Flow Over Heated Flat Plate Subjected to Uniform Temperature Condition

Shubhanshu Rai

M.Tech, Thermo-Fluids Engineering

Department of Mechanical Engineering

Indian Institute of Technology, Jodhpur, Rajasthan

Abstract

The aim of this work is determining the variation of temperature over the length of the heated plate having unheated starting and end length, each having dimension of 1m x 1m x 0.3m. The fluid is atmospheric air flowing at a speed of 1m/s, and the flow is laminar. Since it is conjugate, laminar, steady-state heat transfer problem, so "chtMultiRegionFoam" solver is to be used in OpenFoam v-7. Further validation of result is done, a similar analysis is done by Mohammad Najafi and Richard R. Scott [1], and obtained the same hydrodynamic and thermal boundary layer variation.

1. Introduction

Conjugate heat transfer analysis refers to the coupled analysis of heat transfer between fluids and solids. Heat transfer in fluids occurs through convection, which is a combination of two distinct mechanisms: diffusion and advection. In solids, only diffusion is, and it simply amounts to the vibration and collision of particles. Different PDEs used to represent the various physics in the fluids and the solids. These PDEs coupled at the fluid-solid interfaces, where the continuity of the heat flux and the temperature should be satisfied.

Conjugate heat transfer is relevant in most of the engineering applications, where an accurate prediction of the transfer of heat and the thermal loading is essential. It includes the design of industrial machines and devices, where a proper thermal design has a direct impact on the performance and the lifetime of the devices. Some example applications:

1) HVAC Application
2) Cooling of electronic devices
3) Heat exchangers
4) Gas turbines
5) Engine cooling

In this report, we initially experience the equations that define heat transfer in both the solid and liquid and some theoretical foundation about them. At that point, we study an open-source CFD program called OpenFOAM and what various solvers are accessible for conjugate heat transfer and how they vary from one another.

1.1 Conduction and Convective Heat Transfer

Heat conduction happens at a molecular level, where energy is transferred from more energetic particles to less energetic particles. The heat transfer rate between two bodies is proportional to the temperature difference between them. Conduction takes place at the molecular level. It is either due to the motion of electrons or due to lattice vibration. The latter contribute less to conduction. Fourier in 1822 gave the law that governs the conduction. It states that the heat transfer rate is proportional to the negative gradient of the temperature.

$$q = -kA_c \frac{dT}{dx} \tag{1.1}$$

This mode of heat transfer is a combination of Conduction & Advection. Convective heat transfer occurs in fluids only. It is the temperature difference and existence of gravity forces that create the difference in the density of fluids due to this difference in density; the buoyant force generated that drives the convection current. Further, it is classified into the following two types

a). Natural Convection
b). Forced Convection

Natural convection happens due to the presence of the gravitational field, which causes warmer, less dense fluid to rise, and colder, denser fluid to sink. Whereas, in the case of forced convection, the motion of the fluid is caused by an external source like fan, blower etc. Newton’s law of cooling defines the convective heat transfer. It states that the heat transfer rate is directly proportional to the temperature difference between the body and its surrounding.

Heat transfer rate is given as,

$$q = hA(T - T_{\infty}) \tag{1.2}$$

When both conduction in the solid and convection in the fluid are calculated together by obeying the first law of thermodynamics, then it is known as the conjugate heat transfer problem [2].

2. Problem Statement

The aim of this work is determining the variation of temperature over the length of the heated plate having unheated starting and end length, each having dimension of 1m x 1m x 0.3m. The fluid is atmospheric air flowing at a speed of 1m/s, and the flow is laminar. Since it is conjugate, laminar, steady-state heat transfer problem, so "*chtMultiRegionFoam*" solver is to be used in OpenFoam v-7.

2.1 Objectives of Study

In many practical applications such electronic component, processor etc. temperature/heat flux value is known, so burning or damage to component due to overheating is a common issue. The objective in this work is to determine the variation of temperature for plate over the entire length of it.

The aim of the work is as follows:

I. To study variation of temperature along the length.

II. In many electronics devise the heat is generated from cell/processor placed at a regular interval, so the adiabatic wall is placed at starting and end of the plate to simulate with similar conditions.

III. Validation of the obtained results.

3. Governing Equations

The CFD tool OpenFOAM is used for numerical analysis. The simulation is done by using the Finite Volume Method (FVM) for solving differential equations. The conservation equations of mass, momentum and energy are solved to find the values of velocity, density, temperature etc. The governing equations are given below.

Mass conservation:

$$\frac{\partial \rho}{\partial t} + \nabla . \rho u = 0 \tag{3.1}$$

For Steady flow:

$$\frac{\partial u}{\partial x} + \frac{dv}{dy} + \frac{\partial w}{\partial z} = 0 \tag{3.2}$$

Momentum Equation:

$$\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right)\rho = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \tag{3.3}$$

$$\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right)\rho = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \tag{3.4}$$

$$\left(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right)\rho = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \tag{3.5}$$

Energy Equation:

$$c_p\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) = \lambda\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \tag{3.6}$$

4. Simulation Procedure

4.1 Geometry and Mesh

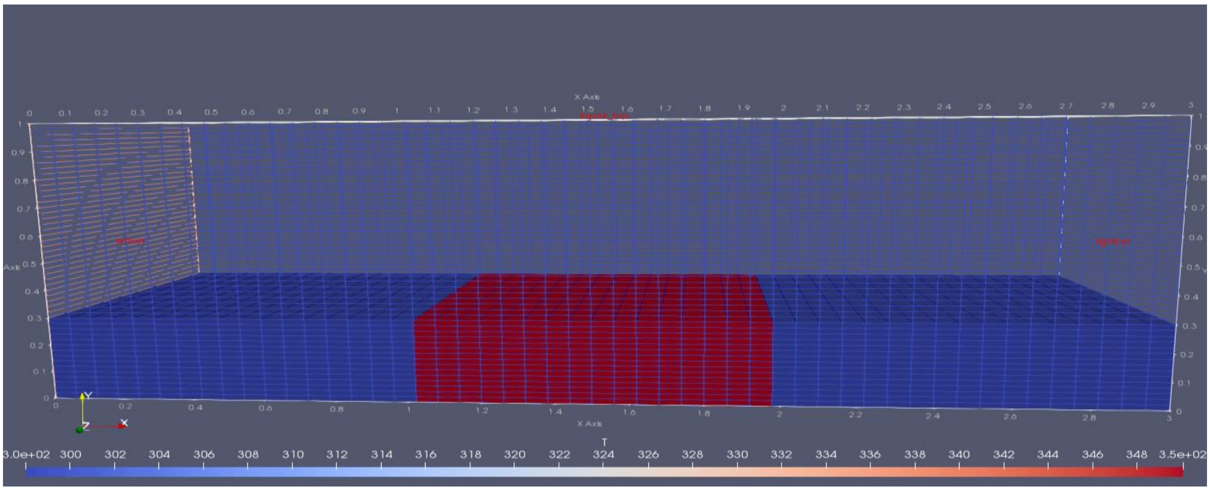

Fig. 4.1: Flat plate geometry/meshing.

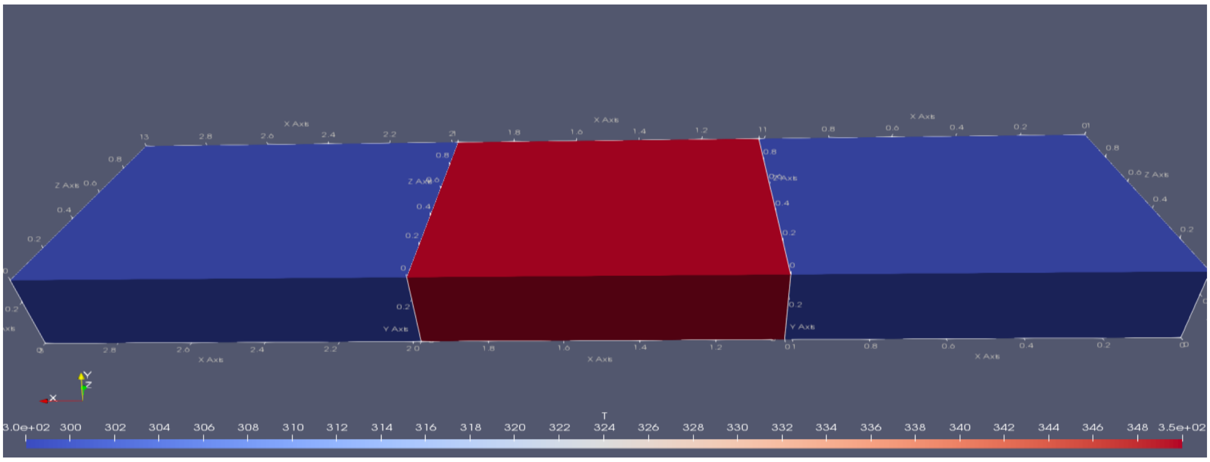

Fig. 4.2: Plate at t = 0 second.

Table 4.1: Description of Plate & Air

Rectangular shape	3m *3m *3m
Plate1(Adiabatic Plate)	1m * 1m * 0.3m
Plate2 (Constant Temperature Plate)	1m *1m * 0.3m
Plate3(Adiabatic Plate)	1m * 1m * 0.3m
Plate Material	Pure Tin
Fluid	Atmospheric Air

Model of the flat plate is made by using utility offered by OpenFOAM. Those are “blockMesh” and “topoSet” together, and meshing is done by “blockMesh” itself. The unstructured hexagonal shape is meshed using “blockMesh”. Figure 4.1 contains only 2500 mesh element. It is done purposely just to show the meshing clearly. However, during computation, it was kept in an account that meshing is very fine. Trade-off is made between mesh refinement and computation time; this analysis is done by using grid independence test.

4.2 Initial and Boundary Conditions

There are three flat pates in this problem different boundary conditions are applied for each of them. Plate1 and plate3 are adiabatic; their all sides are assumed as adiabatic and maintained at 298K. Plate2 is adiabatic from all side except the top side, where coupled boundary condition is applied. Initially, plate2 is maintained at 350K, and the fluid inlet is set to a uniform temperature of 298 K. Apart from these two boundaries and the conjugate boundary, the thermal boundary condition for the rest of the faces is adiabatic. The inlet velocity is set to a uniform value of u = 1m/s. Bottom of the fluid domain is set to slip before the leading edge of the plate, and no-slip after it. In order to simulate two-dimensional analysis by using the three-dimensional meshes, symmetry boundary conditions are used for the front and back faces.

4.3 Solver

"*chtMultiRegionFoam*" solver is used. SIMPLE algorithm is used to couple pressure and velocity equation. Residues taken for equation is of the order of 10^{-04}.

5. Results and Discussions

Temperature variation is plotted over the plate surface at different time. Fig. 5.5- 5.7 shows variation of plate 2 temperature with time. Since, the plate 1 and plate 3 are adiabatic. Therefore, their temperature will remain the same, while plate 2 temperature changes with time. Figure 5.8 shows variation of fluid(air) temperature along the length of plates at the plate surface i.e. within the boundary layer. At the first plate, air temperature is constant because there no temperature difference. As air flow over plate, it will get heated up which increases its temperature and further air temperature decreases as it flows over adiabatic plate 3.

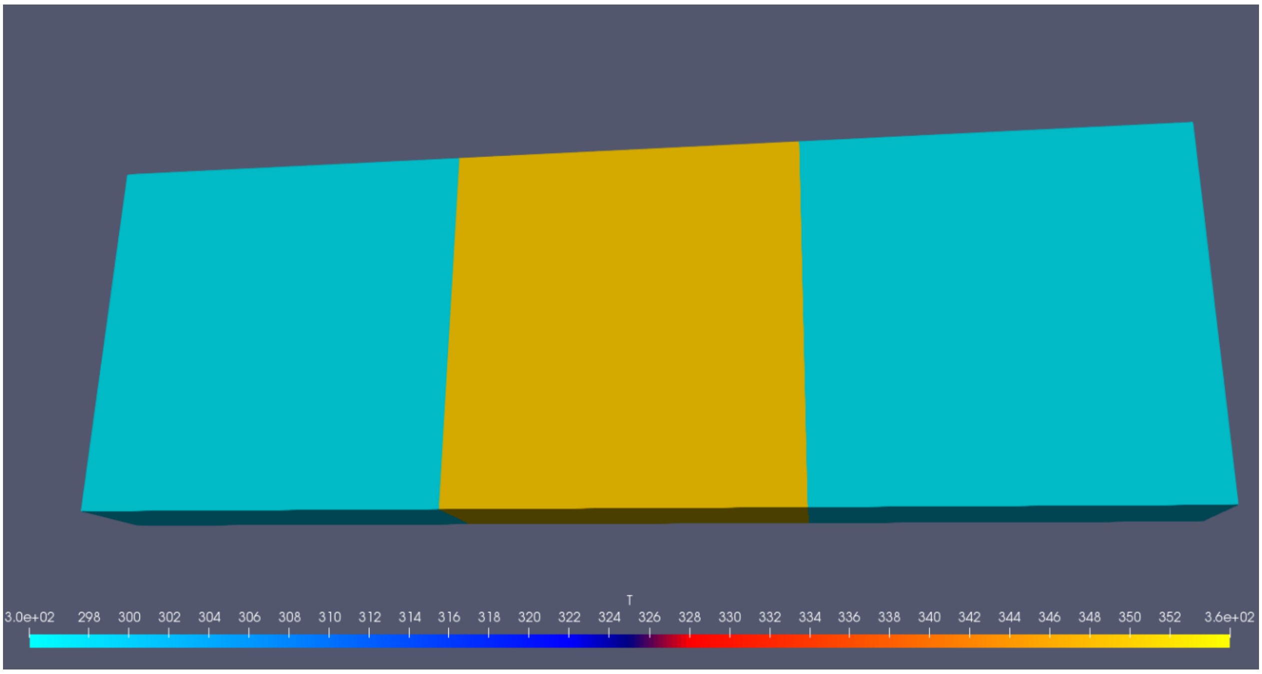

Fig 5.1: T at t = 0 second.

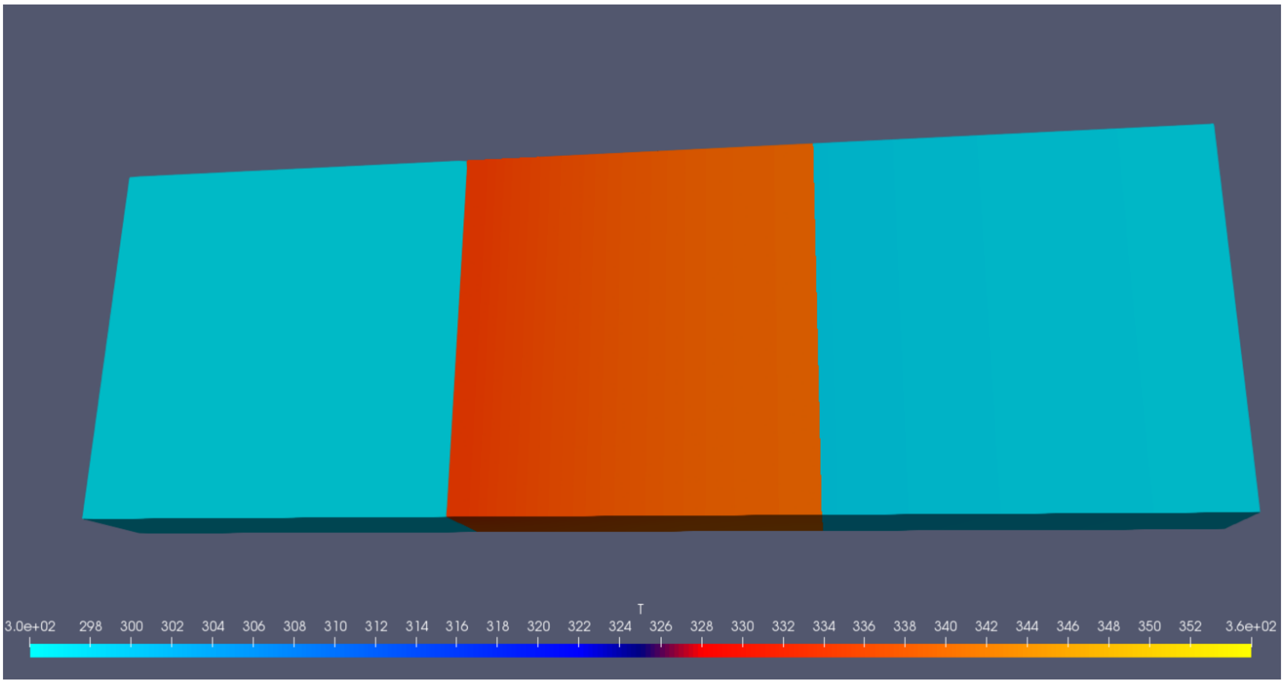

Fig 5.2: T at t = 50 second.

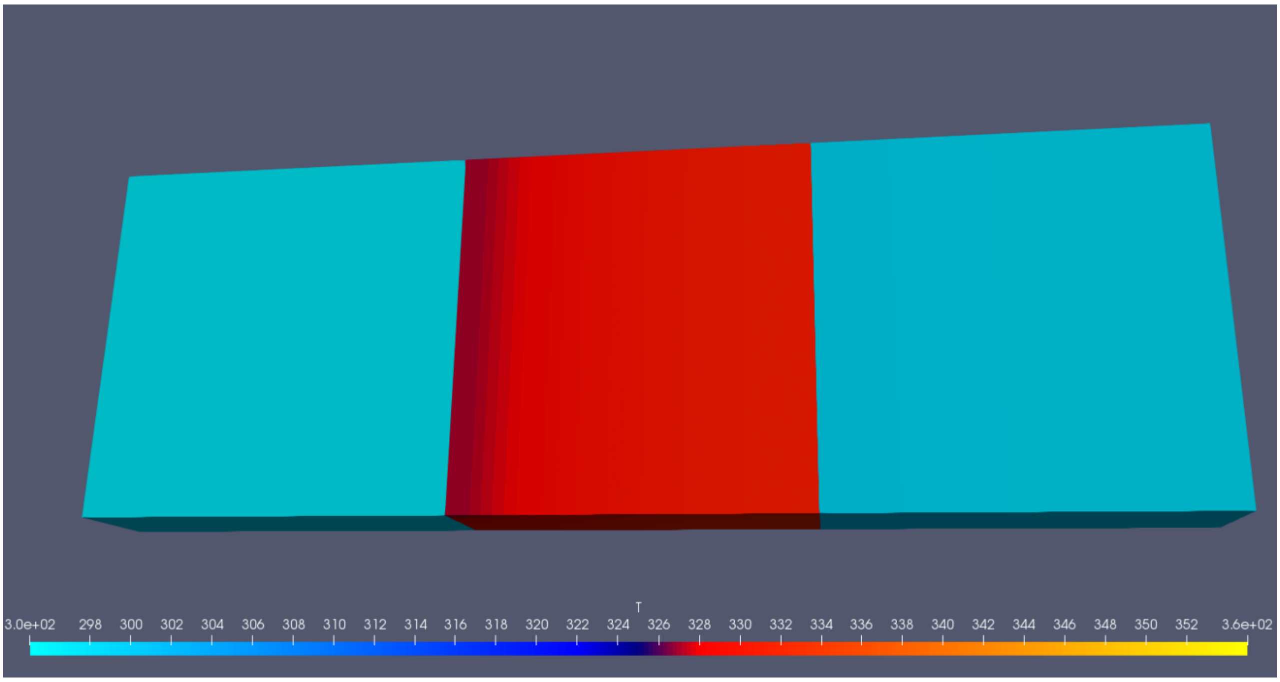

Fig 5.3: T at t = 100 second.

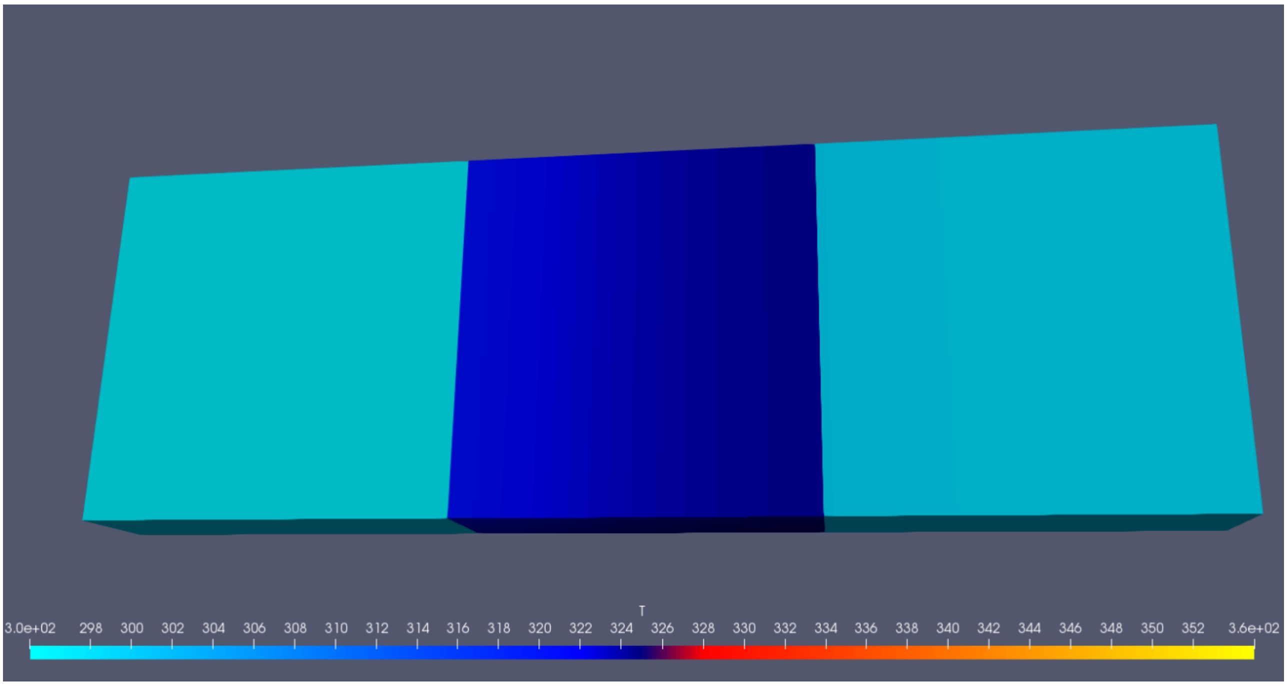

Fig 5.4: T at t = 150 second.

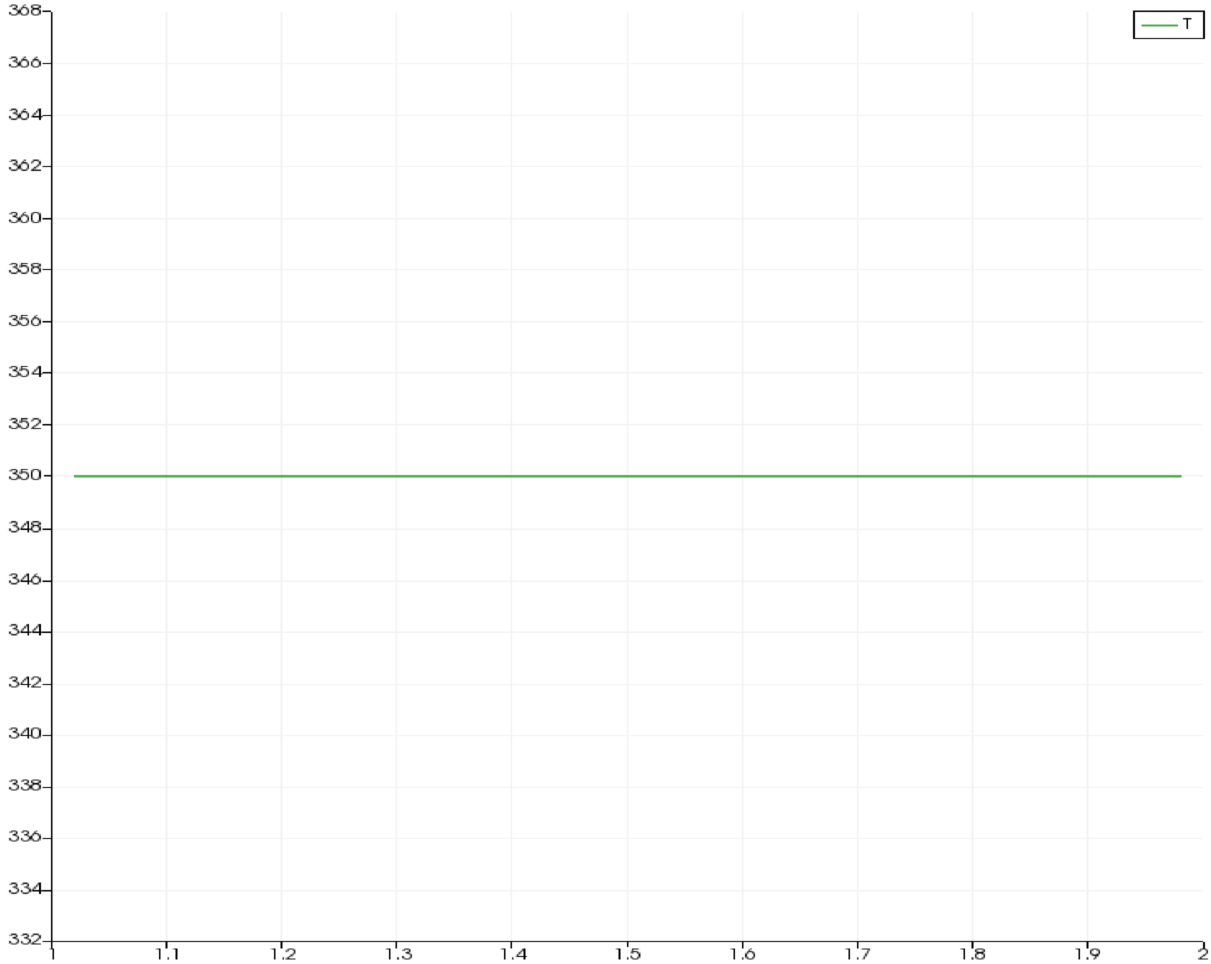

Fig 5.5: T at t = 0 second.

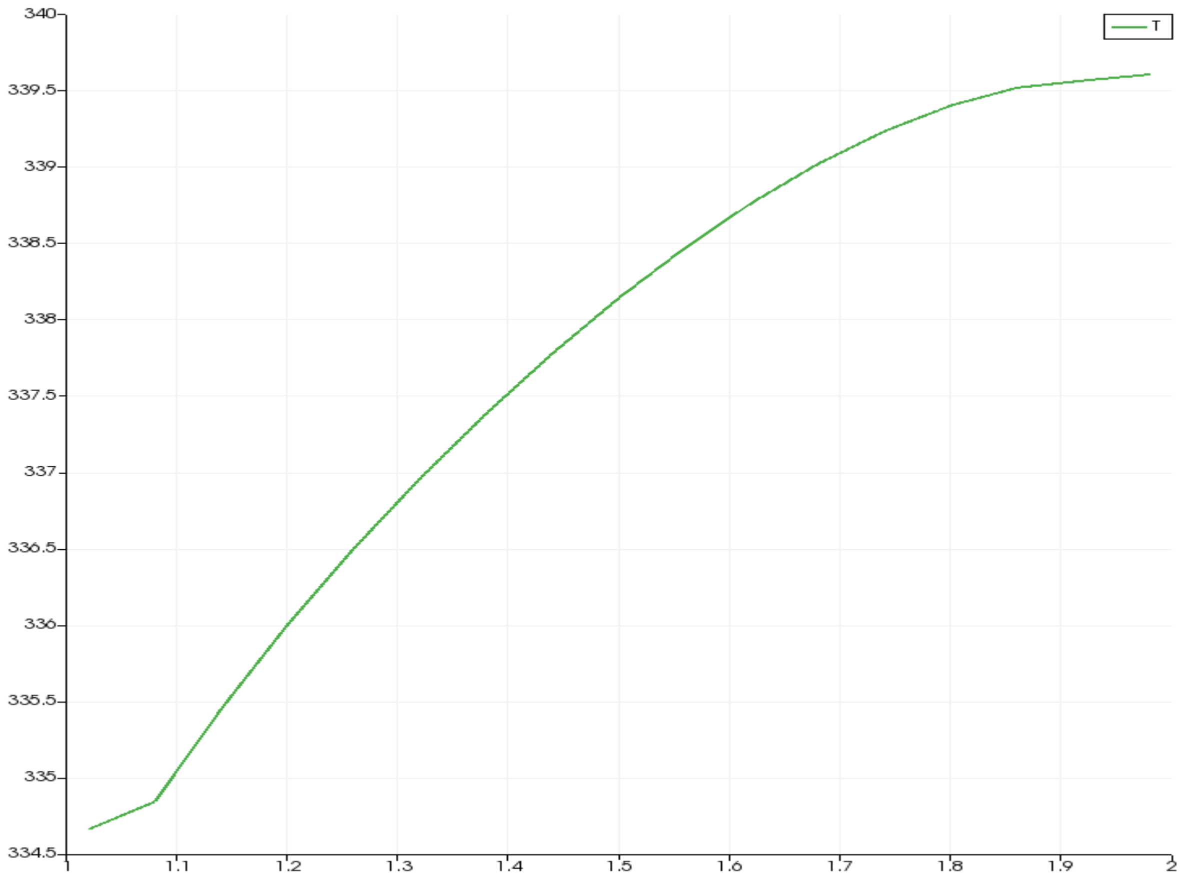

Fig 5.6: T at t = 50 second.

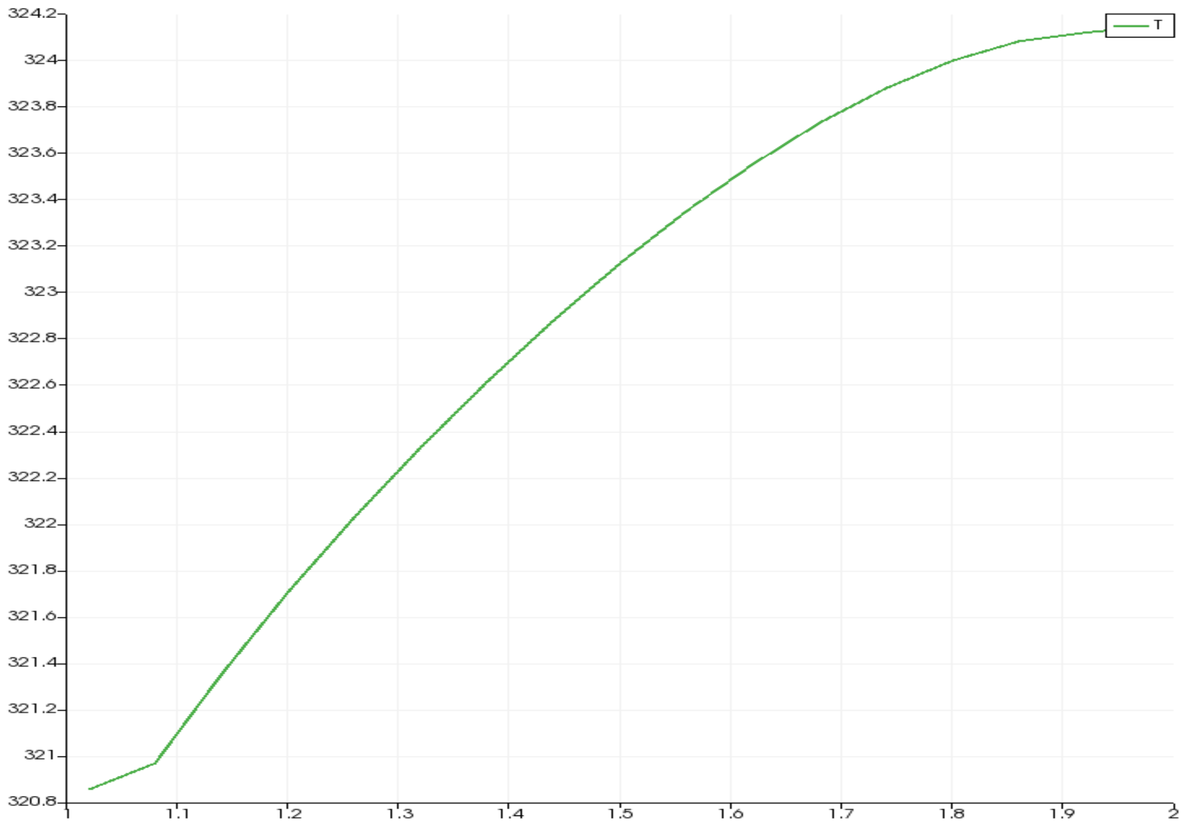

Fig 5.7: T at t = 150 second.

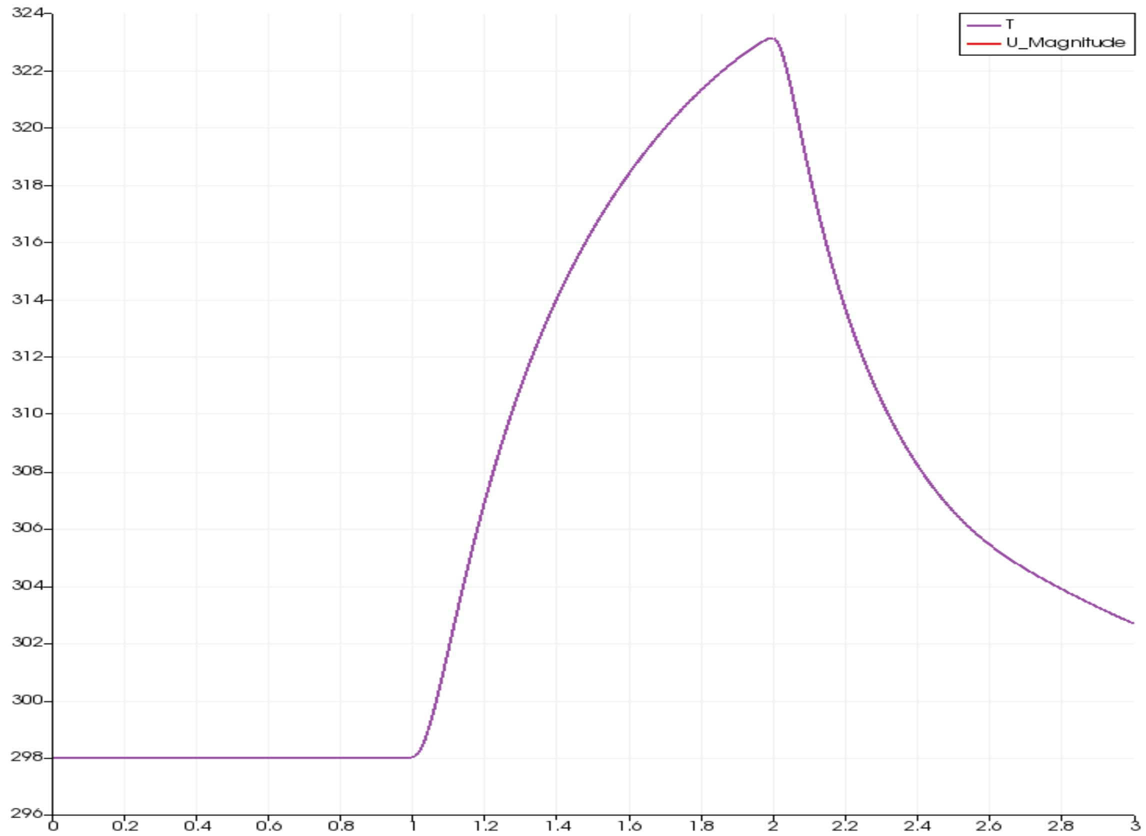

Fig 5.8: Temperature of air within the boundary layer.

5.1 Boundary Layers

As shown in Fig. 5.9 hydrodynamic boundary layer starts developing from the leading edge of the plate 1 itself. Whereas, thermal boundary layer start growing after leading edge of plate 2 because before plate 2 there is no temperature gradient that exists between air and plate as shown in Fig. 5.10.

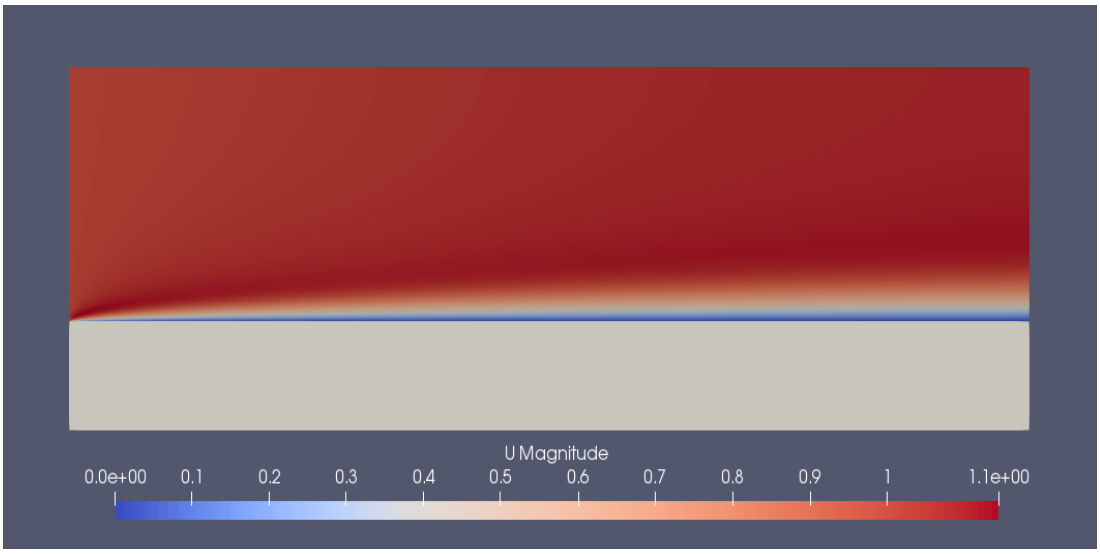

Fig 5.9: Hydrodynamic boundary layer development.

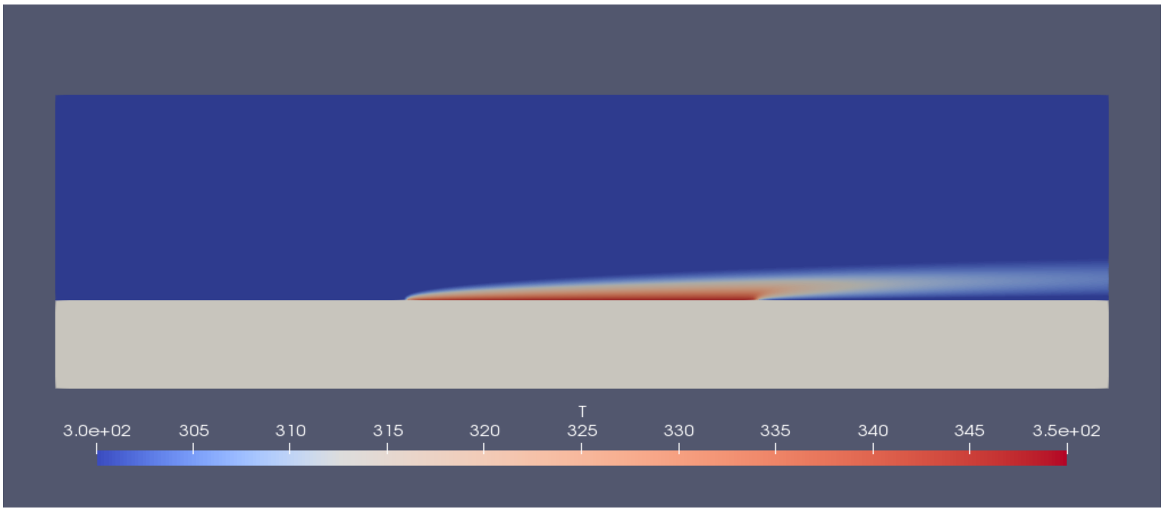

Fig 5.10: Thermal boundary layer development.

5.2 Validation

A similar analysis is done by Mohammad Najafi and Richard R. Scott[1], and they got the same Temperature, hydrodynamic and thermal boundary layer variation. Hydrodynamic boundary layer will start developing from the leading edge of the plate. In contrast, the thermal boundary layer starts growing from the heated plate as fluid is in thermal equilibrium with the starting plate, as shown in the above Fig. 5.10.

References

[1] Mohammad Najaf'i and Richard R. Scott, Average heat transfer coefficient for laminar & turbulent flow over partially heated flat plates, *IEEE*, 1996.

[2] R.L. Webb, Principles of Enhanced Heat Transfer, John Wiley & Sons, New York, 1994.