FOSSEE, IIT Bombay
OpenFOAM Case Study Project
May 10, 2022

Comparison of Fin Design for Convective Cooling of Heat Sinks

J Harikrishnan
Department of Mechanical Engineering, IIT Bombay

Abstract

This report aims to study the heat transfer between two heated blocks and water which flows around it. Three different shapes of fins are used in the study. The shapes used are square of 20 *mm* side, a trapezium of 20 *mm* height and 18 *mm* and 22 *mm* length for shorter and longer sides respectively. The flow is laminar and simulation type is 2D. OpenFOAM's conjugate heat transfer solver, *chtMultiRegionFoam*, is used for the study. The blocks are heated for 1 second. The temperature variation of each shape is plotted, compared and the efficient shape among those selected for this study is identified.

1 Introduction

Heat sinks are used widely in many industries. Especially in the field of electronics, where dissipation of heat from components is crucial to protect them from thermal degradation and to prolong their lives. Fins are also used for automotive applications. The fins increase the effective area of heat transfer. The most important application of fins comes where the heat transfer coefficient is less and there is the requirement to remove heat. The flow over blocks is a generalized case. The case can be applied to areas to study cooling characteristics. An area of direct application may be the study of fin. The shapes can be considered as cross sections of fins in 2D.

The flow simulated in this is a derived case from conjugate heat transfer tutorial. Here 3 different geometries of iron, namely square, circle and trapezium are placed in a duct and water flows over them from the inlet. The dimensions of geometry are given such that all 3 shapes have equal volume per unit length. For a particular shape, two of those geometries are placed one behind the other.

2 Problem Statement

The problem statement is to simulate a laminar flow around two heated blocks placed back to back. Three shapes (a square, circle and trapezium) for the cross section of the fins are considered. The heating of the block has to be given by *fvOptions* and specific heat generation is 1e7. The flow inlet velocity is 0.001m/s and the fluid is water. The block material is iron. The heating have to be given for 1 second and total flow is simulated for 10 seconds

3 Governing Equations

The equations solved in the fluid region are conservation of mass, incompressible Navier Stokes and Energy Equation.

Fluid equations:

Mass Conservation equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0$$

Momentum conservation equation

$$\frac{\partial \rho u_j}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_j} + \rho \varepsilon_{ijk} \omega_i u_j = \frac{-\partial p_{rgh}}{\partial x_i} - \frac{\partial \rho g_i x_j}{\partial x_i} + \frac{\partial (\tau_{ij} + \tau_{tij})}{\partial x_j}$$

Energy conservation equation

$$\frac{\partial \rho e}{\partial t} + \frac{\partial \rho u_j e}{\partial x_j} + \frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_j k}{\partial x_j} = \frac{-\partial (q_i + q_{ti})}{\partial x_i} + \rho r + Rad - \frac{\partial \rho u_j}{\partial x_i} - \rho u_j \mathrm{g}_j + \frac{\partial \tau_{ij} u_i}{\partial x_j}$$

in our case MRF and radiation model is not used

Solid equation:

And for the solid region 2D unsteady state conduction equation with heat generation is used.

$$\frac{\partial (\rho h)}{\partial t} = \frac{\partial}{\partial x_j} \{ \alpha \frac{\partial h}{\partial x_j} \}$$

The source term will be added to the above equation in the fvOptions file present in the constant folder.

Coupling between solid and fluid:

At interface temperature and heat flux are considered the same for both solid and fluid.

$$T_f = T_s$$

$$K_f \frac{dT_f}{dn} = -K_s \frac{dT_s}{dn}$$

The convective heat transfer is found by the following equations

$$Nu=0.664Re^{1/2} Pr^{1/3} \quad (Re<5\times105)$$

$$Nu=0.037Re4/5\ Pr^{1/3} \quad (Re\geq5\times105)$$

4 Simulation Procedure

4.1 Geometry and Mesh

The flow domain here is a 2D domain. The flow comes from the left and top and bottom walls are noSlip walls. The blocks are shown in the middle with red and green colour.

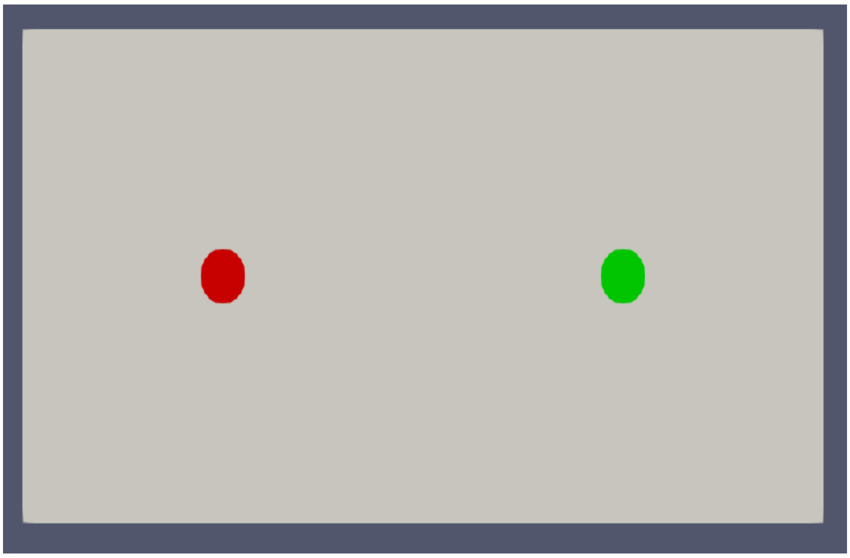


Figure1: Geometry for Circular Cross Section

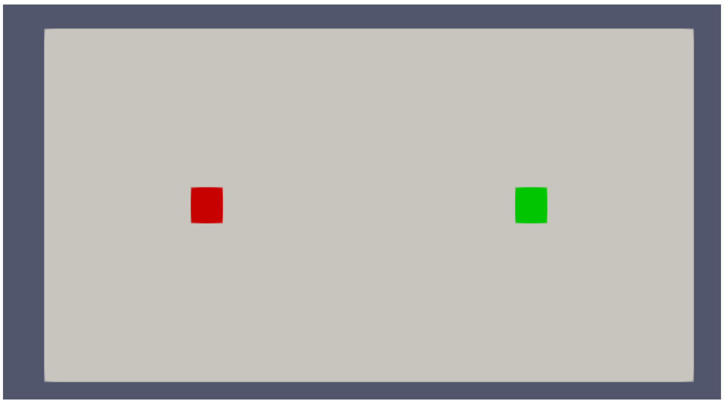


Figure 2: Geometry for Square Cross Section

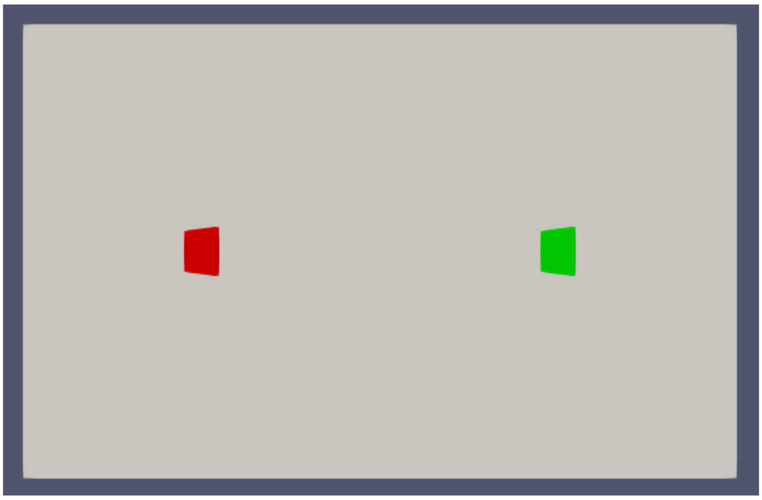


Figure 3: Geometry for Trapezoidal Cross Section

The meshes were generated using OpenFOAM's *blockMesh* command and all the necessary parameters are defined in each cases' blockMeshDict file, present in the *system* directory of each case.

The blocks dimensions are

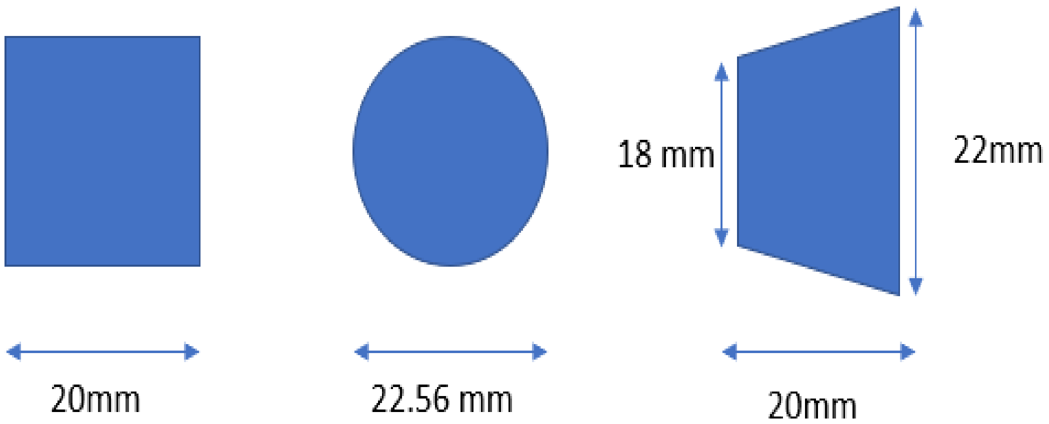


Figure 4: Dimensions of the Cross Sections of Fins

The dimensions of block are selected that the volume per unit length is same for all shapes

A structured uniform mesh is used. The images of mesh are shown below. To name zones topoSet is used for square and cylinder. For trapezium, naming of the zone is directly done through blockMeshDict.

In TopoSetDict, the commands *cylinderToCell* and *boxToCell* are used for creating the round cell set and square cell set for heaters. And for fluid the heater cell sets are subtracted from the whole domain.

After creating cellZones regions are created using *splitMeshRegions -cellZones -overwrite* command

For square and cylindrical mesh, the number of nodes used are 1003002. The high number of nodes are used to capture the circular profile of the circle or cylinder. The skewness for both these shapes are negligible in the order of 10e-13. For trapezoid the number of nodes is 1039554 with max skewness of 0.123. the images of meshes are as given

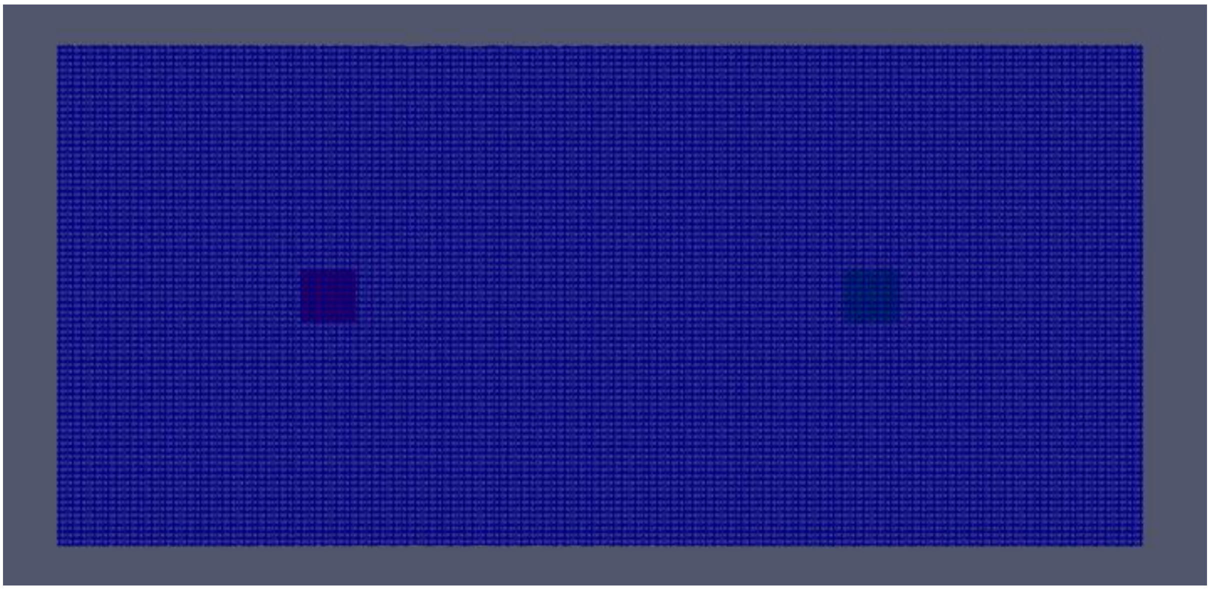


Figure 5: Total domain mesh for square shape

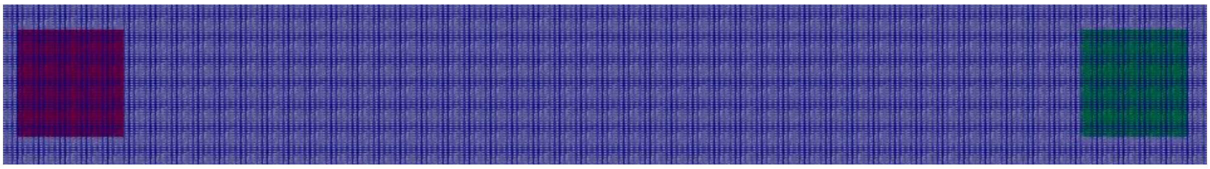


Figure 6: Mesh near heated block

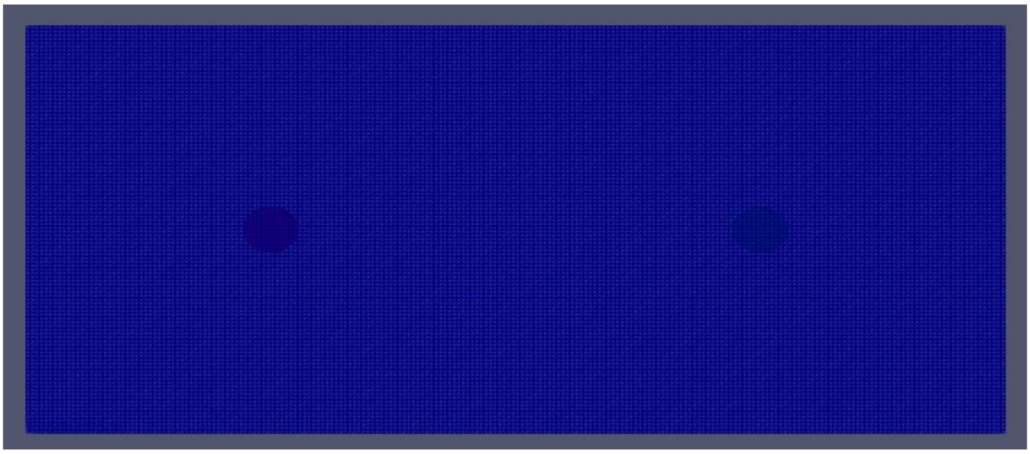


Figure 7: Total domain mesh for circular shape

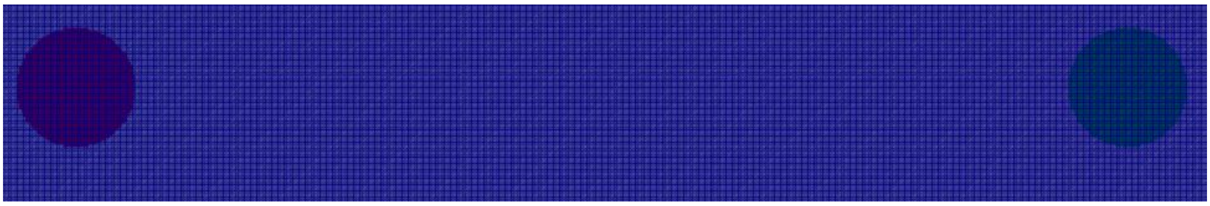


Figure 8: Mesh near heated block

For trapezium naming of regions is done in the blockMeshDict file.

Later *splitMeshRegions -cellZones -overwrite* command is used to create cell regions

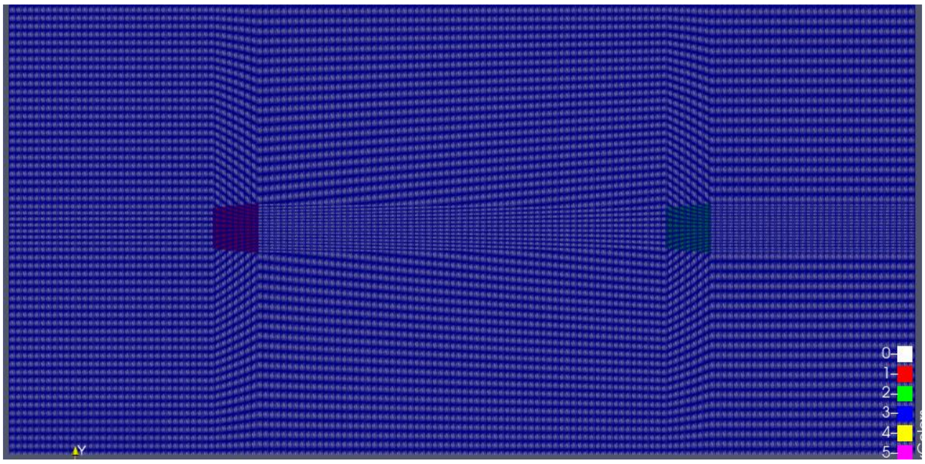


Figure 9: Total domain mesh for trapezium shape

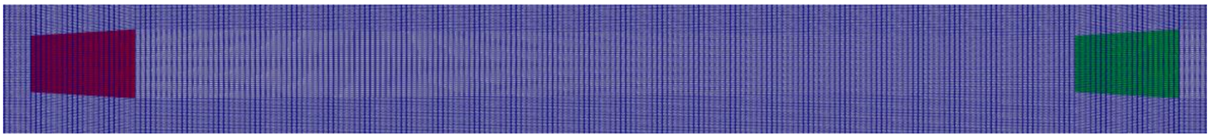


Figure 10: Mesh near heated block

4.2 Boundary and Initial Conditions

Velocity boundary conditions

inlet	Fixed Value (0.001m/s)
Outlet	pressureInletOutletVelocity
Fluid to heaters	No slip
Fluid to walls	No slip
Initial condition	Set to inlet value

P_rgh boundary conditions

Outlet	Fixed value (0 pa)
All other boundaries	fixedFluxPressure
Initial condition	Set to 0 at all places

Pressure boundary conditions

All boundaries (both water and heaters)	Calculated

Temperature boundary conditions

Inlet (water)	Fixed Value (300 K)
Outlet	inletOutlet
Fluid to heaters and heater to water	compressible::turbulentTemperatureCoupledBaffleMixed
All walls	Zero Gradient
Initial condition	300K at all region

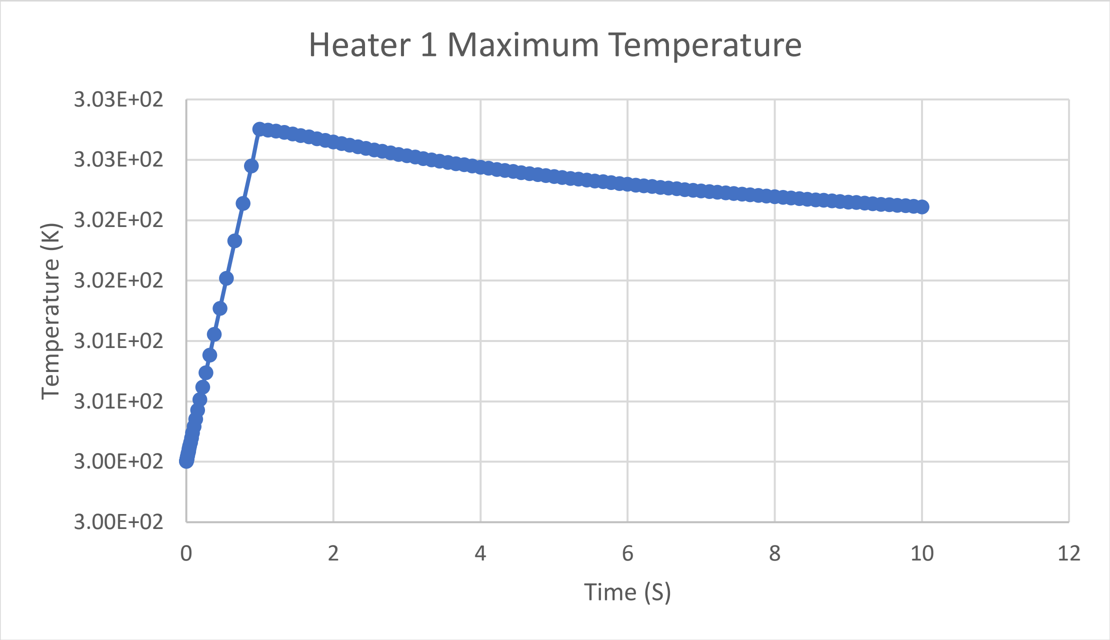
4.3 Solver

The solver used is *chtMultiRegionFoam* solver. The solver used a segregated approach to solve the solid and fluid equations. The solver is a pressure-based solver and uses pressure equations to establish connection between momentum and continuity equations.

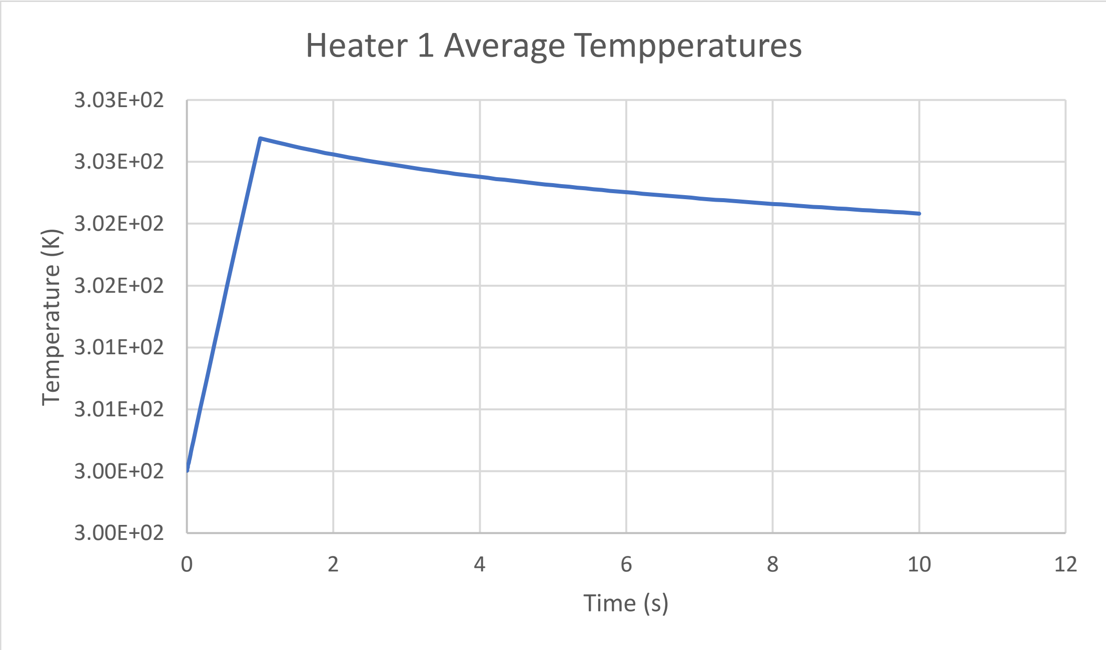
5 Results

For cylindrical block

The variation of Tmax in heater 1 was found as follow



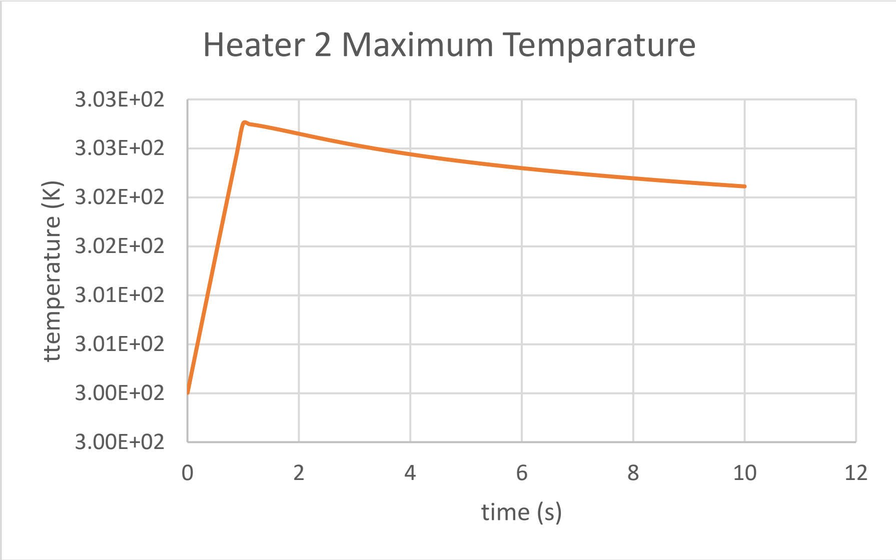
The variation of Tavg in heater 1 was found as follow

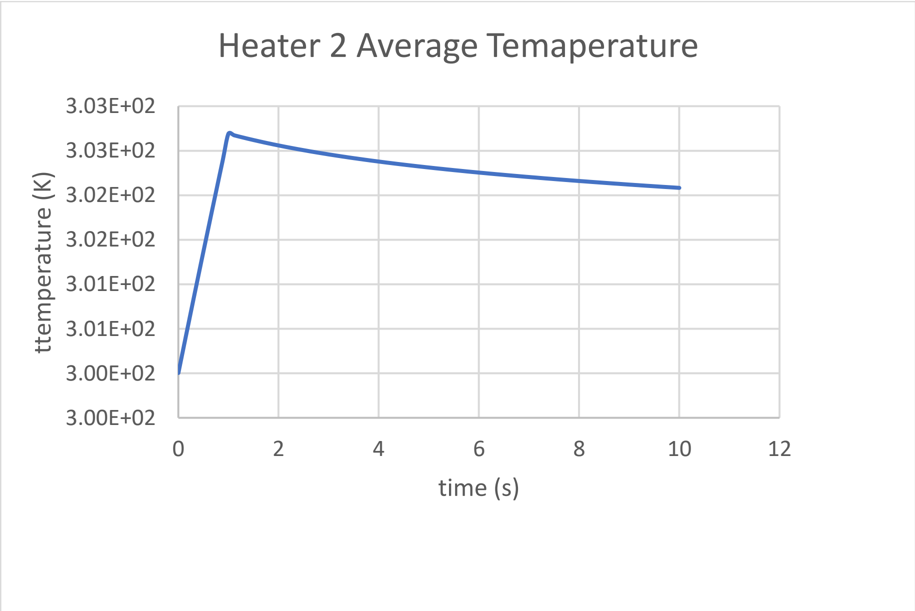


It can be seen that the temperature of the block increases steadily for the first second. This is due the heating provided in fvOptions and then due to convection by water the temperature decreases steadily.

A similar vitiation was observed for block2 or heater 2 also

Block 2 average and maximum temperature





The temperature distribution was as follows

T=1s

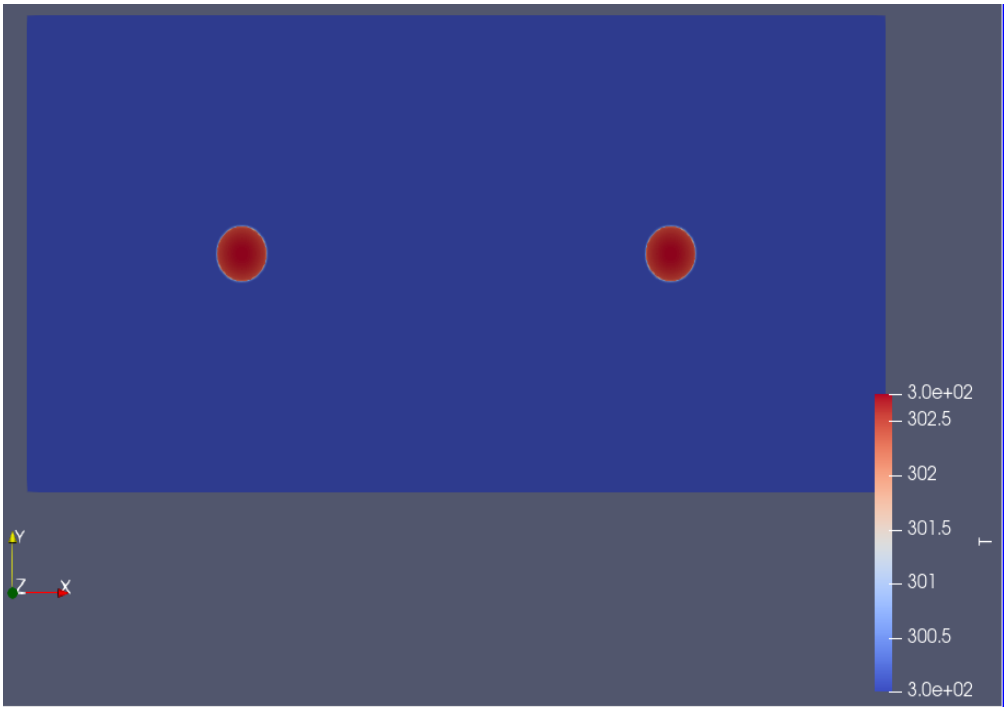


Figure 11: Temperature distribution at 1s

T=10s

Figure 12: Temperature distribution at 10s

The velocity distribution was found as

T=1s

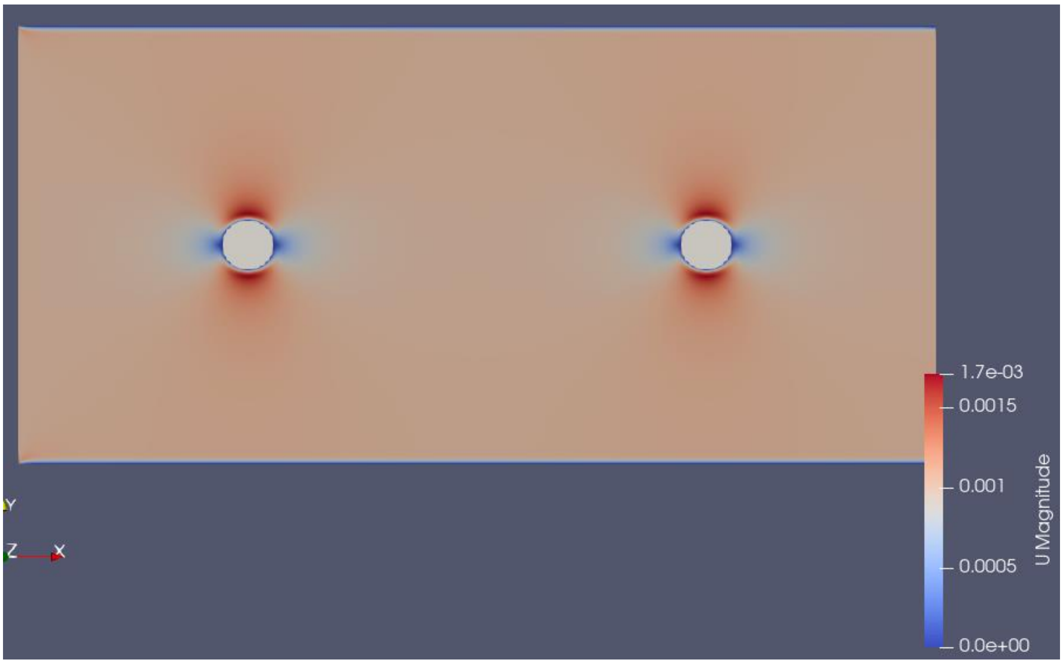


Figure 13: velocity distribution at 1s

T=10s

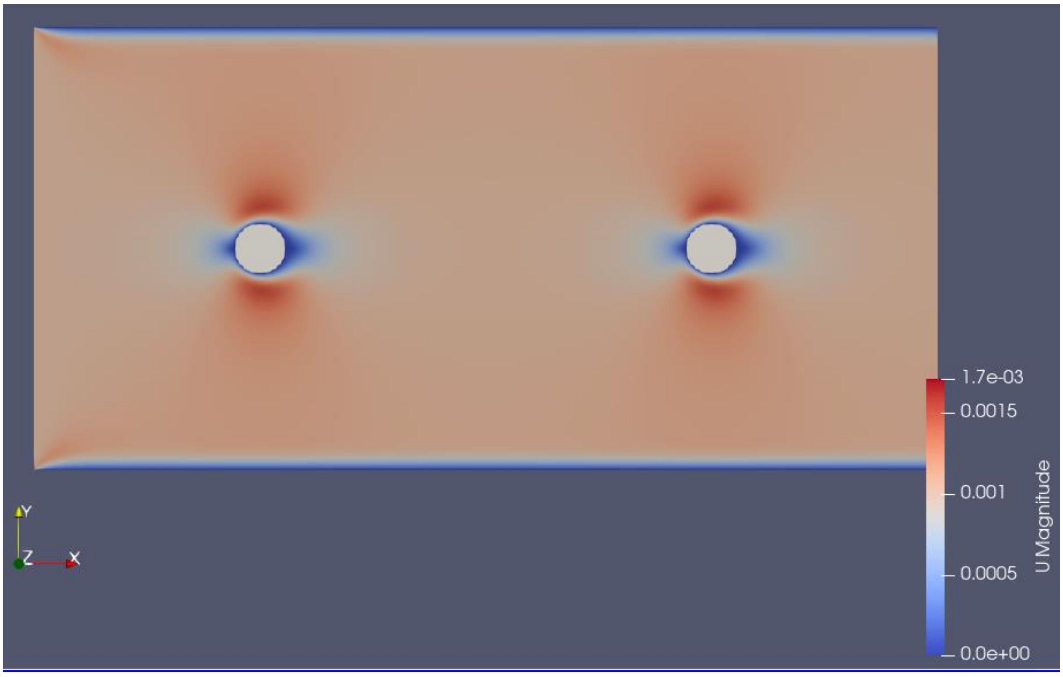


Figure 14: velocity distribution at 10s

The Pressure distribution was found as

At t=1s

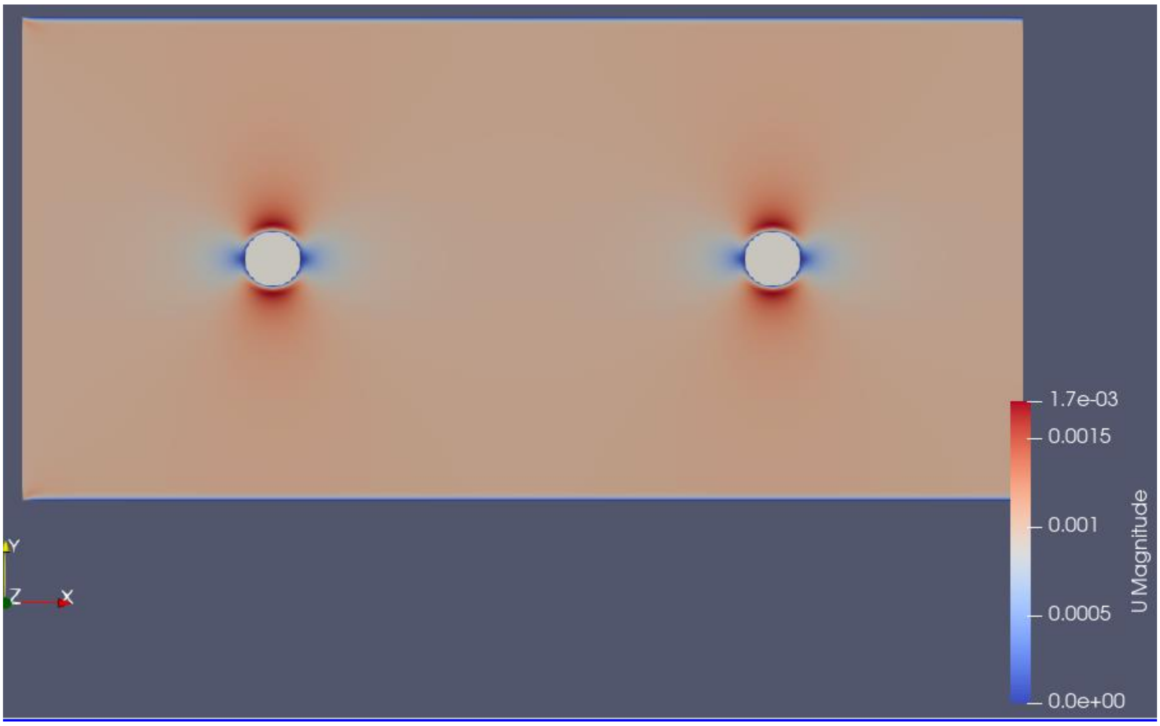


Figure 15: Pressure distribution at 1s

At t=10s

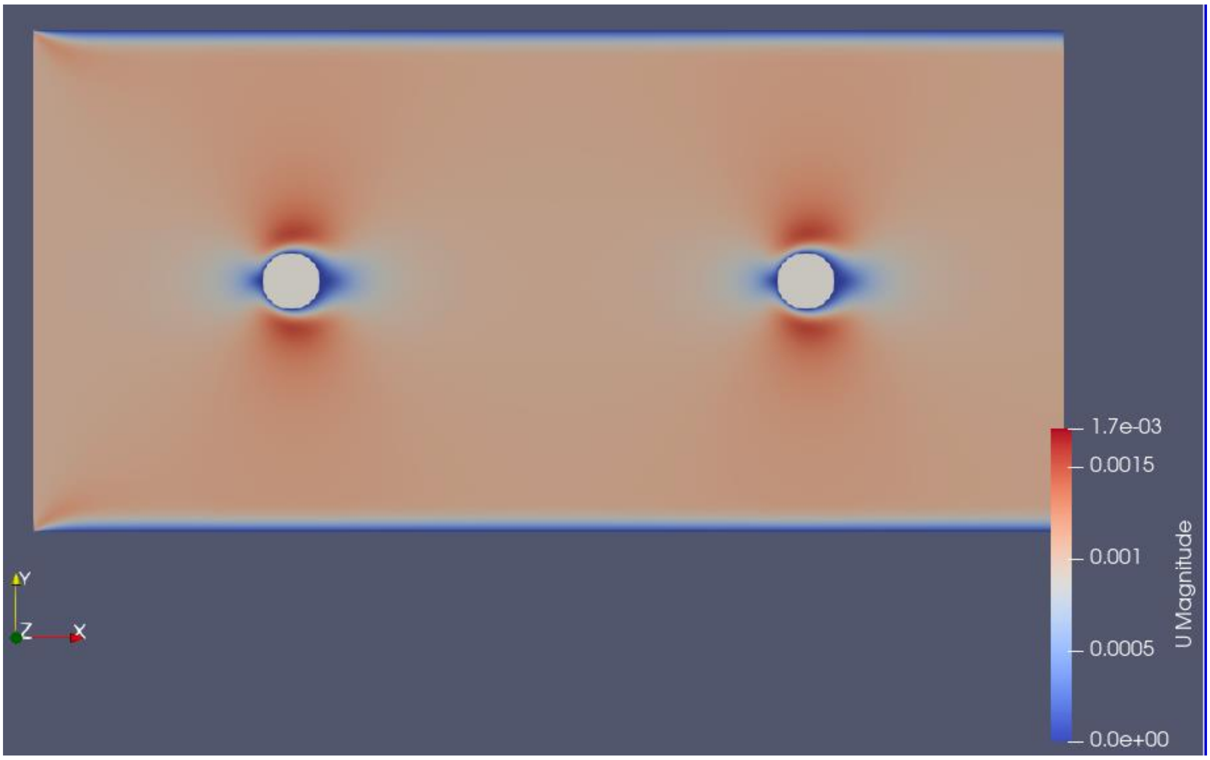
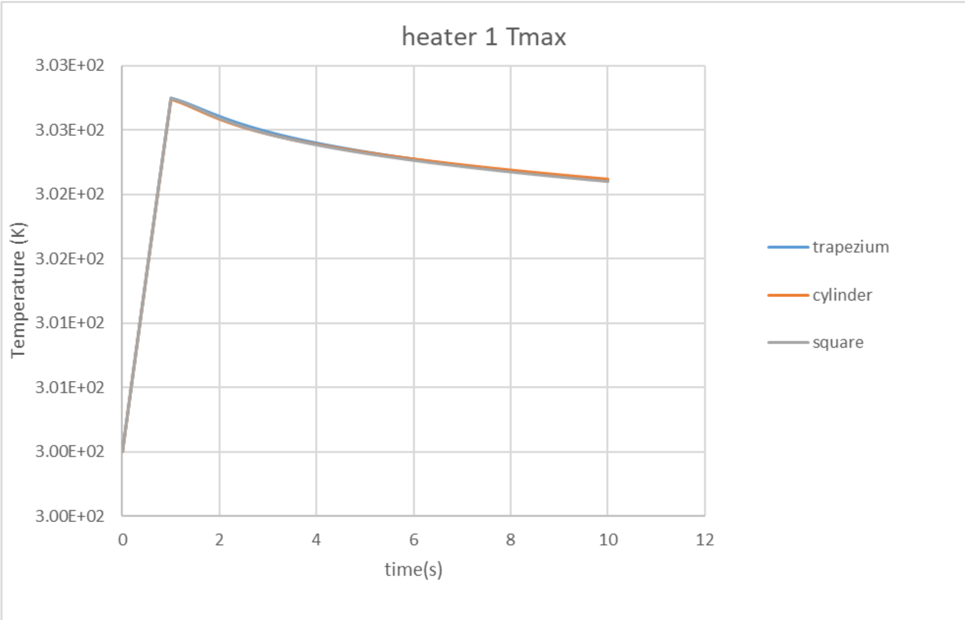


Figure 16: Pressure distribution at 10s

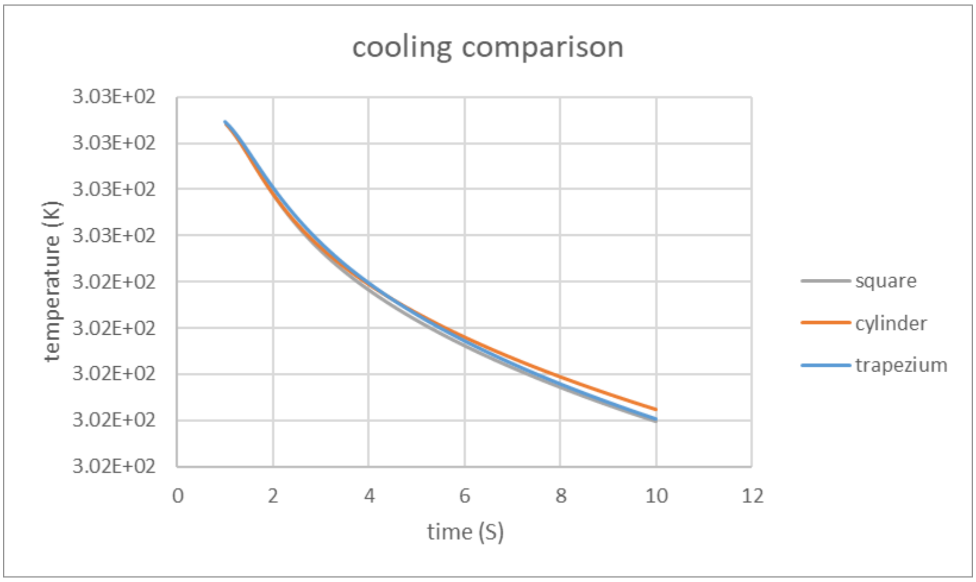
6 Conclusions

A similar result was obtained for other shapes too. Now to compare the results of the three shapes for heater 1

Tmax comparison



Comparison of cooling part



From the above plots, it can be seen that the temperature of the cylinder is the highest at the end of 10sec. This can be explained by here the volume of all shapes per unit length are taken as the same and surface area to volume ratio is lowest to circle or cylinder. It can be concluded that among the above shapes, fins with square cross section would be ideal for designing heat sinks for maximum dissipation.

7 Reference

[1] Ali, Hafiz & Ashraf, Muhammad & Giovannelli, Ambra & Irfan, Muhammad & Hamid, Muhammad & Hassan, Faisal & Arshad, Adeel. (2018). Thermal management of electronics: An experimental analysis of triangular, rectangular and circular pin-fin heat sinks for various PCMs. 10.13140/RG.2.2.29812.81285.