MEAM 5360: VISCOUS FLUID FLOW

Final Project: CPU Heat Sink Flow Analysis

Ilia Kheirkhah and Luyando Kwenda

Executive Summary

In this project, we investigate the impact of flow velocity, represented by the Reynolds Number, on the convective heat transfer of a plate heat sink using computational fluid dynamics (CFD) simulations. These simulations allow us to establish a correlation between the Nusselt Number and Reynolds number, with the exponent associated with the Reynolds number term in the correlation increasing, dependent on the Prandtl number. Our analysis focuses on Reynolds Numbers on the order of 10^2, maintaining laminar flow conditions for the plate heat sink. ANSYS Fluent is employed for these simulations, preceded by verification of its capability to accurately capture convective heat transfer at these Reynolds numbers through simulations over a flat plate. Comparison with correlations from the textbook "Viscous Fluid Flow" by Frank White validates our approach. Subsequently, we conduct simulations for four heat sink cases to gather sufficient data points for proposing a tailored correlation for this specific geometry.

Introduction

Through the years, computers have become more powerful and smaller. This leads to a steady increase in the power density as can be seen in Figure 1. A majority of this energy will be dissipated as heat, leading to increases in temperature for important components such as the CPU and GPU. Although these are designed to operate at a large range of temperature, performance can significantly decrease as we reach the upper range of allowable temperatures. Modern CPUs are programmed to limit their power draw as they approach their maximum operating temperature to ensure no damage is sustained. This results in the drop in performance commonly known as thermal throttling [4]. To keep temperatures under control, research is being done in cooling systems for computers [3].

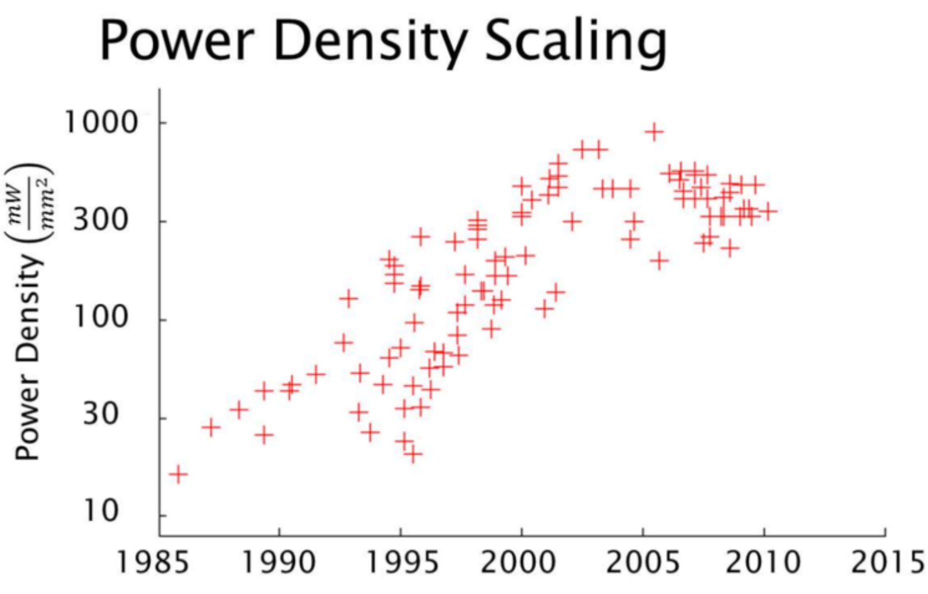

Figure 1: Power Density Trends[1]

With the wide range of possibilities in working fluid, geometry, and cost among other variables there is still no definitive best way to cool systems yet. This gap is only increasing with advances in manufacturing techniques such. The development of metamaterials, metal 3D printing, and advanced composites means a new range of possible design choices exist, much of which is yet to be explored. Some of these advances, such as metal 3D printing and other advanced manufacturing techniques, allow heat sinks to take geometries not possible through conventional means. In all these cases, we would like to compare the maximum temperature and the convective heat transfer. The maximum temperature can be calculated easily, but comparing convective heat transfer accurate can be difficult. Instead of using this dimensional value, we instead report the Nusselt Number, a nondimensional number looking at the ratio of total heat transfer to the conductive heat transfer.

$$Nu = \frac{hL}{k_f} \tag{1}$$

In this case, h is our convective heat transfer coefficient, L a characteristic length of the system, and k_f is the thermal conductivity of the working fluid. When working on forced convection systems, we can also use the Reynolds Number, which gives us an understanding of the flow characteristics, and the Prandtl number which gives us an understanding of the thermal and momentum diffusivities. These three nondimensional numbers fully characterize our system, and it is common to write power law correlations (see Eq. 3)

$$Nu = C \cdot Re^a Pr^b \tag{2}$$

At the same Reynolds number, we can consider a heat sink to be better for heat transfer if the Nusselt Number is higher. Establishing these correlations allows us to compare different heat sinks outside of cases which we have explicitly analyzed. In this paper, we will use CFD Simulations to establish a correlation to relate Nusselt Number to Reynolds number. As most computers are air-cooled, we will assume that the Prandtl number stays constant and only look at the Reynolds number variation.

Due to the scale of modern heat sinks, the Reynolds Number is often on the order of 10^2. In this study, we will look at Reynolds number ranging from 170 to 800. This is well within the Laminar flow regime. For flows at certain Reynolds number ranges and relatively simple geometries, correlations have already been established. A well known case of this is the convective heat transfer over a heated flat plate. For this case, the Local Nusselt Number (hx/k at some position X) on this geometry is given by:

$$Nu_x = 0.332 Re_x^{1/2} Pr^{1/3} \tag{3}$$

A flat plate can be used to approximate CPU cooling without any heat sink attached, as a CPU is thin itself. Improvements could be seen in two factors. Assuming we are using the same working fluid (Pr = constant), Increases to the exponent of the Reynolds number and increases to the constant coefficient will result in better average heat transfer.

Problem Formulation

To analyze the fluid flow, we can use the Incompressible Navier-Stokes Equations. Although temperature changes in air can lead to density differences, causing natural convection, these flow patterns are of a much smaller order of magnitude than the freestream flow. Natural convection generally has a speed on the order of 10^{-3}, very small in comparison with our forced flow.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad \text{(Continuity Equation)}$$

$$\rho\left(\frac{\partial u}{\partial t}+\vec{u}\cdot\nabla u\right)=-\frac{\partial p}{\partial x}+\mu\left(\nabla^2 u\right)+\rho g_x \quad \text{(X-Momentum Equation)}$$

$$\rho\left(\frac{\partial v}{\partial t}+\vec{u}\cdot\nabla v\right)=-\frac{\partial p}{\partial y}+\mu\left(\nabla^2 v\right)+\rho g_y \quad \text{(Y-Momentum Equation)}$$

$$\rho\left(\frac{\partial w}{\partial t}+\vec{u}\cdot\nabla w\right)=-\frac{\partial p}{\partial z}+\mu\left(\nabla^2 u\right)+\rho g_z \quad \text{(Z-Momentum Equation)}$$

Since we are ignoring the influence of gravity in this driven flow, we can drop the last term. If we then nondimensionalize these equations with a reference length L, a reference velocity U and the viscosity μ, one can show that the nondimensional Navier Stokes Equations Become:

$$\begin{aligned}
\nabla^\star\cdot\vec{u}^\star &= 0 && \text{(Continuity)}\\
\frac{\partial u^\star}{\partial t^\star}+\vec{u}^\star\cdot\nabla^\star u &= -\frac{\partial p^\star}{\partial x^\star}+\frac{1}{Re}{\nabla^\star}^2 u^\star && \text{(X-Momentum)}\\
\frac{\partial v^\star}{\partial t^\star}+\vec{u}^\star\cdot\nabla^\star v &= -\frac{\partial p^\star}{\partial y^\star}+\frac{1}{Re}{\nabla^\star}^2 v^\star && \text{(Y-Momentum)}\\
\frac{\partial w^\star}{\partial t^\star}+\vec{u}^\star\cdot\nabla^\star w &= -\frac{\partial p^\star}{\partial z^\star}+\frac{1}{Re}{\nabla^\star}^2 w^\star && \text{(Z-Momentum)}
\end{aligned} \tag{4}$$

Where

$$\vec{u}^\star=\frac{\vec{u}}{U} \tag{5}$$

and other similar nondimensionalizations. From this nondimensionalization, we can understand why the Nusselt number primarily depends on the Reynolds Number.

In addition using the Navier-Stokes equation to model the fluid flow, we need to include the Energy equation to model the temperature transport.

$$\frac{\partial T}{\partial t}+\vec{u}\cdot\nabla T=\alpha\nabla^2 T \quad \text{(Energy Equation)}$$

These equations are all coupled. While the Navier-Stokes equations have two-way coupling, where all equations are coupled with all other equations, the Energy equation only has one-way coupling as temperature does show up in the Navier-Stokes equations. This means that the Navier-Stokes equations can be solved first and the velocity information used in the heat equation to advance the temperature. This is done in ANSYS using the SIMPLE algorithm.

If we nondimensionalize the energy equation just as we did the Navier-Stokes equations, the result is below. From the equation, we see the only parameters that can be controlled are the Reynolds and the Pradtl Numbers. This is shows us why, in addition to Reynolds Number, our correlation can depend on Prandtl Number. However, since we will be using a constant working fluid (air) for our entire analysis, this can be ignored.

$$\frac{\partial T^\star}{\partial t^\star}+\vec{u}^\star\cdot\nabla^\star T^\star=\frac{1}{RePr}{\nabla^\star}^2 T^\star \tag{6}$$

From this, we can define our Nusselt Number at the surface as:

$$Nu = \frac{hL}{k} = \left| \frac{\partial T^\star}{\partial y^\star} \right|_{y^\star=0} \tag{7}$$

In this case $y^\star = 0$ does not necessarily mean $y = 0$, but instead the surface of heat sink. Once this is calculated, we can get the area averaged Nusselt Number as:

$$\overline{Nu} = \frac{1}{A} \iint NudA \tag{8}$$

Where we integrate over the surface of the heat sink.

We know that we have 5 Unknowns in temperature, pressure, and 3 components of velocity. The Momentum, Continuity, and Energy equations give us 5 equations for these unknowns, allowing us to solve this system. However, due to the complexity of this geometry and the type of flow, this is not a problem that can be solved by hand. This is why we use computational fluid dynamics (CFD) to solve the problem.

The idea with CFD is to break our domain of interest up not many control volumes. We then assume that the properties of our fluid are constant in each of these small control volumes. We then solve the differential equation on this discretized domain. The method used by ANSYS Fluent is the Finite Volume Method, which integrates our governing equation over each control volume (with the constant properties assumption) and gets a large system of equations. This can then be solved using a nonlinear equation solver to get a final result. The Nonlinear solver ANSYS uses is the SIMPLE method with Pseudo-Time Stepping for steady problems. To use this method, ANSYS assumes the problem is unsteady and integrates in time until a steady state solution is reached. In each time step, ANSYS solves the governing equations on each control volume many times until our residuals are below a certain threshold. This gives us a result that is accurate and stable within a reasonable time.

Although the methods ANSYS, as with most commercial software, use are almost unconditionally stable, convergence of residuals does not necessarily imply a trustworthy solution. A good quality mesh (Our discretized domain) is required to get trustworthy results. In addition, proper selection of the models, settings, and boundary conditions are required to make sure that the simulation reflects reality as closely as possible. As such, it is important to validate all methods to ensure they give trustworthy results. For our problem, we need to ensure that ANSYS can capture forced convection heat transfer well. As we discussed before, a canonical case of forced convection heat transfer is the flat plate thermal boundary case. This case has a well-established solution and well-defined boundary conditions.

Given the fundamental equations, boundary conditions are needed to use. Below, one can see the geometry we will be analyzing. This is a plate heat sink on the scale of common CPUs. There is a square base size of approximately 40mmx40mm, a plate height of 25mm, a pin spacing of 3.3mm, and a pin width of 1mm. The fluid domain was established to ensure no numeric boundary effects will influence the flow.

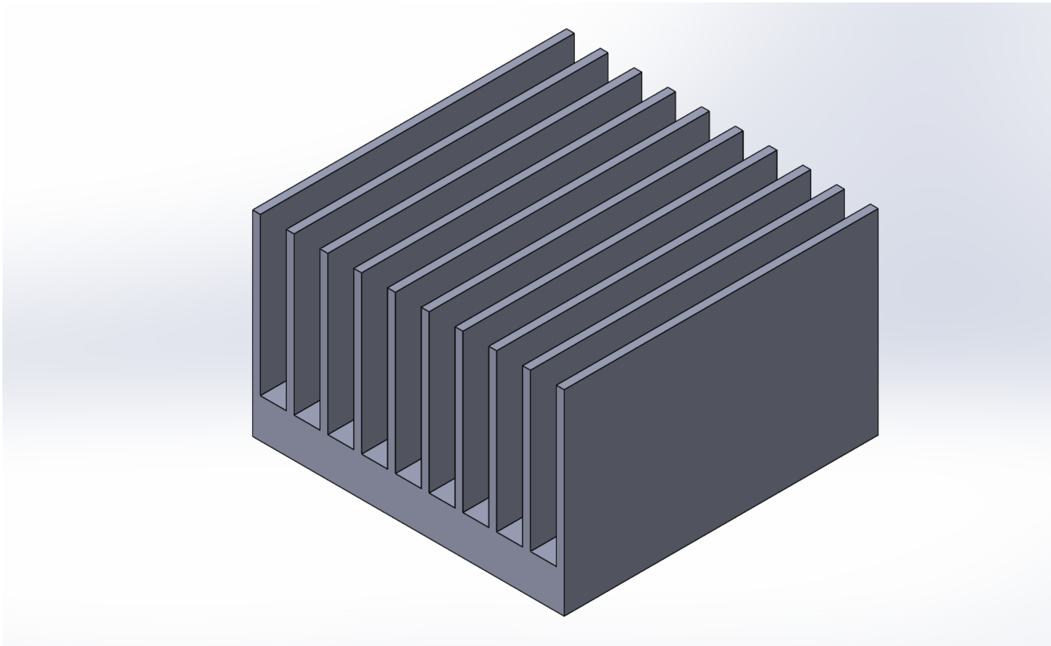

Figure 2: Geometry

First, we will look at our flow boundary conditions. At the inlet, we prescribe a flow velocity corresponding to each Reynolds number. At the outlet, we prescribe a pressure outlet. This is a simple outlet that allows for backflow if necessary. On the bottom surface and all the heat sink surfaces we apply a no slip condition. Finally, on the sides of the domain we apply a freestream symmetry boundary condition as we assume these boundaries are far enough away to have little influence on the flow near the heat sink.

We also need boundary conditions for the energy equation. At the Inlet, we prescribe a temperature of 20C, corresponding to room temperature. At all other walls except the heat sink, we prescribe an adiabatic condition. To treat the Heat sink surface requires a little bit more thought. For analysis of computer components, we are often provided a heat flux applied to the bottom of the heat sink, but this is not uniformly emitted on the surface of the heat sink. To see how this varies, what is required is to mesh the heat sink and solve the heat equation for the solid heat sink also. This means that we have two heat equations to solve, one for the fluid domain and one for the heat sink domain. By solving the heat sink heat equation with convection boundary conditions on the fluid surfaces and our prescribed heat flux (18,750 W/m^2) at the bottom surface, we can get the temperature distribution of the heat sink. From here, we can assign this temperature to the fluid on the heat sink boundaries. This is done through ANSYS by doing an explicit coupling of these systems.

Now that we understand our boundary conditions, it is important to make a good mesh for this problem. Since ANSYS has a mesh limit of about 1M cells, we got as close to the maximum allowable cell count, at around 900K cells. This gives us a mesh that is fine enough to resolve all the flow properties necessary. Now that we have a fully defined problem, we can simulate to get results. However, before we simulate the full system, we need to validate our system. This serves to ensure ANSYS Fluent, with our current settings, can capture forced convection heat transfer accurately.

Validation

To validate our system, we ran a sample simulation of a laminar flat plate thermal boundary layer. As discussed before, the local Nusselt Number correlation for this problem is well known.

$$Nu_x = 0.332 Re_x^{1/2} Pr^{1/3}$$

By simulating this system in ANSYS, we can get the local Nusselt Number also. The mesh for this problem can be seen in Figure 5 of A1. The inlet flow velocity was picked to correspond to Re_x ranging from 0 to 1000, around what we expect the heat sink Reynolds number to be.

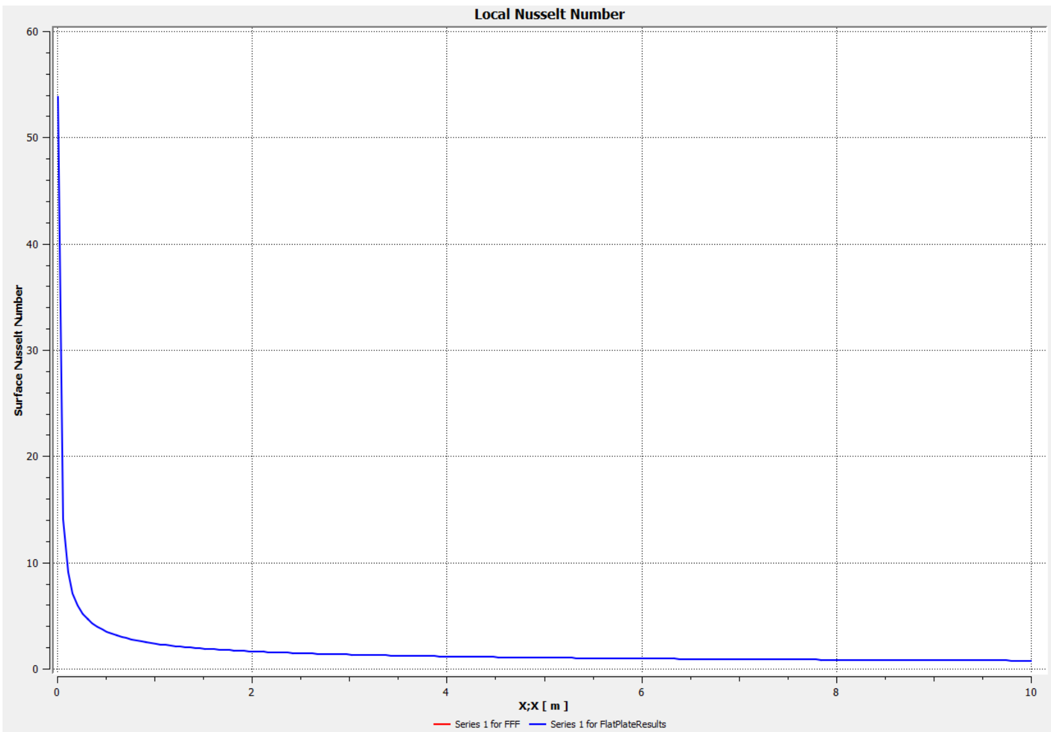

Figure 3: Flat Plate Nusselt Number

One can see the Nusselt Number as a function of Re_x for both the ANSYS simulation and the correlation in 3. We see good agreement between the established correlation and our simulation results. This tells us that ANSYS Fluent is fully capable of analyzing forced convection problems and accurately finding the Nusselt Number.

In addition to accurately finding the Nusselt Number accurately, we can see that the temperature profile also shows the clear development of the thermal boundary layer as expected (A1, Fig. 6). This shows us that we can also trust the temperature profiles we will get from the analysis of our heat sink.

Given this validation study, we know that we can trust the Nusselt Number and maximum temperature results we get from our simulation of the heat sink. It also reinforces the credibility and effectiveness of our computational approach in analyzing and optimizing heat sink designs for enhanced convective heat transfer performance.

Results

We can the simulations of the Heat sink to a steady state solution. This was done using the SIMPLE algorithm and by keeping track of residuals of energy, momentum, and mass. All residuals converged to our threshold of 10^{-3}. We also kept track of the Residual of our Nusselt number and average heat sink temperature to ensure they do not vary significantly once we reach steady state. These values also converged with our simulation.

We can four cases of our simulation, corresponding to $Re = 170, 400, 600, 800$. We calculated the area averaged Nusselt Number. Results for this are shown in Table

Re	170	400	600	800
Nu	945	1676	2250	2829

Table 1: Nusselt Number Results

We can use Desmos to do Least Square Curve Fitting to fit our Nusselt Number to the form $Nu = C \cdot Re^n$. Doing this, we can values of $C = 22.24, n = 0.72$. Although it may be difficult to compare this to other values, as our characteristic length was set to 1 by default in ANSYS, we can compare the growth behavior easily as this is encapsulated in the exponent. Recall that for the flat plate case, we have an exponent for our Reynolds Number of 0.5. However, for this case we have a value significantly larger than this. This information tells us that having a heat sink allows us to get more heat transfer out of the same increase in flow speed. Since the flow is normally induced by a fan we must power, this is essentially telling us we get significantly more heat transfer per unit of power we supply for cooling.

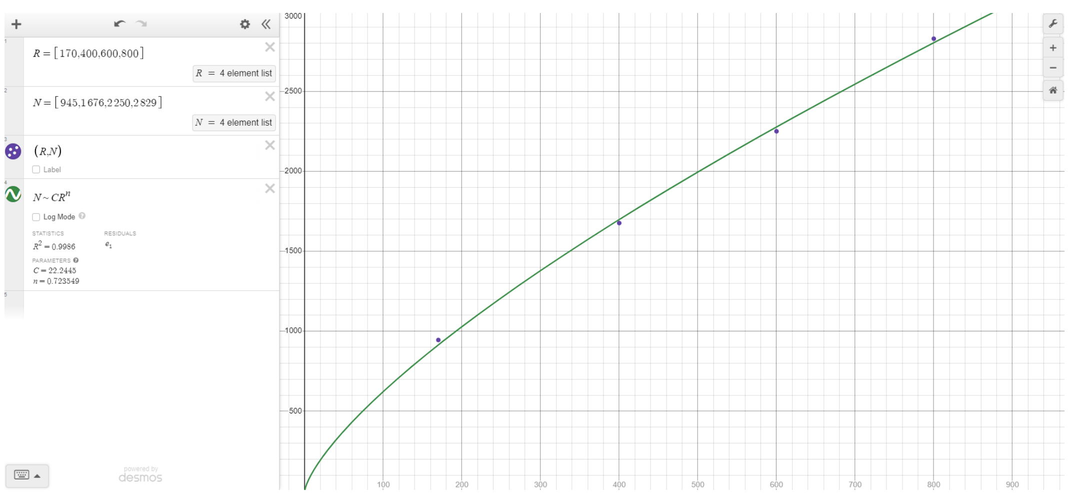

Figure 4: Nusselt Number Curve Fit

We can also look at the maximum temperature on the heat sink at each time. This is directly proportional to the temperature of our CPU, and will allow us to get another idea of the efficiency of our heat sink.

Re	170	400	600	800
Tmax [C]	89	58	48	42

Table 2: Nusselt Number Results

As can be seen, increasing the Reynolds number by increasing the flow velocity directly leads to a decrease in the maximum temperature. Doing a similar curve fitting, the following relationship can be developed:

$$T_{max} = 1095Re^{-0.49} \tag{9}$$

By looking at the Midline Velocity Contours in teach simulation, we can see that although he flow fields seem similar, the velocity gradients increase as we go up in Reynolds number. As a result, we know that this can lead to greater energy transport. This is the driving force in reducing the temperature.

Looking at the behavior of Nusselt number on the heat sink, we see that, just like in the flat plate cases, the Nusselt number is high at the front. This is more clearly seen in the Nusselt Number curves taken along the top of the central pin for each simulation. In these graphs, we see that although the Local Nusselt number decreases quickly, there is a significantly higher Nusselt number at the front of a pin.

Conclusion

From our analysis, we know we that a heat sink has a significant influence in increasing the convective heat transfer. This was encapsulated in the increase of the exponent associated with the Reynolds number term in the Nusselt Number correlation. The increased cooling effectiveness leads to a decrease in the maximum temperature. For a real computer, this would lead to better performance in two ways. For a constant power draw, our temperature would be lower which prevents thermal throttling. For a given ideal operating temperature, improved heat sinks lead to more power being able to be drawn by the CPU, increasing performance.

Although this analysis gives good insight into the performance of this specific geometry and conditions, there is still further work that can be done. A first step would be to introduce different working fluids and get an understanding of how Prandtl number influences this correlation. By doing this, we can change our model to account for other possible working fluids such as water or specialized fluids.

If one is to use a different working fluid, different limitations can be applied. With air, we assumed a symmetry boundary conditions on most of our sides, as a fan inducing flow often does not restrict the air in any way. As such, the air can flow out easily. If one is using water as a coolant, piping is necessary to keep the water in control and in cycle. As such, different boundary conditions may need to be used in a simulation in addition to changing the enclosure geometry. Although the heat sink geometry itself may not change, this change in surrounding geometry can have a significant impact on the transport properties. Recall that water in a pipe generally has a parabolic velocity profile if it is laminar. In these cases, our inlet velocity profile would also need to change.

In designing a high power computing system, our only concern may not be simply maximizing heat transfer. Cost is a significant factor, especially in larger systems. For example, many high power computing (HPC) systems need cooling systems. Although liquid cooling is often considered more efficient, not all HPC systems use liquid cooling as it can be expensive. In addition, the work required to pump the fluid through a large system will often be greater than the work required to use a fan to slightly accelerate air. This means that there is also a continuously increased cost due to the increased work required to induce the necessary flow rates. Similarly, the geometry we analyzed is relatively simple and would be cheap to manufacture in bulk. If a more complex geometry was to be used, increased manufacturing costs would also need to be considered. A full scale analysis of a cooling system should include all of the components and may look at factors such as the heat transfer per dollar, heat transfer normalized by pump/fan work, or a Return on Investment analysis.

In addition to this, a wider range of Reynolds numbers should be analyzed to get a full understanding of our heat transfer. If one was to assume the correlation we discovered held for all Reynolds number, error would accrue. To get an idea of this, consider the Nusselt Number correlations for a cylinder. These are given in Table 3

Re Range	Correlation
0.4-4	$\overline{Nu_D} = 0.989 Re_D^{0.33} Pr^{1/3}$
4-40	$\overline{Nu_D} = 0.911 Re_D^{0.385} Pr^{1/3}$
40-4,000	$\overline{Nu_D} = 0.683 Re_D^{0.466} Pr^{1/3}$
4,000-40,000	$\overline{Nu_D} = 0.193 Re_D^{0.618} Pr^{1/3}$
40,000-400,000	$\overline{Nu_D} = 0.027 Re_D^{0.805} Pr^{1/3}$

Table 3: Cylinder in Cross Flow Correlations

As can be seen, the correlation for different Reynolds Number ranges changes. A similar analysis should be done for our geometry to understand the full range of possibilities.

Overall, we can see that CFD is a powerful tool in analyzing complex geometries with multiple physical mechanisms. Numerical methods allow engineers to get insight into these complex problems that cannot be found easily with hand calculations and could sometimes be expensive to set up experimentally.

References

[1] Andrew Danowitz. "Exploring abstract interfaces in system-on-chip integration". In: (July 2014).

[2] Basim Freegah et al. "CFD analysis of heat transfer enhancement in plate-fin heat sinks with fillet profile: Investigation of new designs". In: *Thermal Science and Engineering Progress* 17 (2020), p. 100458. ISSN: 2451-9049. DOI: https://doi.org/10.1016/j.tsep.2019.100458. URL: https://www.sciencedirect.com/science/article/pii/S2451904919302604.

[3] Jackson B Marcinichen et al. "Advances in Electronics Cooling". In: *Heat transfer engineering.* 34.5-6 (2013-01). ISSN: 0145-7632.

[4] Diary R. Sulaiman. "Microprocessors thermal challenges for portable and embedded systems using thermal throttling technique". In: *Procedia Computer Science* 3 (2011). World Conference on Information Technology, pp. 1023–1032. ISSN: 1877-0509. DOI: https://doi.org/10.1016/j.procs.2010.12.168. URL: https://www.sciencedirect.com/science/article/pii/S1877050910005430.

Appendix

A1. Flat Plate Thermal Boundary Figures

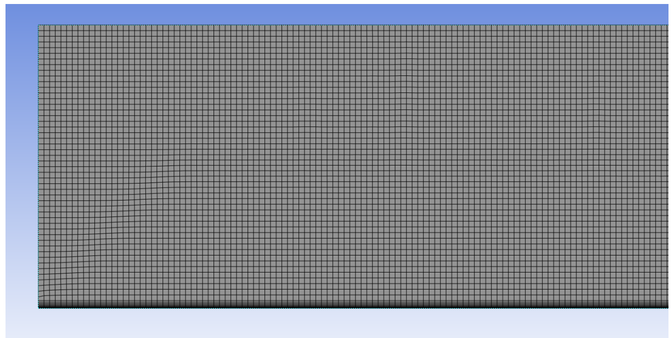

Figure 5: Flat Plate Boundary Layer Mesh

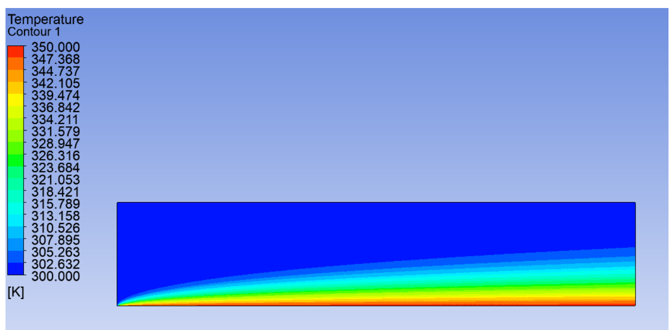

Figure 6: Flat Plate Thermal Boundary Layer

A2. Mesh

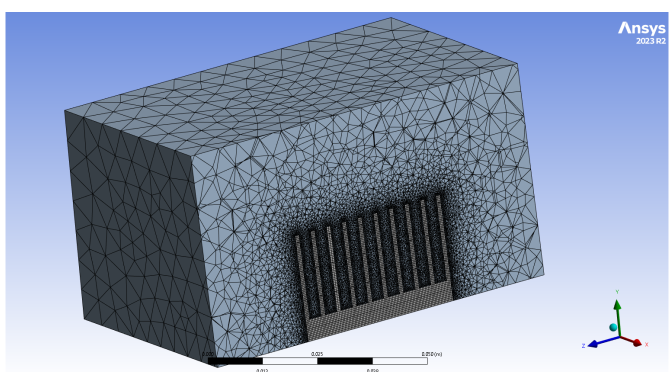

Figure 7: Mesh Image

A3. Heat Sink Results

Re 170

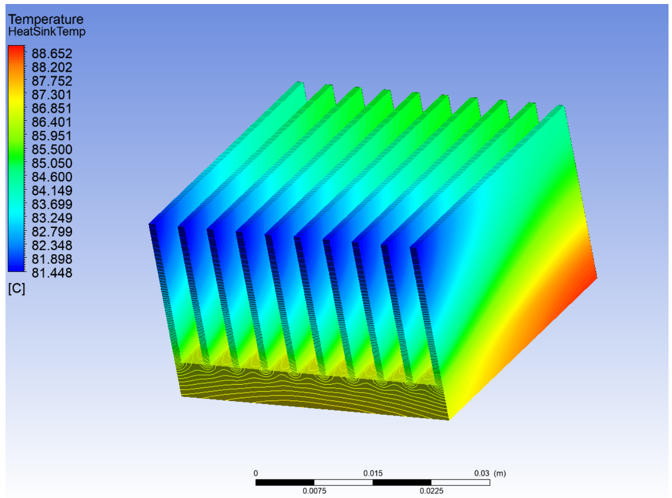

Figure 8: Heat Sink Temperature

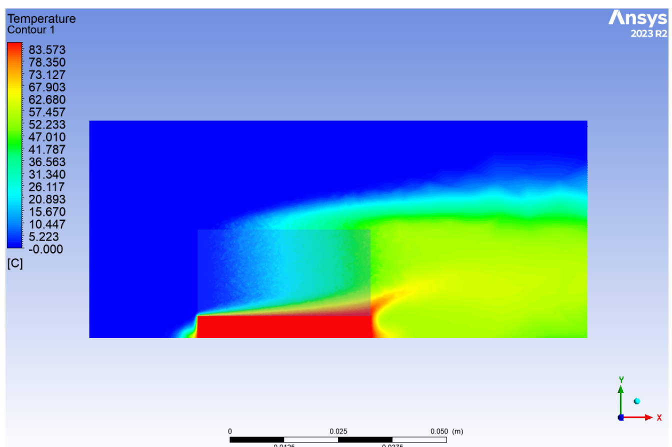

Figure 9: Midline Temperature

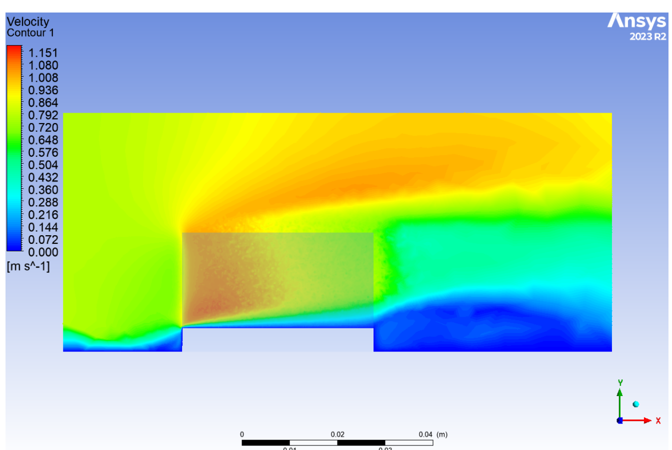

Figure 10: Midline Velocity

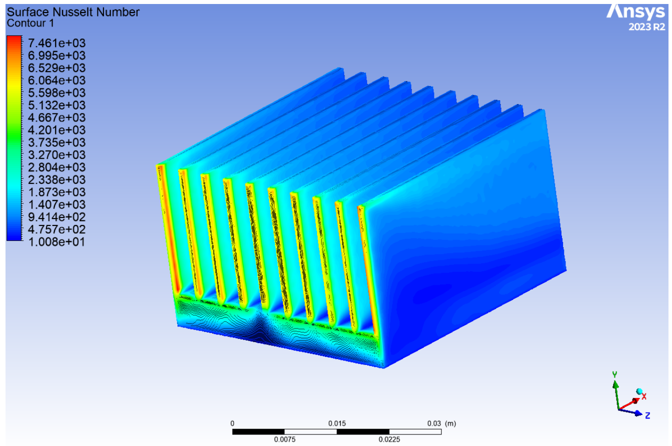

Figure 11: Local Nusselt Number

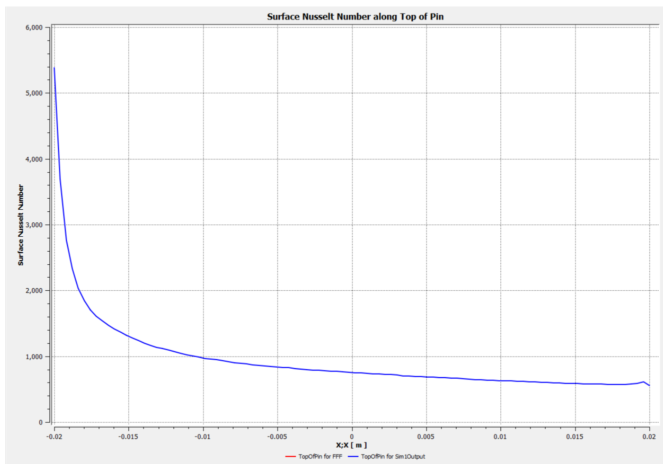

Figure 12: Nusselt Number at Top of Pin

Re 400

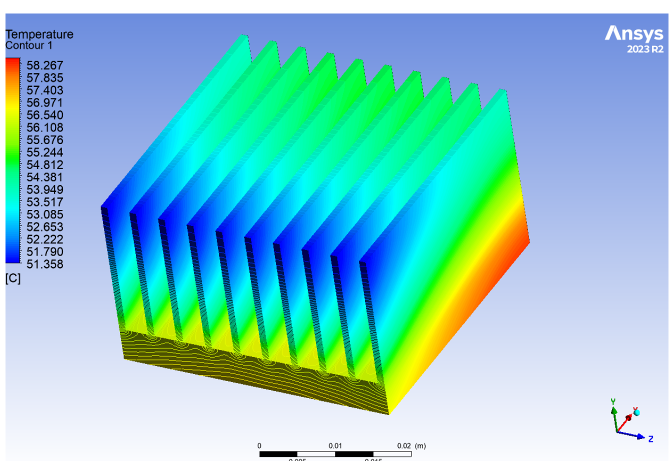

Figure 13: Heat Sink Temperature

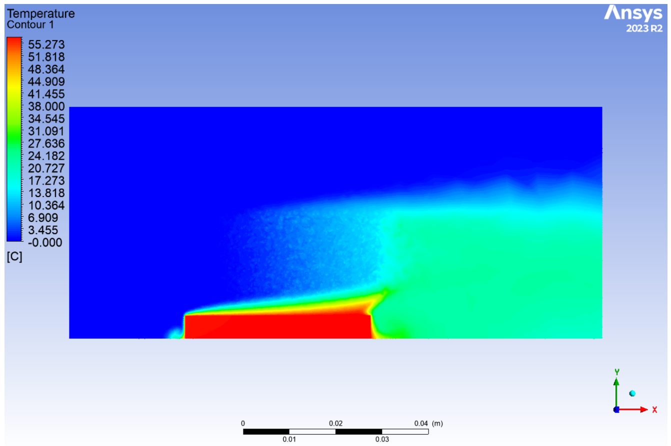

Figure 14: Midline Temperature

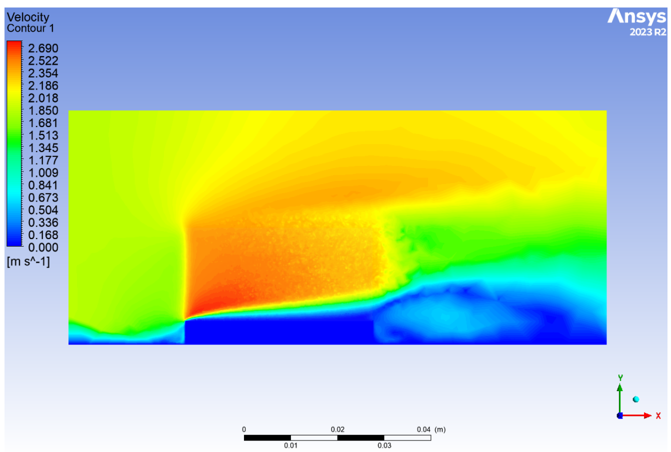

Figure 15: Midline Velocity

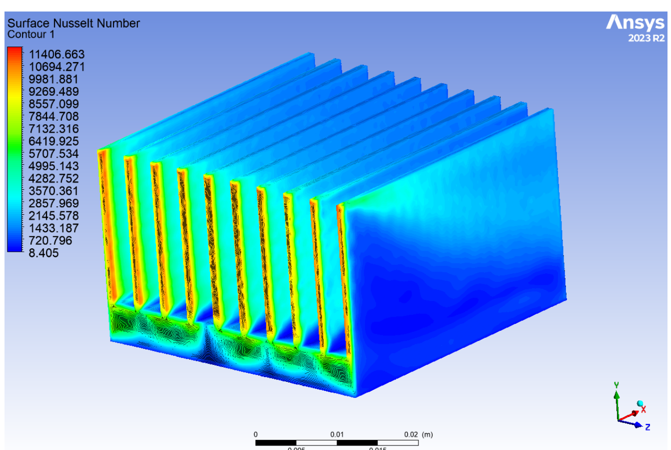

Figure 16: Local Nusselt Number

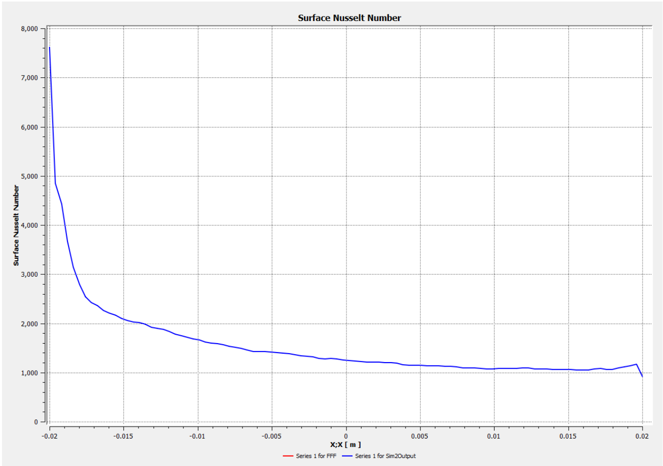

Figure 17: Nusselt Number at Top of Pin

Re 600

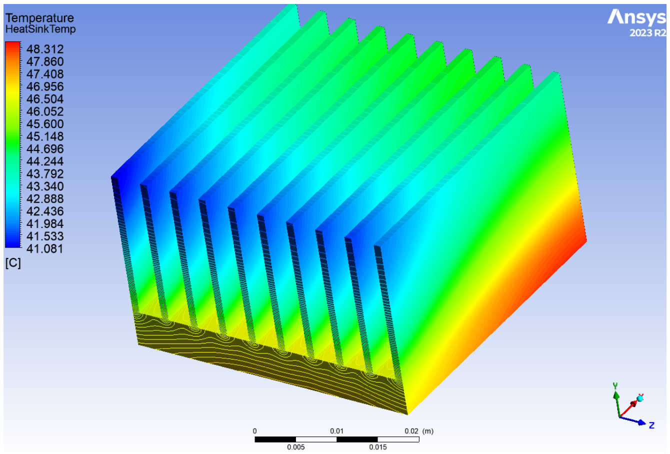

Figure 18: Heat Sink Temperature

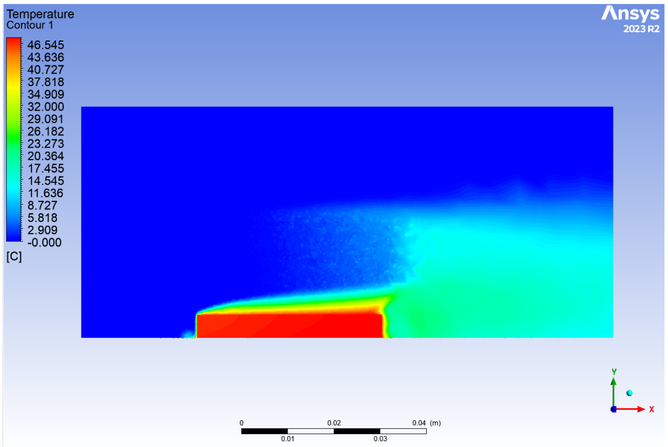

Figure 19: Midline Temperature

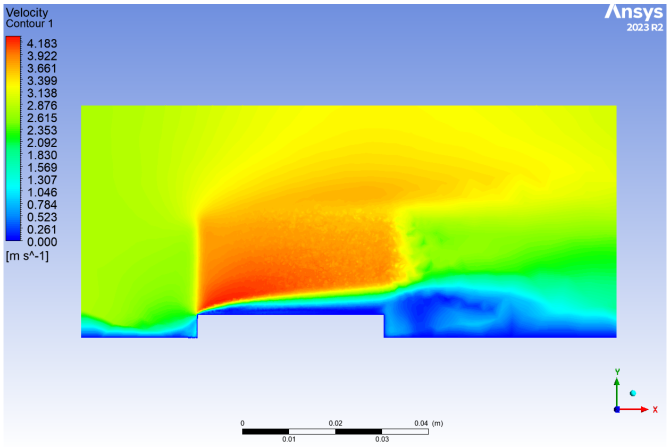

Figure 20: Midline Velocity

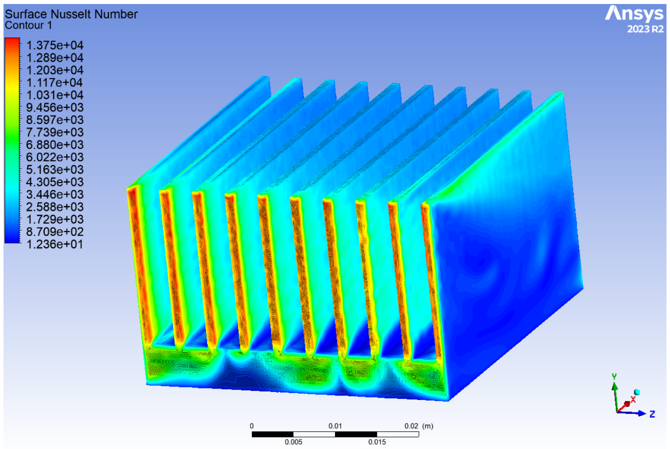

Figure 21: Local Nusselt Number

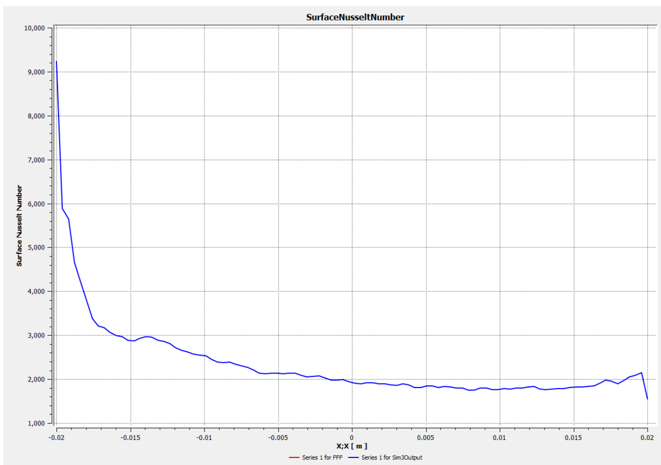

Figure 22: Nusselt Number at Top of Pin

Re 800

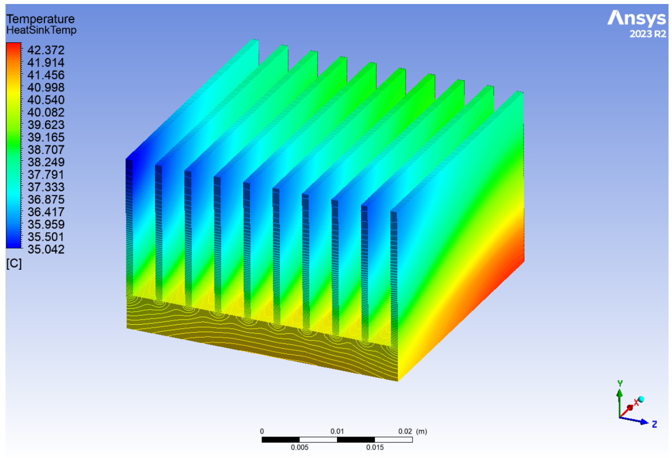

Figure 23: Heat Sink Temperature

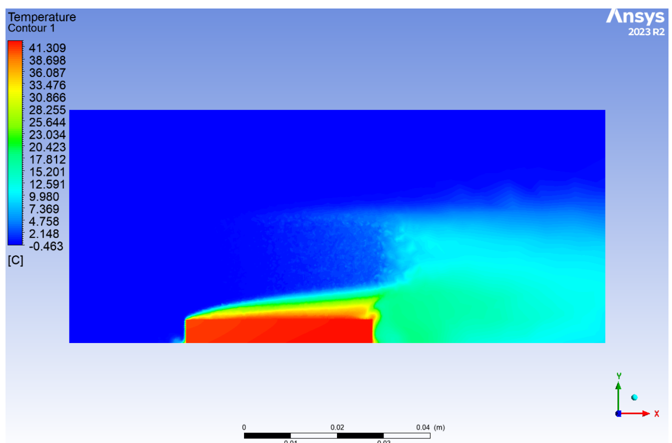

Figure 24: Midline Temperature

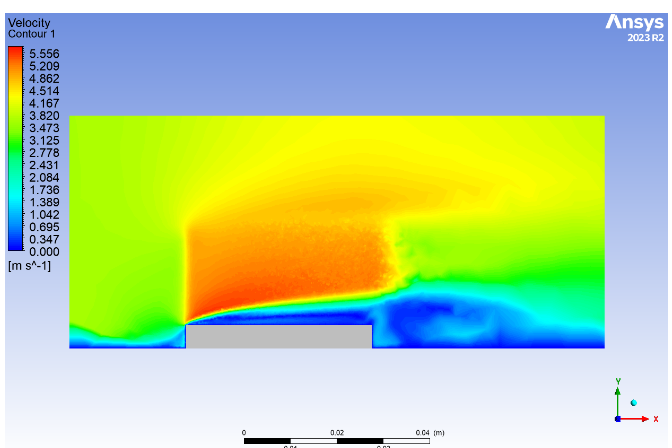

Figure 25: Midline Velocity

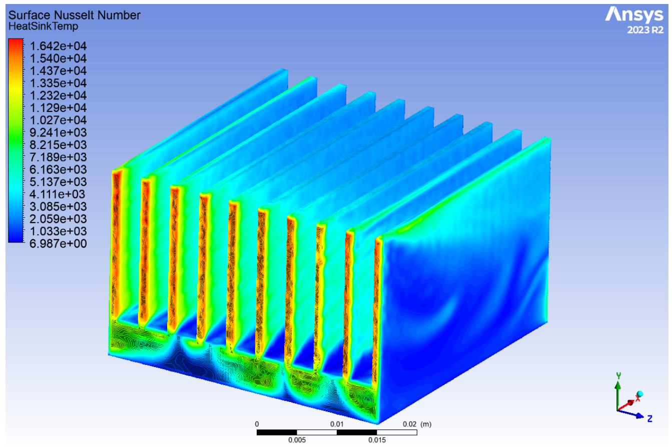

Figure 26: Local Nusselt Number

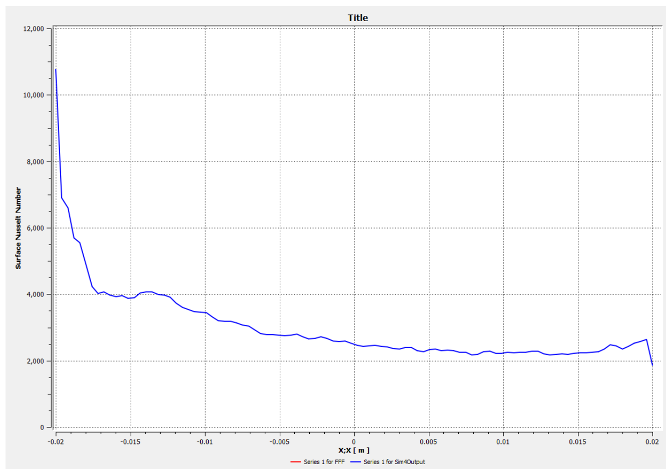

Figure 27: Nusselt Number at Top of Pin