# 2D Battery Cooling using Natural and Forced Convection

**Jose Paul B**

College Graduate

Department of Mechanical Engineering

Government Engineering College, Thrissur

Abstract

The objective of the present project is to study the cooling of the battery model in open source CFD package openFOAM. Separate case studies with forced and natural convection are considered. The first case involves fluid flowing through the domain and in the second case fluid is stationary in the domain. There is unsteady heat transfer between the cover region and the fluid of the battery model. The Temperature versus time and average Nusselt number versus time are plotted as graphs.

## 1. Introduction

The development of batteries have brought changes in the transport system developments and alternative ways of storing energy. The actual working of batteries is quite complex and many complex models have been built. The long term cell performance is very much dependant on the operating Temperature and may degrade if it isn't maintained within the desirable range. So it is required to cool the battery using some mechanisms. In this two cases of battery are studied by considering it as a two concentrical cylinders, cooling by natural convection and forced convection. The inner region has heat generation for a short span of time and heat is conducted to outside cylinder which is cooled by fluid.

## 2. Problem Statement

Consider a 2D rectangular domain with 0.2 m X 0.2 m width and height. Inside the domain, at the centre a circular battery is placed which has a core region and a cover region. Both the core of a battery and cover are solid. Diameter of the core is 20 mm and the cover is 50 mm. Core of the battery is being heated during the operation with $10^7$ $W/m^3$ energy for 1 second and heat is transmitted to cover and then to domain. Solve for two different cases:

1) Fluid flowing with 1 m/s velocity and going out at 0 pa pressure
2) Fluid is stationary in the domain

| | Water (domain) | Aluminium(Battery cover) | Iron (Battery core) |
|---|---|---|---|
| Density | 1000 | 2700 | 8000 |
| Molecular weight | 18 | 27 | 50 |
| Kappa | ___________ | 200 | 80 |
| Cp | 4181 | 900 | 450 |
| Mu | 959e-6 | _____________ | ____________ |
| Pr | 6.62 | _____________ | ____________ |
| Initial Temperature | 300 | 300 | 300 |

Table 1: Values of properties

Simulate conjugate heat transfer problem using OpenFOAM and compare results of Temperature for,

1) Maximum Temperature at core in both the cases
2) Average Temperature at core in both the cases
3) Maximum Temperature at cover in both the cases
4) Average Temperature at cover in both the cases
5) Average Temperature at core and cover interface for both the cases

## 3. Governing Equations

The governing differential equations of fluid flow are as follows:

- Conservation of mass

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{U}\right) = 0 \qquad 3.1.a$$

- Conservation of momentum
  In X-direction

$$\frac{\partial \rho u}{\partial t} + \frac{\partial \rho u u}{\partial x} + \frac{\partial \rho u v}{\partial y} + \frac{\partial \rho u w}{\partial z} = -\frac{\partial p}{\partial x} + \frac{\mu \partial}{\partial x}\left(\frac{\partial u}{\partial x} + \frac{\partial u}{\partial y} + \frac{\partial u}{\partial z}\right) + \rho a_x \qquad 3.1.b$$

In Y-direction

$$\frac{\partial \rho v}{\partial t} + \frac{\partial \rho v u}{\partial x} + \frac{\partial \rho v v}{\partial y} + \frac{\partial \rho v w}{\partial z} = -\frac{\partial p}{\partial y} + \frac{\mu \partial}{\partial y}\left(\frac{\partial v}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial v}{\partial z}\right) + \rho a_y \qquad 3.1.c$$

In Z-direction

$$\frac{\partial \rho w}{\partial t} + \frac{\partial \rho w u}{\partial x} + \frac{\partial \rho w v}{\partial y} + \frac{\partial \rho w w}{\partial z} = -\frac{\partial p}{\partial z} + \frac{\mu \partial}{\partial z}\left(\frac{\partial w}{\partial x} + \frac{\partial w}{\partial y} + \frac{\partial w}{\partial z}\right) + \rho a_z \qquad 3.1.d$$

Where ρ is density, t is time ,$\vec{U}$ represents velocity vector , u is x component of velocity, v is y component of velocity ,w is z component of velocity , p is pressure and $a_i$ is component of acceleration in $i^{th}$ direction.

- Conservation of energy

$$\rho Cp \frac{dT}{dt} + \rho Cp \vec{U}.\nabla T = \nabla(K \nabla T) + G \qquad 3.1\ e$$

Where ρ is density, t is time, $\overrightarrow{U}$ represents velocity vector, Cp is specific heat, T is Temperature, K is thermal conductivity and G is heat generation.

- Coupling condition at boundary

$$Q(\text{flux}(\text{Boundary i})) = -Q(\text{flux}(\text{Boundary j})) \qquad 3.1\ f$$

Where $Q_{flux}$ is heat flux at boundary of nearby regions i or j; i , j may be solid or fluid

# 4. Simulation Procedure

## 4.1 Geometry and Mesh

The geometry was created in openFOAM itself. The grid generation was done using blockMesh tool. It was made using hexahedral blocks with simplegrading 1 in all direction and number of

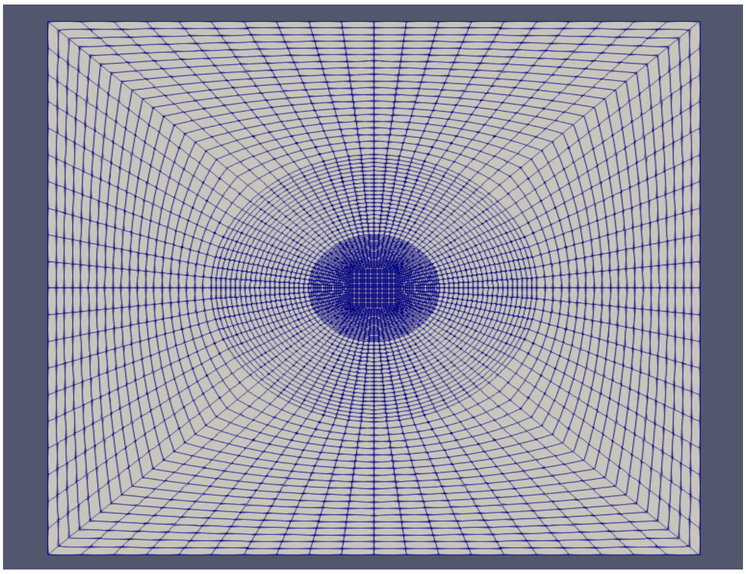

Figure 1: Geometry after grid generation

blocks in x, y and z direction as 20, 20 and 1.The z direction needed only one block as it is not used for calculation in 2D case. The mesh was split into regions using toposet and splitmeshregion command.

## 4.3 Initial and Boundary Conditions

The initial and boundary conditions for fluid are included in the 0 folder as fluid as p, p_rgh, T and U. The internal fields were kept as uniform 0 for pressure and hydrostatic pressure and uniform 300 K for Temperature field. There are two cases for velocity.

Case 1: Velocity is 1m/s in x direction at inlet with no slip at solid boundaries.

Case 2: Fluid is stationary

| | Pressure | Hydrostatic pressure | Temperature |
|---|---|---|---|
| inlet | calculated | calculated | fixedValue Uniform 300 |
| outlet | fixedValue Uniform 0 | fixedValue Uniform 0 | inletoutlet |
| fixedWalls | calculated | calculated | zeroGradient |
| frontandback | empty | empty | empty |
| fluid_to_cover | calculated | calculated | compressible:: turbulentTemperatureCoupledBaffleMixed |

Table 2: Boundary conditions

## 4.4 Constant Properties of the system

The constant properties of the system defines various characteristics defining mesh information, transport properties and turbulence properties in each of their own folder in constant folder as follows:

- **polyMesh**

The polyMesh folder contains the mesh information which is created using blockMesh command.

- **regionProperties**

This file contains the information about the regions. The fluid region is defined under fluid and cover and core under solid region.

• **turbulenceProperties**

This file contains the information about the turbulence properties of the system. The simulation type was kept as laminar.

• **thermophysicalProperties**

The thermo physical properties are kept as below in individual folders:

| | Fluid | Cover | Core |
|---|---|---|---|
| Density | 1000 | 2700 | 8000 |
| Molecular weight | 18 | 27 | 50 |
| Kappa | ________ | 200 | 80 |
| Cp | 4181 | 900 | 450 |
| Mu | 959e-6 | ________ | ________ |
| Pr | 6.62 | ________ | ________ |

Table 3: Thermo physical properties of region

• **g**

The gravity is set as 0 in all direction in the dictionary g.

• **fvOptions**

The source is set as semi implicit source in core region. The value is set as 1e7 for a duration of 1s from starting time.

## 4.5 Setting the runtime conditions and output control

The simulation control files, solution schemes, solution controls and graph plotting files are defined in the system folder.

• **controlDict**

The number of iterations were kept as 10 with time step write control and deltaT was set as 1e-3. Functions to find maximum core Temperature, maximum cover Temperature, average interface Temperature, average core Temperature and average cover Temperature were included. Wall heat flux was also defined in the controlDict to find the heat flux at cover boundary.

• **fvSchemes**

This file contains the information about the discretization and interpolation schemes for various mathematical expressions such as gradient, Laplacian terms etc. along with temporal discretizetion.

• **fvSolution**

This solution contains the information about the solvers to be used, tolerances and solution algorithms.

• **cellMaxcoreT**

This file is to find the maximum Temperature in the core region.

• **cellMaxcoverT**

This file is to find the maximum Temperature in the cover region.

• **VolAvgcoreT**

This file is to find the average Temperature in the core region.

• **VolAvgcoverT**

This file is to find the average Temperature in the cover region.

• **VolAvgInterface**

This file is to find the average Temperature in the core and cover regions.

### 4.6 Solver

The simulation is meant to be for heat transfer problem between multiple regions. The solver used in this case is chtMultiregionFoam. It is a solver for steady and transient fluid flow and solid heat conduction, with conjugate heat transfer between regions.

## 5. Results and Discussions

The solutions were obtained for 10 seconds .Two cases were done for this problem. The plots for temperature were made for both the cases using gnuplot. The average Nusselt number was calculated using calculator and integrate variable filter from wall heat flux of cover fluid boundary. The values were exported into files which was used to plot graphs.

## 5.1 fluid flowing with 1 m/s velocity and going out at o pressure

The table 4 shows the Temperature at various region. Figure 2 to figure 4 shows the contours of the Temperature. Figure 5 shows the streamline contour for the forced convection.

| | At 1 second | At 10 seconds |
|---|---|---|
| Maximum core Temperature | 302.7716 K | 300.8755 K |
| Average core Temperature | 302.2177 K | 300.7424 K |
| Average interface Temperature | 301.0389 K | 300.6337 K |
| Maximum cover Temperature | 300.9257 K | 300.6286 K |
| Average cover Temperature | 300.1561 K | 300.5506 K |

Table 4: Temperature at the end of 1 second and 10 seconds for forced convection

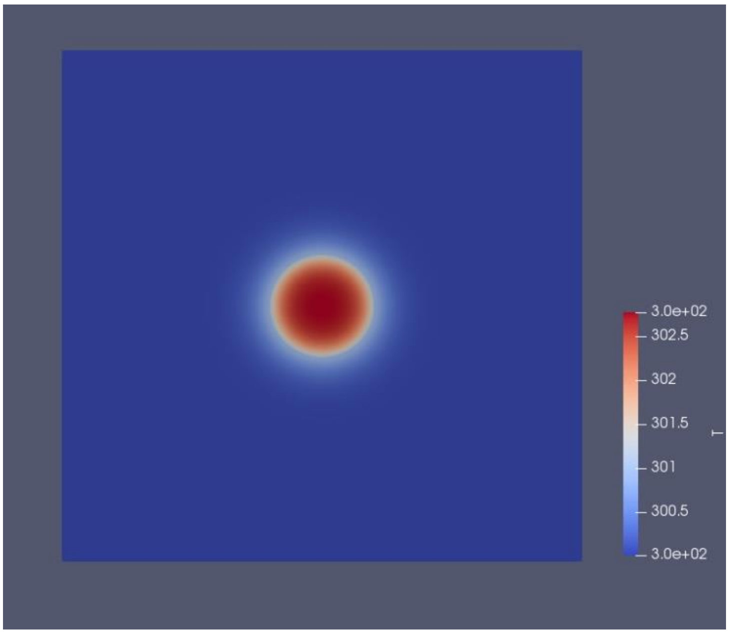


Figure 2: Temperature contour at 1s

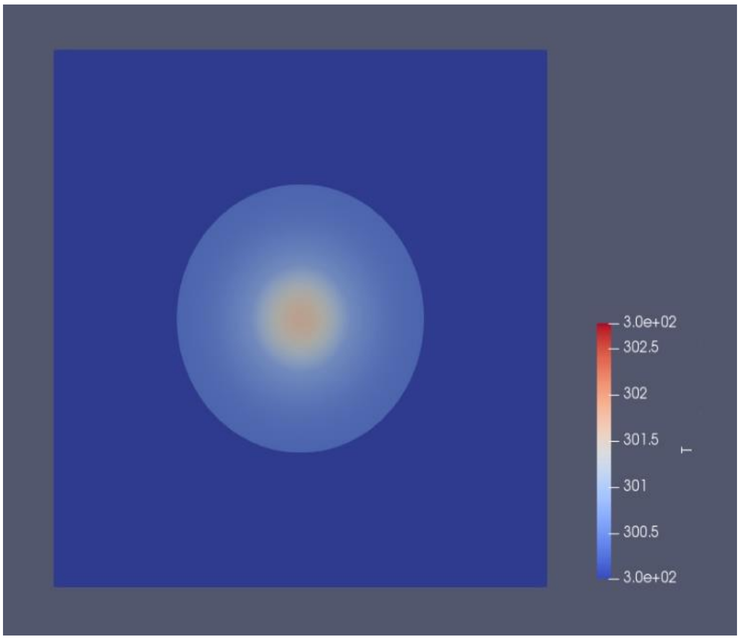


Figure 3: Temperature contour at 5 s

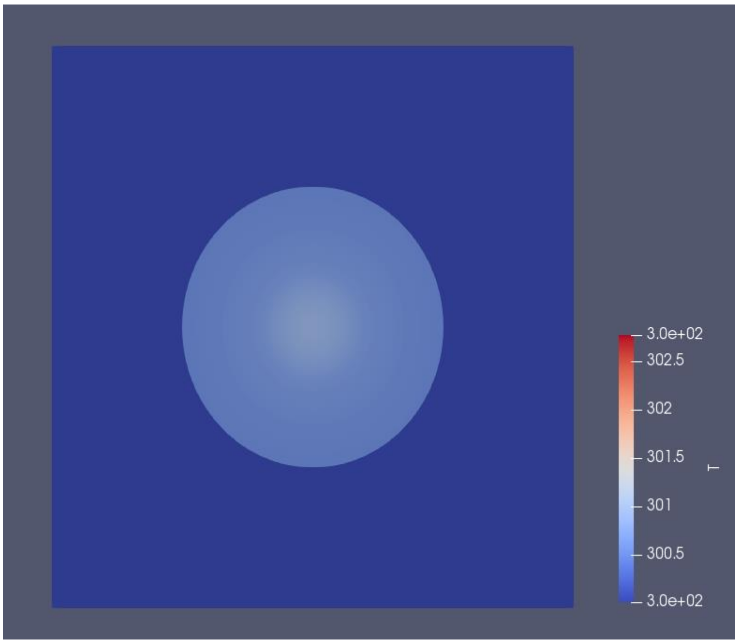


Figure 4: Temperature contour at 10 s

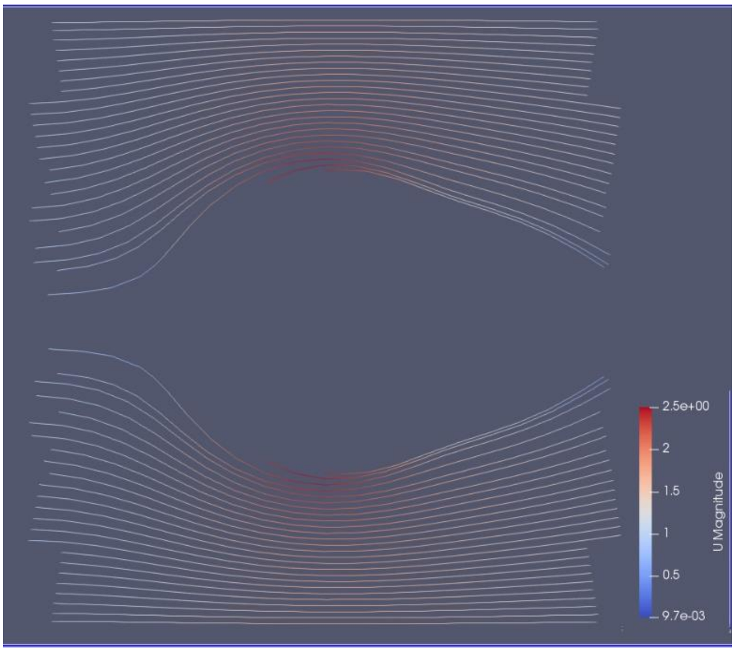


Figure 5: Streamline contour

Reynolds number was found to be 52137.64338. The parameter $Nu/Re^{0.5}$ was plotted against angle from front point of stagnation till the rear point of stagnation. The plots from 2 seconds to 10 seconds are overlapped. Average Nusselt number was also plotted against time with an interval of 1 second.

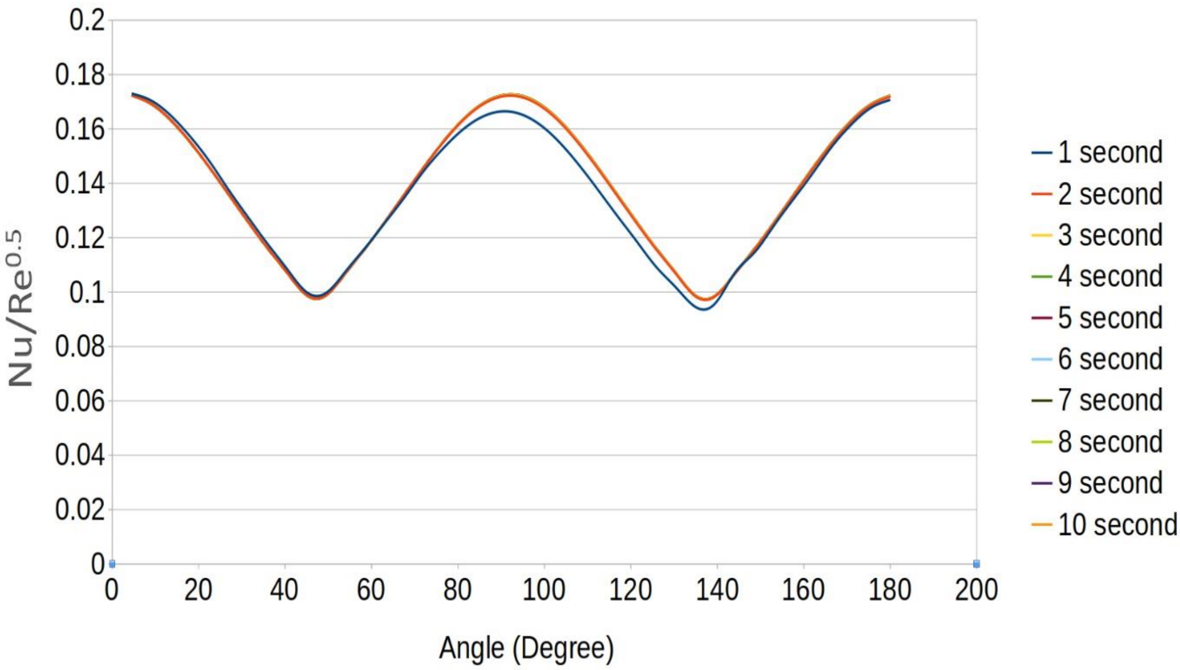


Figure 6: Variation of the parameter $Nu/Re^{0.5}$ against angle

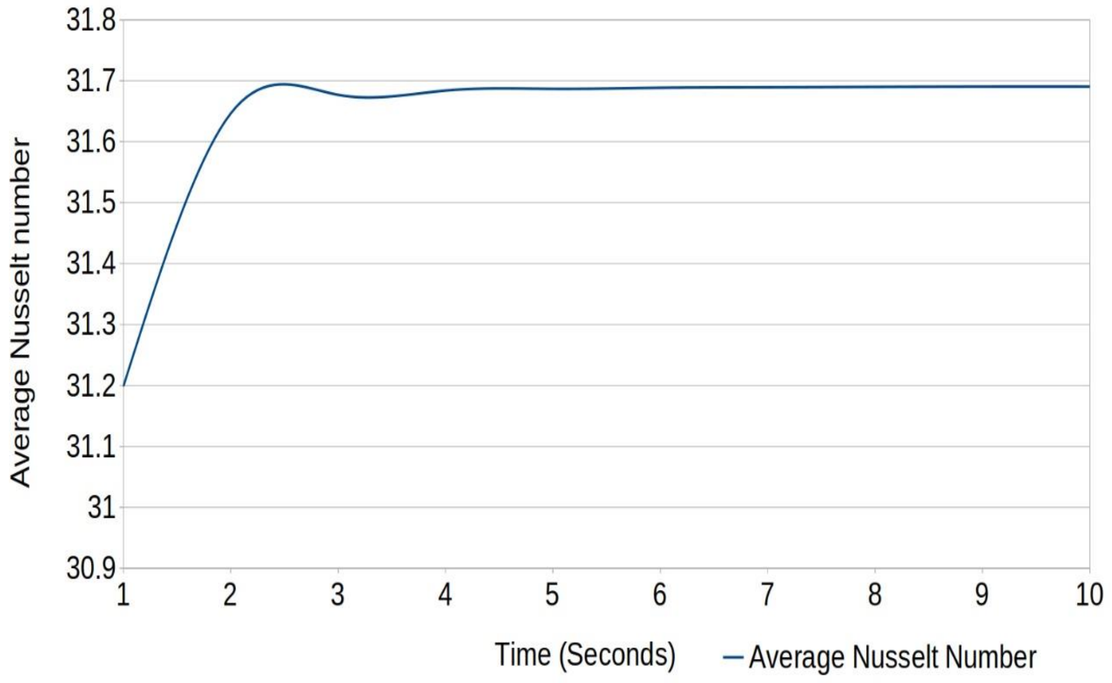


Figure 7: Variation of average Nusselt number with time for forced convection

## 5.2 Fluid is stationary in the domain

The table 5 shows the Temperature at various region. The figure 8 to 10 show the contours of Temperature versus time. Figure 11 to figure 15 shows the plots of the Temperature against time.

| | At 1 second | At 10 seconds |
|---|---|---|
| Maximum core Temperature | 302.7687 K | 300.8651 K |
| Average core Temperature | 302.1809 K | 300.7329 K |
| Average interface Temperature | 300.9382 K | 300.6254 K |
| Maximum cover Temperature | 300.8359 K | 300.6202 K |
| Average cover Temperature | 300.1398 K | 300.5415 K |

Table 5: Temperature at the end of 1 second and 10 seconds for forced convection

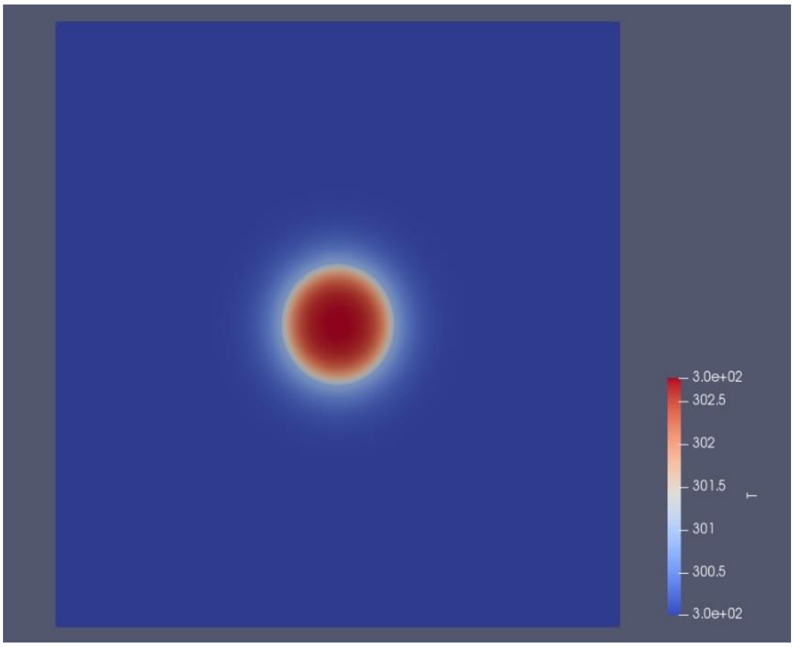


Figure 8: Temperature contour at 1s

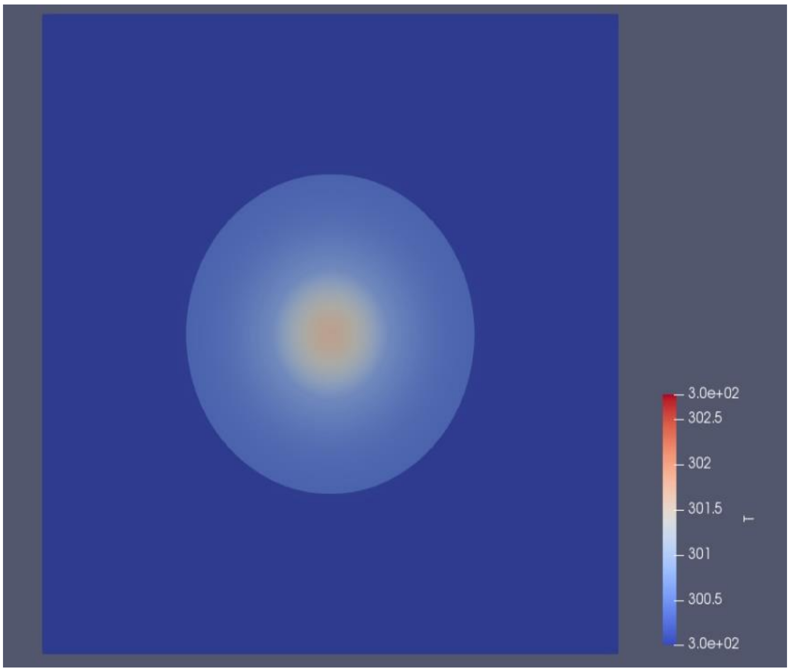


Figure 9: Temperature contour at 5 s

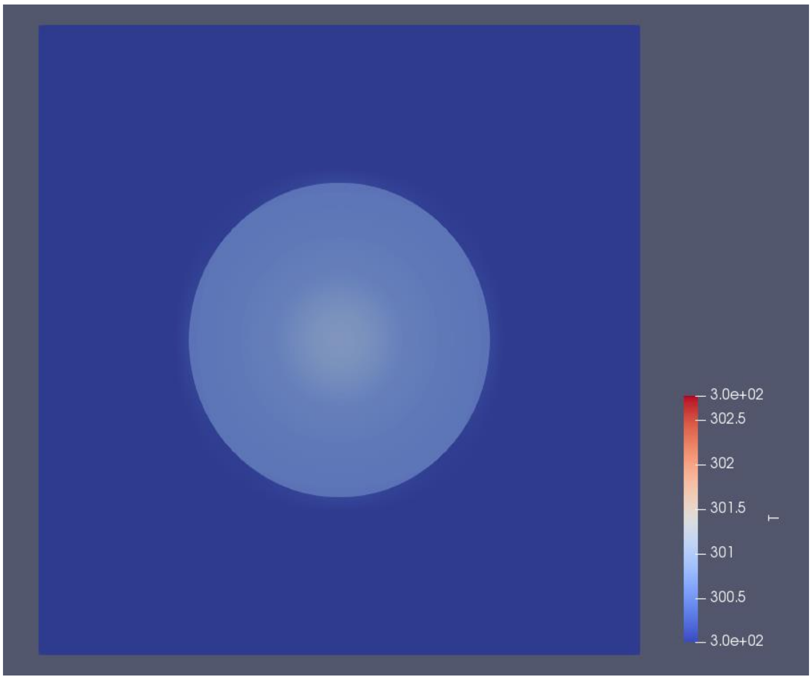


Figure 10: Temperature contour at 10 s

The following shows the comparison of different Temperature plots versus time for the two cases.

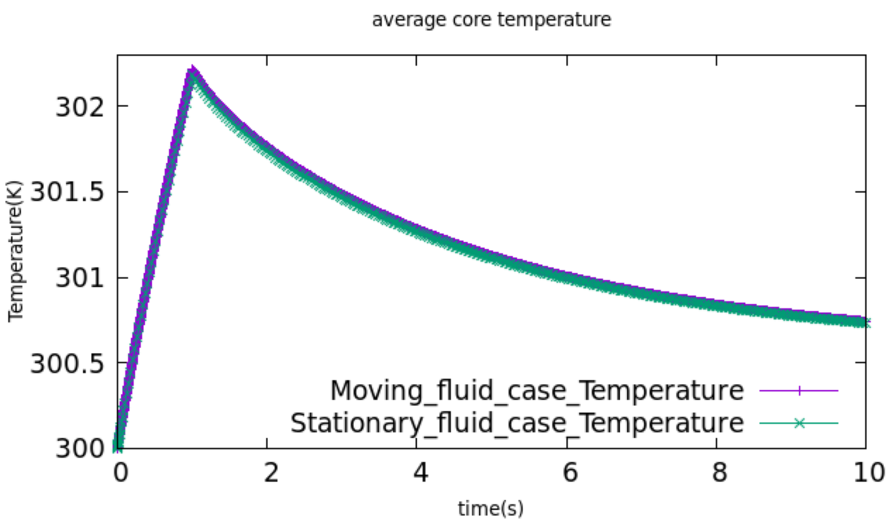


Figure 11: Comparison of average core Temperature of two cases

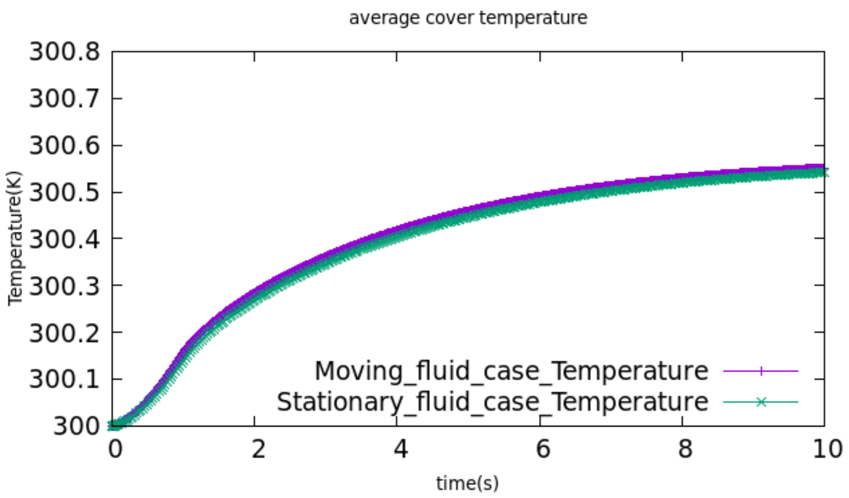


Figure 12: Comparison of average cover Temperature of two cases

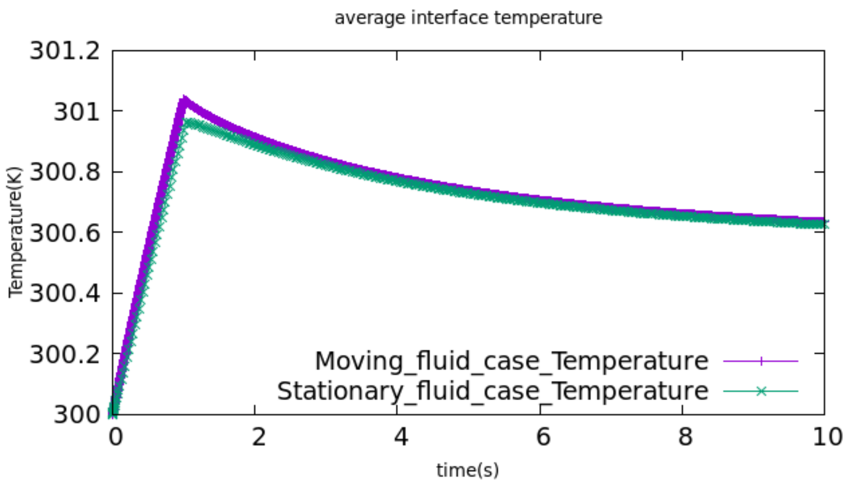


Figure 13: Comparison of average interface Temperature of two cases

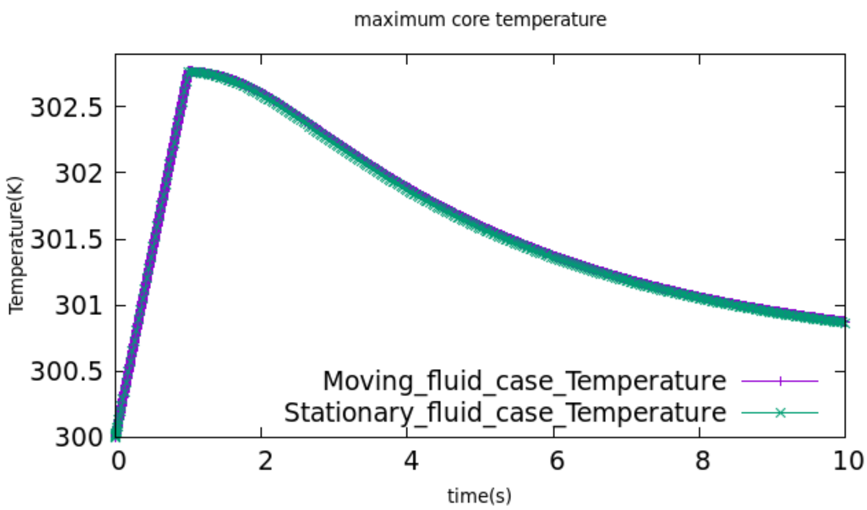


Figure 14: Comparison of maximum core Temperature of two cases

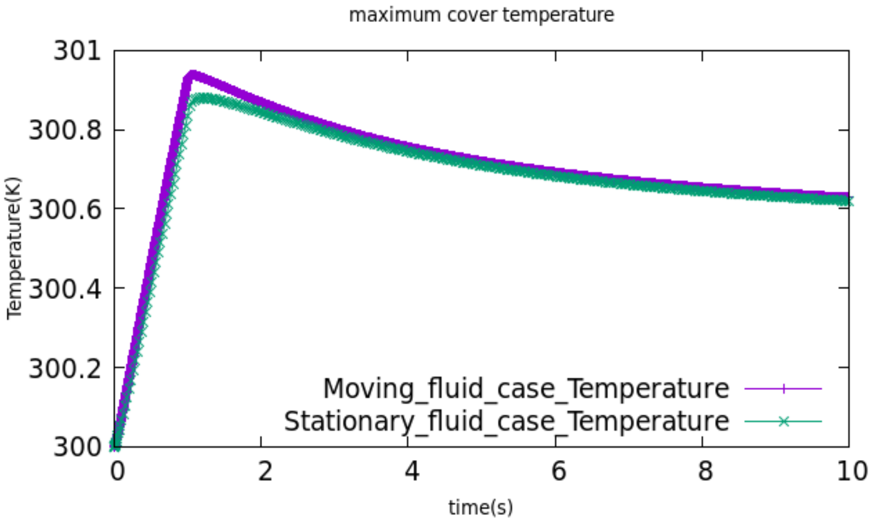


Figure 15: Comparison of maximum cover Temperature of two cases

Average Nusselt number was plotted against time with an interval of 1 second. Nusselt number is decreasing with time.

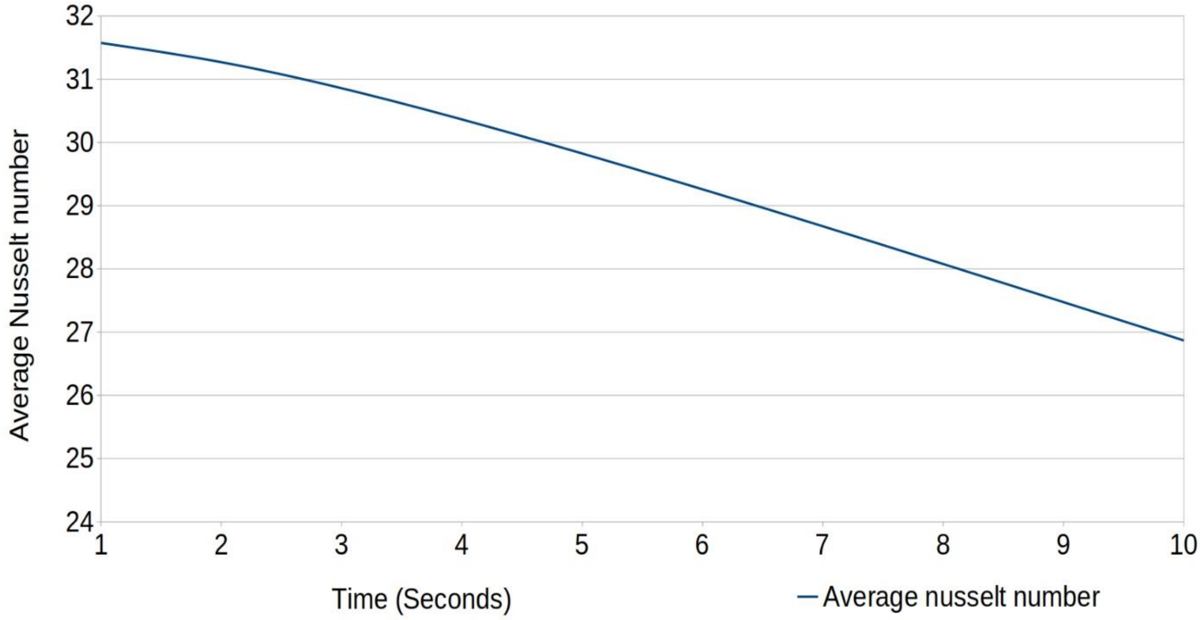


Figure 16: Variation of average Nusselt number with time for natural convection

## Conclusions

The following table shows the increase in percentage of Temperature in case 1 with respect to case 2.

| | Percentage variation at 1 second | Percentage variation at 10 seconds |
|---|---|---|
| Maximum core Temperature | 0.000957% | 0.00345% |
| Average core Temperature | 0.01217% | 0.00315% |
| Average interface Temperature | 0.03346% | 0.00276% |
| Maximum cover Temperature | 0.02985% | 0.002794% |
| Average cover Temperature | 0.00543% | 0.00302% |

Table 6: Percentage variation in Temperature

The maximum variation is in the case of average interface Temperature at 1 second. The least variation is in the case of maximum core Temperature at 1 second. Streamline countour is similar to figure 5.22 in (Batson, 1990). The variation of parameter $Nu/Re^{0.5}$ with angle is similar to figure 5.24 in (Batson, 1990) at initial time period with an increase in local Nusselt number for middle range of angle.

The average heat transfer coefficient for the first case is increasing until it is almost constant while it is decreasing for the second case. Due to this more heat is being transferred to the fluid in forced convection than in natural convection and will cool faster. (Nawaf & Bashir, 2017) also agrees with this as Reynolds number increases the cooling time will be less.

# References

[1]Batson, Brett William, "Prediction of steady and unsteady flow and heat transfer over a cylinder in crossflow using a space-time marching algorithm", (1990), *Retrospective Theses and Dissertations*, 9390.
https://lib.dr.iastate.edu/rtd/9390

[2]Nawaf H. Saeid, Bashir S. Abusahmin, "Transient cooling of a cylinder in cross flow bounded by an adiabatic wall", (2017), *Asian Journal of Chemical Engineering* ,17(2), 17-26.
https://doi.org/10.22146/ajche.49552

[3]*FOSSEE spoken tutorial*, 2021, Retrieved from https://spoken-tutorial.org/tutorial-search/?search_foss=OpenFOAM&search_language=English

[4]*CFD Direct User Guide*, 2021, Retrieved from https://cfd.direct/openfoam/user-guide/

[5]*CFD online forum*, 2021, Retrieved from www.cfd-online.com/Forums/openfoam/