Transient Conduction

Abstract—*In this exercise, transient one-dimensional heat conduction is modeled. The solid is represented in two dimensions. The length of the solid may be specified. Coarse, medium, and fine mesh types are available. The thermal conductivity of the solid can be specified within limits. The temperature at the left wall is specified along with the temperature at the right wall. Temperature gradient and heat flux are reported. In addition to contours of temperature, a plot of temperature distribution at different time steps is also available. The temperature contours may be animated to visualize the time dependent temperature field.*

1 Introduction

Thermal conduction is an important mode of heat transfer. Fourier's law of heat conduction relates the heat transfer rate to the temperature gradient, where thermal conductivity is represented by a constant of proportionality.

For unsteady heat conduction, the temperature may vary with respect to time and location within the material. If the thermal conduction mechanism is one-dimensional in nature and the domain is infinite, the analytical solution depends upon the similarity variable $x/\sqrt{t}$, where x represents position along the axis and t is time. Using this exercise, it is possible to verify that temperature profiles obtained at different time steps collapse into a single curve when represented in terms of the similarity variable, $x/\sqrt{t}$.

2 Modeling Details

The solid is represented in two dimensions by a rectangle. The procedure for solving the problem is:

1. Create the geometry.
2. Set the solid properties and boundary conditions.
3. Mesh the domain.

You must specify the following information:

- Time frequency at which the results and the plots should be saved.
- Number of time steps.
- Number of iterations per time step.

Because the domain is finite, the effect of boundary conditions on the solution will be experienced when the temperature front reaches the right boundary. *Penetration time* represents the approximate time required for the temperature front to reach the right boundary.

The following formula is used for the calculation of *Penetration time*.

$$ptime = \left[\frac{L}{1.65}\right]^2 \times \frac{\rho C_p}{4k} \tag{2-1}$$

where,

L = Length of the domain
ρ = Density of the solid material
C_p = Specific heat of the sold

FlowLab creates the geometry and mesh, and exports the mesh to FLUENT. The boundary conditions and flow properties are set through parameterized case files. FLUENT continues to solve the problem until the convergence limit is met or the number of iterations specified is achieved.

2.1 Geometry

The geometry consists of four walls (Figure 2.1).

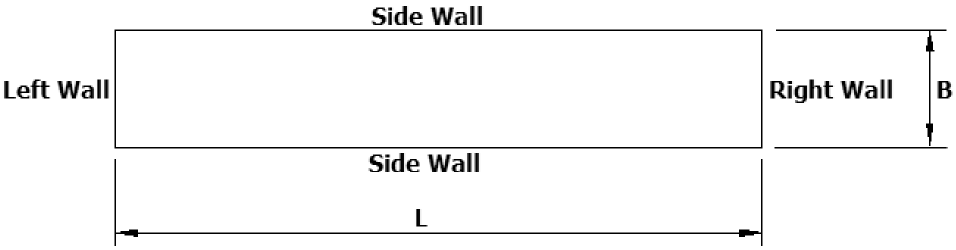

Figure 2.1: Schematic of the Solid with Boundaries

You have to specify the length L of the domain. The breadth B of the domain is calculated using the formula B = 0.1 × L.

The geometry is created from a set of four vertices. Edges are created over the vertices and are stitched with a face.

2.2 Mesh

Coarse, medium, and fine mesh types are available. The discretization scheme applied is based on the following logic:

- `$S1` = Number of cells along the length, L.
- `$S2` = Number of cells along the breadth, B.

The numerical values for the two discretization parameters are given in Table 2.1.

Mesh Type	$S1	$S2
Coarse Mesh	50	5
Medium Mesh	100	10
Fine Mesh	200	20

Table 2.1: Mesh Discretization Logic

The face is meshed using a map scheme after the edges are discretized into intervals (Figure 2.2).

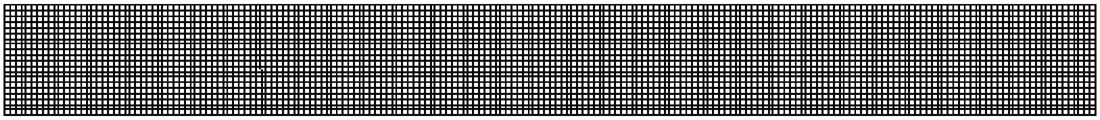

Figure 2.2: Mesh Generated by FlowLab

2.3 Physical Models for FLUENT

The energy equation is solved in FLUENT for the solid domain. Since there is no fluid flow, mass and momentum equations are not solved.

2.4 Material Properties

The default solid material is Aluminum. The following material properties can be specified

- Thermal conductivity
- Density
- Specific Heat

Other materials such as Copper, Steel, Wood and a User Defined solid can also be selected.

2.5 Boundary Conditions

You can specify the following boundary conditions:

- Left wall temperature
- Right wall temperature

The boundary conditions are assigned in FLUENT:

Boundary	**Assigned As**
Left Wall	Wall
Right Wall	Wall
Side Walls	Symmetry

Table 2.2: Boundary Conditions Assigned in FLUENT

2.6 Solution

The mesh is exported to FLUENT along with the physical properties and the initial conditions specified. The material properties and the initial conditions are read through the case file. Instructions for the solver are provided through a `journal` file. The `execute` commands in the `journal` file specify the frequencies at which the `results file` (`neutral file`) and `xy-plot` files are saved. When the solution is converged or the specified number of iterations is met, FLUENT exports the data to a `neutral` file and to `.xy` plot files. GAMBIT reads the `neutral` file for postprocessing.

3 Scope and Limitations

The ending (steady) state solution for the problem is a linear profile of temperature between the two walls. The penetration time is useful for setting up a realistic basis for comparing simulation results to theory. Because the mesh does not vary with time, large gradients at the beginning of the simulation may not be accurately captured.

Difficulty in obtaining convergence or poor accuracy may result if input values are used outside the upper and lower limits suggested in the problem overview.

4 Exercise Results

4.1 Reports

The following reports are available:

- Heat flux through the wall
- Average temperature gradient

4.2 XY Plots

The plots reported by FlowLab include:

- Residuals
- Temperature distribution at specific time steps

Figure 4.1 represents temperature profiles at different time steps.

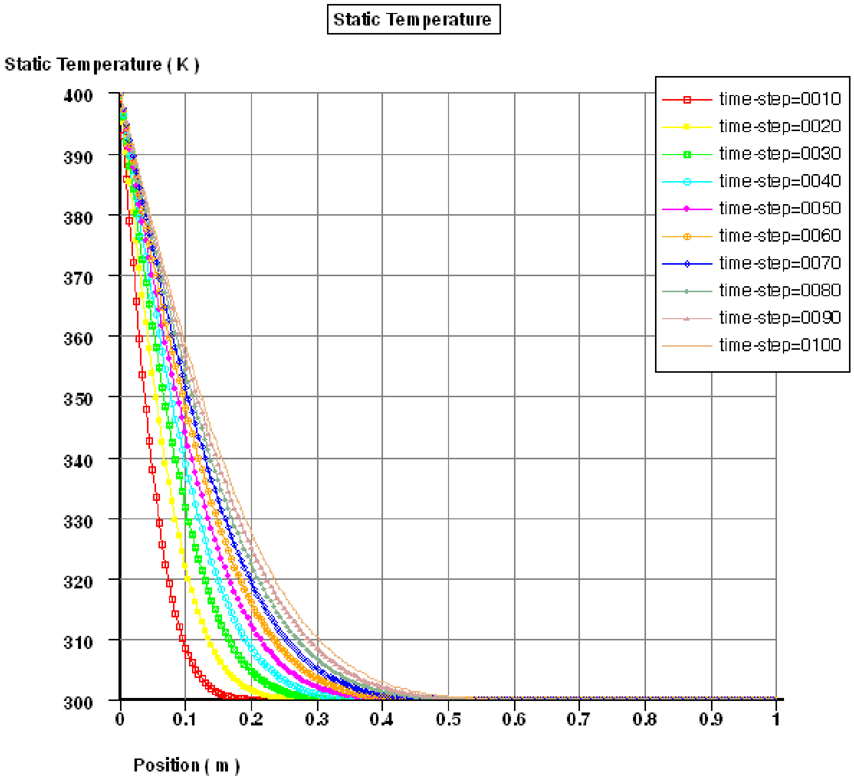

Figure 4.1: Temperature Distribution at Different Time Steps

4.3 Contour Plots

A contour plot of temperature at different time steps is shown in Figure 4.2.

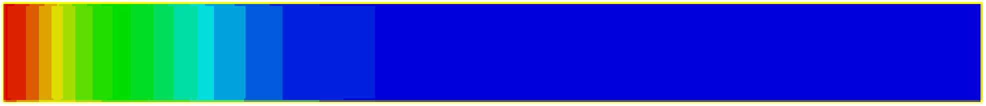

Figure 4.2: Contour Plot of Temperature at $t = 200$ s

5 Verification of Results

Heat flux at the left wall ($x = 0$) at any time step is given by:

$$q_x = \frac{k(T_L - T_R)}{(\pi\,\alpha\,t)^{0.5}} \tag{5-1}$$

where,

k = thermal conductivity of the solid
T_L = left wall temperature
T_R = right wall temperature
α = $\frac{k}{\rho\,C_p}$ = thermal diffusivity of the solid

For the default settings in the exercise, compare the predicted heat flux with the theoretical value at different time steps (See Table 5.1).

- Length = 1 m
- Thermal conductivity = 202.4 W/m-K
- Density = 2719 kg/m^3
- Specific heat = 871 J/kg-K
- Time step = 2 s
- Left wall temperature = 400 K
- Initial temperature = 300 K
- Iterations per time step = 50
- Fine mesh option
- Convergence criteria = 1.0e-9

Time (s)	**Heat Flux (FlowLab)**	**Heat Flux (theory)**
200	87,682.8	86,075
400	61,882.1	60,865
600	50,494.6	49,696
1800	29,218.3	28,692

Table 5.1: Heat Flux Verification

6 Sample Problems

1. Investigate the distribution of temperature at various time steps using default settings for the exercise.
2. Vary the material properties and investigate how these properties influence the heat flux at the left wall.

7 Reference

[1] *Sukhatme, S. P*, "Text Book on Heat Transfer", Universities Press, 1996.