem_type = 1
Number_of_elements = 3
Number_of_nodes = 4
```

3. Specify coordinates of the model

```
coordinates = np.array([[0, 0], [0.30, 0], [0.45, 0], [0.60, 0]])
```

4. Specify Geometric properties and Material Properties of every element

```
geometric_properties = np.array([[1], [1], [1]])
material_properties = np.array([[20], [30], [50]])
```

5. Specify Boundary Conditions

```
bound_cond = np.array([[3, 1, 25, 800, 1], [1, 4, 20, 0, 0]])
```

6. Specify type of element and element nodes

```
elements = np.array([[1, 1, 2], [1, 2, 3], [1, 3, 4]])
```

7. Call for assembly of matrix function, boundary conditions application and solution of the Finite Element problem (this lines must be placed for every problem).

```
KG,FG=m_a.stiffness_matrix_assembly(problem_type,coordinates,geometric_prope
rties,material_properties,elements,Number_of_elements,Number_of_nodes)
KG, FG = bc.thermal_boundary_conditions(bound_cond,elements,KG, FG)
Temps = f_s.steady_state_solution(KG, FG)
print(Temps)
```

The nodal temperatures results are:

$$\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} 304.76190477 \\ 119.04761905 \\ 57.14285715 \\ 20.00000001 \end{Bmatrix} \text{°C}$$

This file is attached as **problem2.py**