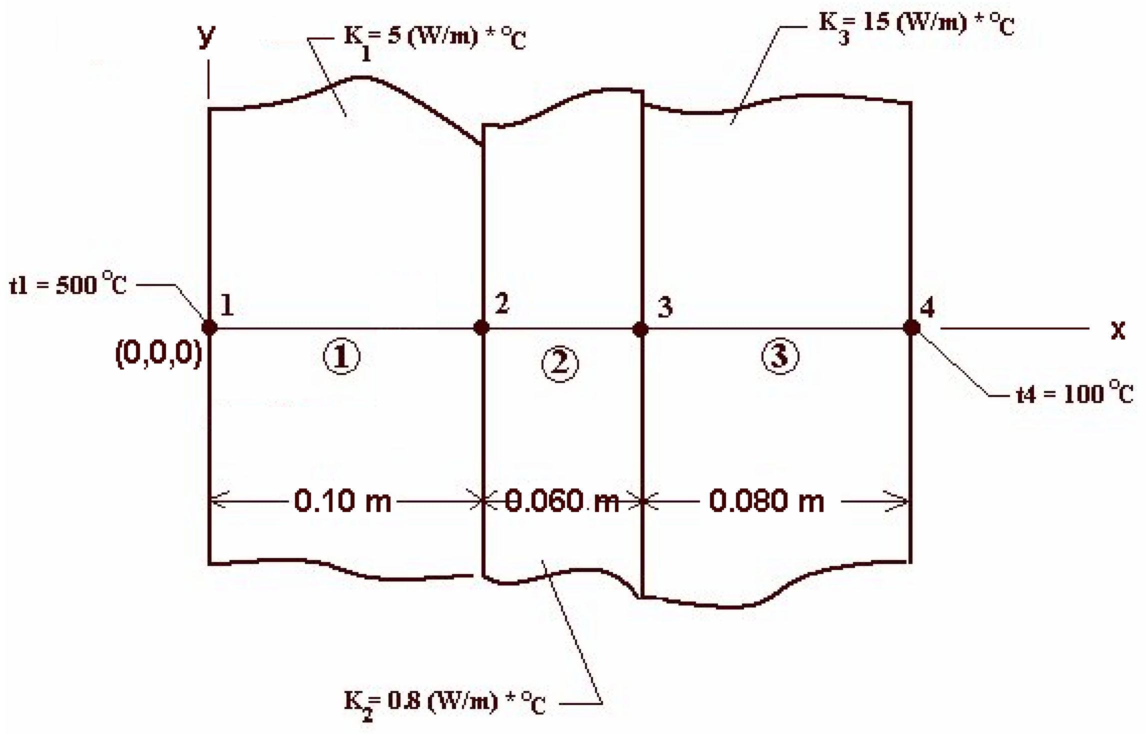
Solve the following heat transfer problem (steady-state conduction). Use Link32 (2-D Conduction Bar) elements for the structure.



<u>Step1: Set Preferences To Include Thermal Analysis</u>
Preferences→(window) select structural and thermal buttons→*OK*.

<u>Step 2: Define Nodal Locations</u>
Preprocessor→*Modeling*→*Create*→*Nodes*→*In Active CS*→(window) define node 1 with label and XYZ coordinates, hit *Apply* and repeat for nodes 2-4→when finished, select *OK* to exit window.

<u>Step 3: Select Element Type</u>
Preprocessor→*Element Type*→*Add/Edit/Delete*→(window) *Add…*→(window) highlight *Themal Mass-Link* and *2D Conduction 32,*→*OK*→*CLOSE*.

<u>Step 4: Define Material Properties</u>
Preprocessor→*Material Properties*→*Material Models*→(window) double click *Thermal/Conductivity/Isotropic*→(window) input value for thermal conductivity of material 1→*OK*→repeat for materials 2 and 3.

<u>Step 5: Define Real Constant</u>
Preprocessor→*RealConstants*→*Add/Edit/Delete* →(window) *Add…*→(window with element type 1 {Link 32} highlighted) *OK*→(window) input Cross-sectional area of 1*→*OK*→*CLOSE.*

*Note: since the material is approximated as infinite in the y-direction, we really have 1-d steady state conduction, which is the same for any arbitrary constant cross section==>use 1m^2 area for all elements.

Step 6: Build Elements Between the Nodes

Preprocessor→*Modeling*→*Create*→*Elements*→*User Numbered*→*Thru Nodes*→(window) assign element as No. 1, select *OK*→(selection window) pick nodes 1 and 2→*OK* (creates element 1). Now change the default material to mat2 and create element 2; then change to mat3 and create element 3. (Remember you can change material type using P*reprocessor*→*Modeling*→*Create*→*Elements*→ *Element Attributes*).

Step 7: Apply Boundary Conditions (Temperature Loads)

Preprocessor→*Loads*→*Define Loads*→*Apply*→*Thermal*→*Temperature*→*On Nodes*→(window) pick node 1 then select *OK*→(window) highlight *TEMP* only; make sure it shows *Apply As: Constant Value*; enter value as 500, select *OK*. Repeat for applying temperature at node 4.

Step 8: Solve

Solution→*Solve*→*Current LS*→(asks you to review summary info) select *OK*→ANSYS will begin solving the problem and will post a message "Solution is done!" when it has finished. Close message windows and go to next step.

Step 9: View Results

Temperatures are analogous to displacements in a structural analysis—they are the dof allowed at the nodes. Therefore, we can list the nodal temperatures similar to how we listed nodal displacements:

List Nodal Temperature: *General Postproc*→*List Results*→ *Nodal Solution*→(window) highlight *DOF solution* and *Temperature*; select *OK*. You should get the following:

NODE	TEMP (°C)
1	500.00
2	420.27
3	121.26
4	100.00

Heat flux is analogous to stress in a structural analysis. Recall for these 1-d type elements, we need to use the element table to access element output:

List Heat Flux: This needs to be done using the command input window. Type the following in the window:

```
etable,flux,smisc,4  (enter)
pretab,flux        (enter)
```

You should get the following:

Element	flux (W/m^2)
1	3986.7
2	3986.7
3	3986.7

Note that these could have been calculated easily from the nodal temperatures using the 1-d form of Fourier's Heat Conduction Law:

$$q_x = -K_{xx}\frac{dT}{dx} \quad \text{which is approximated as:} \quad q_x = -K_{xx}\frac{\Delta T}{\Delta x}$$

So, for example, the heat flux in element 2 could be calculated using know distances and temperatures as:

$$q_2 = -(0.8\frac{W}{m}{}^\circ C)\frac{(121.26 - 420.27)^\circ C}{(0.060m)} = 3986.8\frac{W}{m^2}$$

```
/com,  Structural                    !set preference to include structural and thermal analysis
/com,  Thermal
/prep7
n,1,0,0,0                            !node 1
n,2,0.1,0,0                          !node 2
n,3,0.16,0,0                         !node 3
n,4,0.24,0,0                         !node 4
et,1,32                              !element type = link32 (2D conduction)
mp,kxx,1,5                           !conductivity of material 1
mp,kxx,2,0.8                         !conductivity of material 2
mp,kxx,3,15                          !conductivity of material 3
R,1,1                                !cross-sectional area = 1
mat,1                                !use material 1
e,1,2                                !create element between node 1 and 2
mat,2                                !use material 2
e,2,3                                !create element between node 2 and 3
mat,3                                !use material 3
e,3,4                                !create element between node 3 and 4
d,1,temp,500                         !apply temperature of 500C to node 1
d,4,temp,100                         !apply temperature of 100C to node 4
fini

/solu
solve
fini

/post1
etable,flux,smisc,4                  !create result table containing heat flux
/output,heat,out                     !switch output to file heat.out
prnsol                               !list nodal solution
pretab,flux                          !list result of heat flux
/output,,                            !switch output back to screen
fini
```

Results from heat.out:

```
PRINT DOF  NODAL SOLUTION PER NODE

 ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

   NODE      TEMP
     1   500.00
     2   420.27
     3   121.26
     4   100.00

 MAXIMUM ABSOLUTE VALUES
 NODE         1
 VALUE    500.00

 PRINT ELEMENT TABLE ITEMS PER ELEMENT

 ***** POST1 ELEMENT TABLE LISTING *****

   STAT    CURRENT
   ELEM    FLUX
     1   3986.7
     2   3986.7
     3   3986.7

 MINIMUM VALUES
 ELEM         3
 VALUE    3986.7

 MAXIMUM VALUES
 ELEM         1
 VALUE    3986.7
```