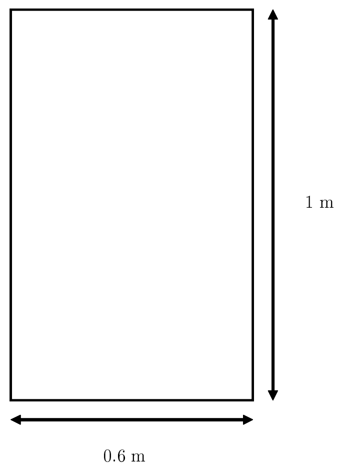
Chapter 7

Heat Transfer

7.1 Introduction

This example shows steady state thermal analysis for two dimensional model which covers heat transfer through conduction and convection to a prescribed external(ambient) temperature. This analysis will be carried out using Femlab. The benchmark result for the target location($x = 0.6$m and $y = 0.2$m) is a temperature of 18.25°C. Successive uniform refinements show a temperature of 18.26 and 18.25, converging toward the benchmark result.



Model Definition:

This model domain is 0.6 x 1 m. For the boundary conditions:

• The left boundary is insulated.

• The lower boundary is kept at 100°C.

• The upper and right boundaries are convecting to 0°C with a heat transfer coefficient of 750W/m^2C. • Ambient temperatre is 0°C.

Domain material properties:

• The density, is 7850 kg/m^3

• The heat capacity is 460 J/kg°C

• The thermal conductivity is 52 W/m°C

7.2 Modeling Procedures:

1)Model Navigator

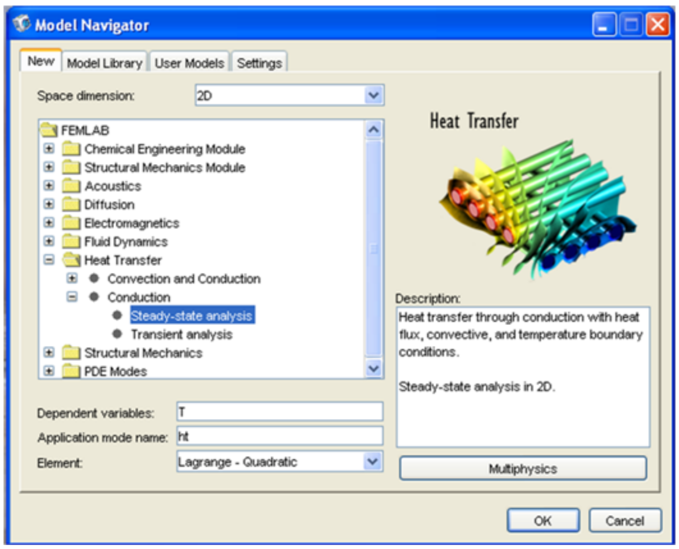
1. Open the Femlab software

2. In the Model Navigator dialog box, select 2D in the Space dimension list.

3. In the Application mode list, open the Heat Transfer folder and then the Conduction node.

4. Select Steady-state analysis.

5. Click OK.



2)Geometry Modeling:

1. On the Draw menu point to Specify Objects and click Rectangle.

2. In the Rectangle dialog box, find the Size area and enter 0.6 in the Width edit field, then enter 1 in the Height edit field.

3. Click OK.

4. Click the Zoom Extents button.

3)Physics Settings

3.1 Boundary Conditions

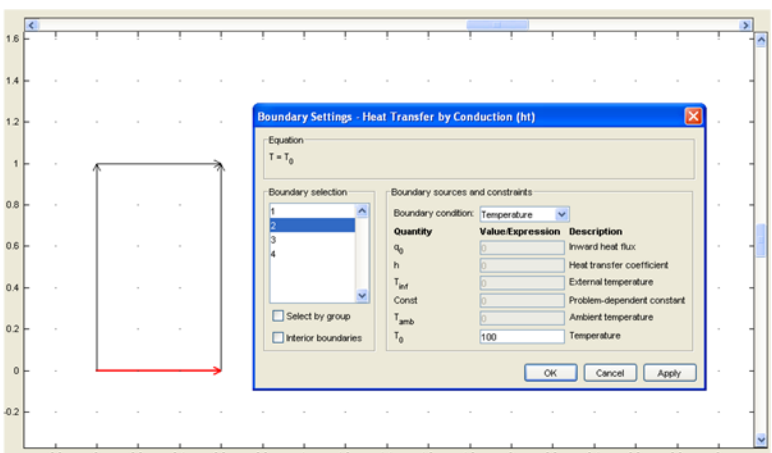
The default boundary condition is thermal insulation, so we must set boundary condition for only three of the boundaries.

1. Go to the Physics menu and choose Boundary Settings.

2. In the Boundary Settings dialog box select boundary 2.

3. In the Boundary condition list select Temperature.

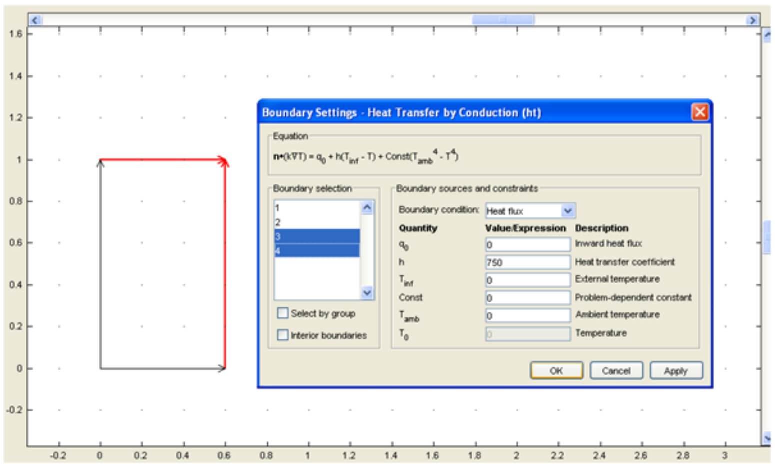
4. Enter 100 in the Temperature edit field.



5. Select boundaries 3 and 4.

6. In the Boundary condition list select Heat flux.

7. Enter 750 in the Heat transfer coefficient edit field.



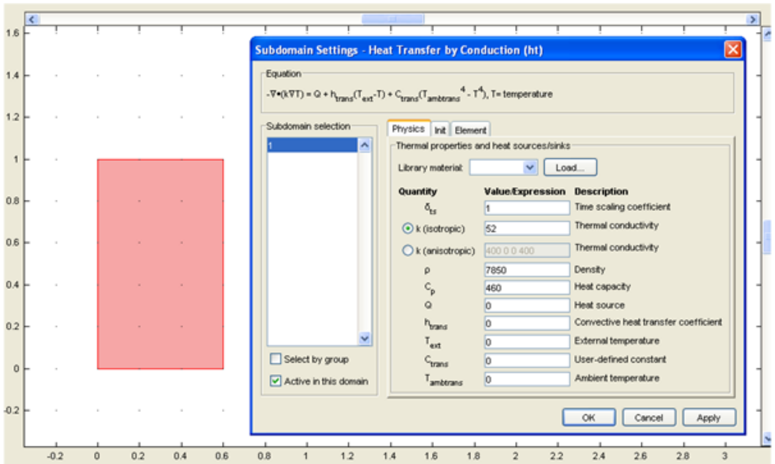
8. Click OK.

3.2 Subdomain Settings

1. Go to the Physics menu and choose Subdomain Settings.

2. In the Subdomain Settings dialog box enter the thermal properties in the domain according to the following table:

Subdomain	1
k(isotropic)	52
ρ	7850
C_p	460

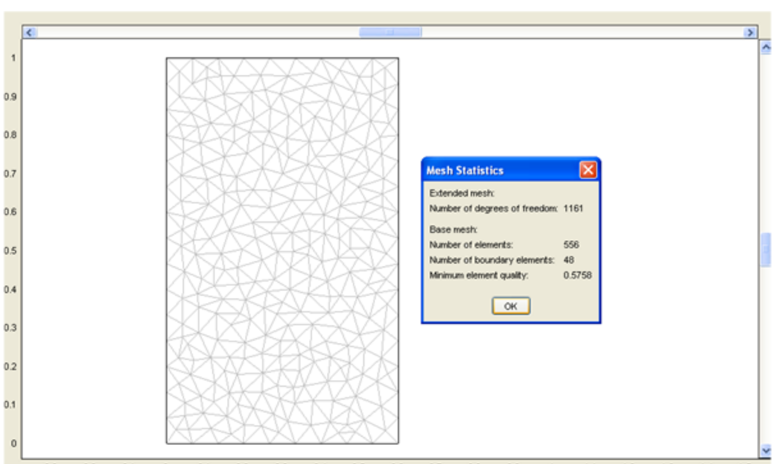


3. Click OK

4)Mesh Generation

1. Initialize the mesh by clicking the Initialize Mesh button on the top toolbar.

2. In the Mesh menu, choose Mesh Statistics. Note that the model had been divided into 556 elements.

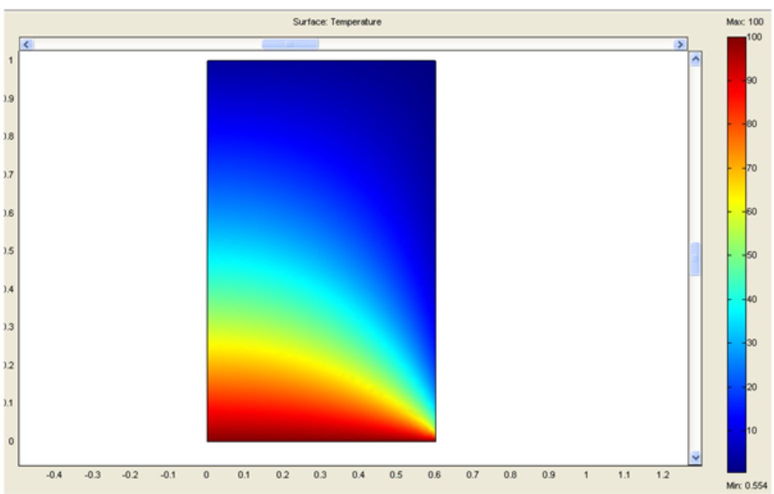


5)Solving The Model

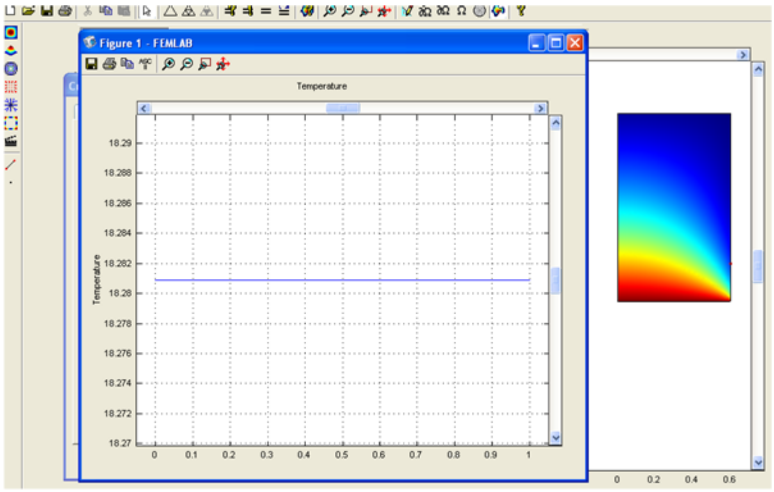
Click the Solve button.

6)Postprocessing and Visualization

The result is as shown below.



1. Go to the Postprocessing menu and choose Cross-Section Plot Parameters.
2. In the Croos-Section Plot Parameters dialog box click the Point tab.
3. In the Coordinates area enter 0.6 in the x edit field and 0.2 in the y edit field.
4. Click Apply.
5. The result is 18.2810^{o}C.



6. Now, click on the Refine Mesh button placed on the top toolbar.

7. Click Solve button.

8. Click on the Postprocessing menu and choose Cross-Section Plot Parameters.

9. In the Coordinates area enter 0.6 in the x edit field and 0.2 in the y edit field.

10. Click Apply.

11. The result is 18.257°C.

12. By refining the mesh once more, the result will give 18.254°C which is very close to the benchmark result.