# CPU COOLING SYSTEMS

Sarvesh Deshpande

November 2020

PART 1: AIR COOLING SYSTEM

## 1 Introduction

The CPUs of computers must be cooled to satisfy the maximum operating temperature limit while removing the heat dissipated by the CPU. Among all the available cooling methods, forced convection air cooling is the most common approach. In this direct heat removal approach, a fan is installed to a heat sink forming an assembly that is attached to the CPU. Air is forced through the heat sink by the fan, thus the heat is directly transferred to the final heat transfer medium, air.

Analyzing active heat sink CPU cooling to obtain the temperature distribution in the heat sink is a conjugate heat transfer problem involving all three modes of heat transfer: conduction in the heat sink, forced convection to chassis air, and radiation to the chassis walls and the other components. Eventually, heat is transferred to the ambient air outside the chassis.

The present work aims to analyze the selected CPU heat sink designs by using commercial CFD software packages and to improve them by using the results of the analyses. There are many parameters affecting the performance of a heat sink. The fin shape, the number of fins, the fin and base materials, and the base thickness are considered as the performance improvement paths for the selected heat sinks. For the sake of simplicity, only the CPU-heat sink-fan assembly is considered to be present inside the chassis.

## 2 CFD Simulation Approach

### 2.1 Computation Domain

The computer chassis is the computational domain. It is assumed to be made of steel with dimensions as follows:

$$H \times W \times D = 720mm \times 420mm \times 190mm \tag{1}$$

In real cases, there will be other components inside the chassis along with heat sink, but we are not considering them here.

### 2.2 The Heat Sink

A copper heat sink is considered for our CFD simulations. The heat is supplied to this heat sink through a square cross section (of approximate dimensions- 0.05mm x 0.05mm = 0.0025 mm2). A constant heat source of 100 W is assumed. So heat flux over the above mentioned cross section:

$$\text{Heat Flux} = \frac{100}{0.0025}\ W/m^2$$
$$\Rightarrow \text{Heat Flux} = 40000\ W/m^2$$

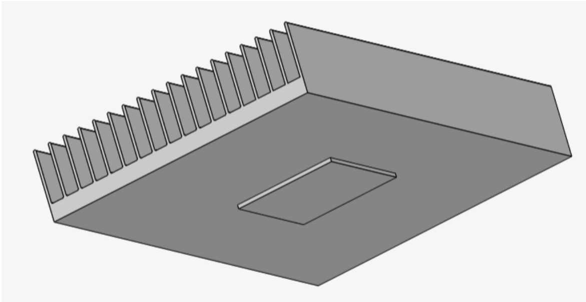


Figure 1: Model of heat sink used for simulation

## 2.3 Boundary Conditions

Since the Navier-Stokes equations are solved inside the domain, no-slip boundary condition is applied to all the walls in the domain. Therefore, at all of the surfaces, u = v = w = 0. The heat transfer mechanism at the chassis outer walls is assumed to be natural convection.

Furthermore, estimation of chassis wall temperature:

$$\text{Rayleigh Number } R_{a_L} = Gr_L.Pr = \frac{g\beta(T_s - T_\infty)L^3}{\nu\alpha}$$
$$\text{Applying corresponding values for air :} R_{a_L} = 3.862 \times 10^6$$
$$R_{a_L} < 10^9$$
$$\Rightarrow \text{Flow is leminar}$$

$$\text{Prandtl Number } Pr = \frac{\nu}{\alpha}$$
$$\Rightarrow Pr = 0.7283$$
$$\text{Nusselt Number } Nu_L = \frac{hL}{k}$$
$$\frac{hL}{k} = (0.069)(R_{a_L})^{1/3}(Pr)^{0.074}$$
$$\Rightarrow \ h = 1.5 \ W/m^2.K$$

By taking the ambient temperature, $T_{amb} = 30\ °C$, with the help of above obtained value, we can approximately calculate the temperature of chassis walls, $T = 36\ °C$

## 2.4 Simulation Results

Simulation is done on Simscale in accordance with above calculated values, and the resulting temperature distributions are shown below:
From CFD results:

$$T_{max} = 336.736\ K$$
$$T_{min} = 328.567\ K$$
$$T_{max} - T_{min} = 8.169\ K$$

Average rise above ambient temperature:

$$\Delta T = \frac{(T_{max} - T_{amb}) + (T_{min} - T_{amb})}{2}$$

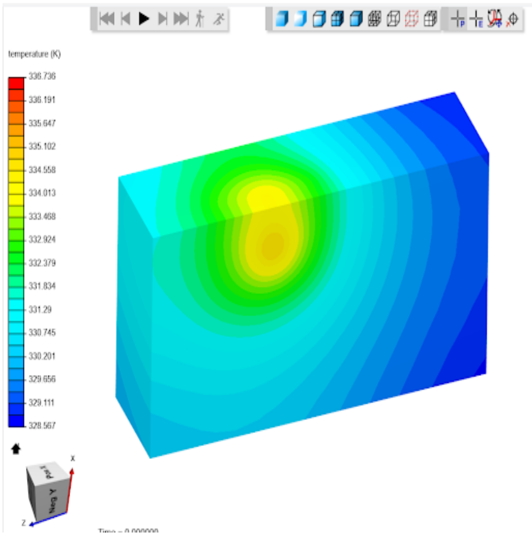


Figure 2: Temperature distribution over computer chassis

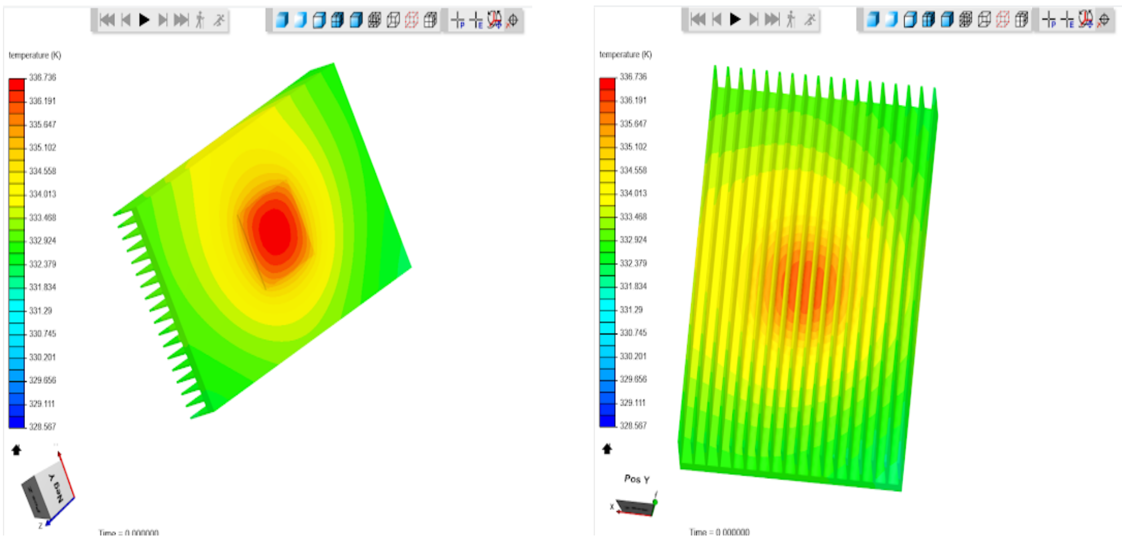


Figure 3: Temperature distribution over the heat sink

# 3 Discussion

$\Delta T$ can be considered as a parameter to analyze the performance of a heat sink. With the inclusion of a fan, temperature distribution will vary. For example the centre part of the heat sink will be relatively cooler due to its swirl. Further factors which can be manipulated to get a better performing heat sink are qualitatively discussed below:

## 3.1 To increase the Efficiency

Now we will look at some steps or modifications we can perform to help increase the efficiency of heat sinks.

Fins play the most important role in heat sinks so increasing their efficiency directly increases the efficiency of the heat sinks. Efficiency of a fin in such a heat sink is governed by the following formula

$$\eta = \frac{q_{real}}{q_{ideal}} = \frac{\sqrt{hPkA}\theta \times (\sinh(mL) + (h/mk)\cosh(mL))}{hPL\theta \times (\cosh(mL) + (h/mk)\sinh(mL))}$$

Heat sinks are designed using materials that have high thermal conductivity (k) such as aluminum alloys and copper.

It is also observed that densely stacked fins do not allow much air to cool the hottest center parts of the heat sink. So removing a few fins helps in providing a better flow route for the air which overshadows the reduction of the surface area for heat transfer due to the removal of those fins.

### 3.2 Machine Learning model to predict dependence of various parameters

While trying to analyze the heat distribution and efficiency of heat sink, many other factors like characteristics of the fan(e.g. rotation speed, flow rate, blade geometry), materials used for the chassis, for the base of heat sink, and material used for other components inside the chassis, etc also play a role in deciding the heat sink temperature distribution. Below is an attempt to build a Machine Learning model based on LASSO Regression to predict $T$.

- The model would be trained by an experimental dataset, which would contain the $\Delta T$ values corresponding to some specific values of the parameters discussed above. A brief implementation of the model through python is shown below.

```
from sklearn import linear_model
import pandas as pd
import numpy as np

#training data is assumed to be collected in a csv file with first N columns
#containing different parameter values and last column containing the values of ΔT

data_train = pd.read_csv('traindata.csv')

parameters = data_train.iloc[:, :-2].values
delta_T = data_train.iloc[:, -1.values

#training the model
clf = linear_model.Lasso(alpha=0.1)
clf.fit(parametrs, delta_T)

#prediction of ΔT based on the trained model
parameters_test = []

print(clf.coef_)
print(clf.predict(parameters_test))
```

Figure 4: Sample code for the ML Model

- Here parameters_test will assume the parameter values for which rise above ambient temperature ($\Delta T$) is to be predicted.
- clf_1.coef_ provides the coefficients (or weights) of different parameters.

PART 2: WATER COOLING SYSTEM

In water-cooled systems, internal liquid flow provides forced convection that can efficiently cool thermal systems. A radiator then rejects the heat remotely before the water is re-circulated to the heated components.

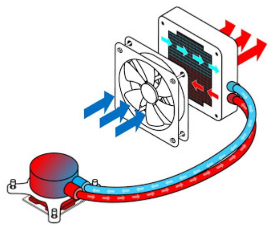


Figure 5: Brief model of a water cooling system

The advantages of using water cooling over air cooling include high specific heat capacity and thermal conductivity. These features allow water to transport heat over a larger distance at reduced flow rates and temperature differences.

# 1 Analysis of radiator tubes

Copper tubes are generally used inside radiators. These tubes carry the heated liquid and transfer heat to the radiator fins.

**Considering water as the liquid:**

For analysis of flow and temperature distribution inside these tubes, let's consider the liquid to be water and the material of the tube to be copper. Let the length of a single tube $L = 291\ mm$ and the diameter of its cross-section be $D = 10\ mm$

**Governing equations:**

Water flow inside the tube will be governed by the following equations

$$\text{x-momentum equation: } \frac{\mu}{r}\frac{d}{dr}(r\frac{du}{dr}) = \frac{dp}{dx}$$

$$\text{Energy equation in cylindrical coordinates: } u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial r} = \frac{\alpha}{r}\frac{\partial(r\frac{\partial T}{\partial r})}{\partial r} + \frac{\partial^2 T}{\partial x^2}$$

**Pressure gradient:**

There will be no pressure gradient along the radius of the tube. Along its axis:

$$\frac{dp}{dx} = -\frac{8\mu}{R^2}\ u_m$$
$$u_m = \frac{V}{2} \quad (at\ r = 0)$$
$$\text{Pressure gradient : } \frac{dp}{dx} = -\frac{16\mu V}{D^2}$$

After putting values, Pressure gradient across the tube $= 87.536 Pa/m$

**Non-Dimensional Temperature:**

$$\theta = \frac{T_w - T}{T_w - T_m}$$
$$\text{where, } T_w = \text{wall temperature}$$
$$T_m = \text{bulk mean temperature}$$

Let the temperature of the water inside the tube be 50 °C.
For water,
Density at 50 $°C$: $\rho = 998.05\ kg/m^3$
Prandtl Number: $Pr = 6.5241$
Coefficient of Viscosity: $\mu = 0.0005471 Pa.s$

Let the inlet velocity of water (due to pump used to pass it through the tube) $V = 1mm/s$

**Reynold's Number:**

$Re_D = \frac{\rho V D}{\mu}$

After putting values, $Re = 18.242$

**Nusselt Number:**

Considering constant wall temperature boundary condition (due to practical application) along with no-slip at the wall and zero temperature gradient at the axis. For a long tube, the Nusselt number comes out to be a constant (between 3.66 and 4.364). However for a finite tube of length L,

$$N_u = 3.66 + \frac{0.0668(D/L)Re_D.\,\mathrm{Pr}}{1 + 0.04[(D/L)Re_D.\,\mathrm{Pr}]^{2/3}}$$

After putting values, $Nu = 3.908$

Also, $T_m = T_w - (T_w - T)e^{-(4.L.N_u)/(D.Re.\,\mathrm{Pr})}$ $\quad$ $(T_w = wall\ temperature)$

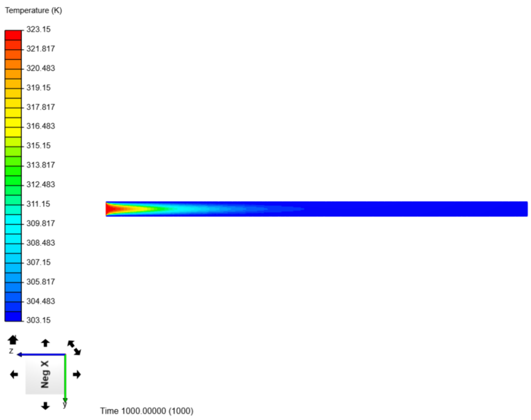
Hence, $T_m = 30.44\ °C = 303.59\ K$

NOTE: The same calculations can be done for different inlet velocities. Considering one more case for V = 0.01. We get the following results:

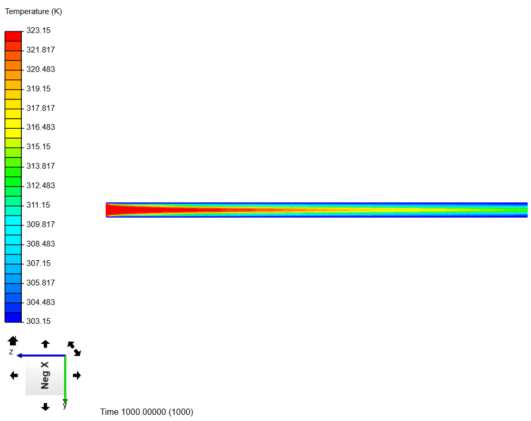
$$Re_D = 182.42 N_u = 4.2 T_m = 41.66\ °C = 314.82\ K$$

## 1.1 Conjugate Heat Transfer (CHT) Simulation

The above calculated results have also been analysed by cfd analysis of the same tube (SimScale). Following is the temperature distribution of the fluid for two different inlet velocities of 0.001 m/s and 0.01 m/s:




(a) for inlet velocity 0.001 m/s

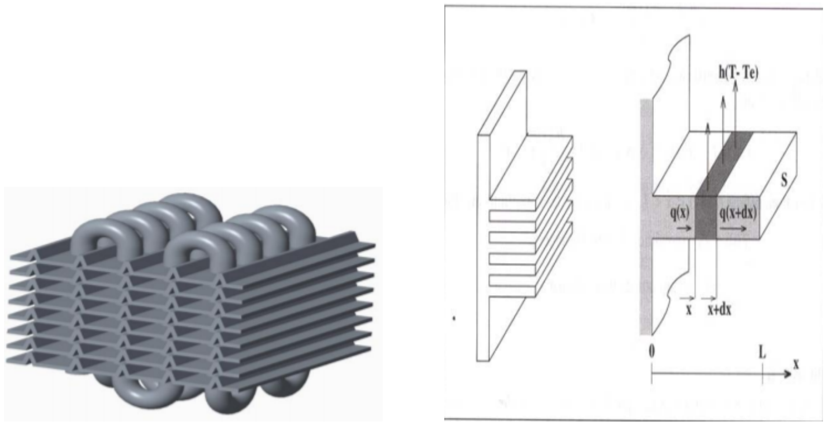



(b) for inlet velocity 0.01 m/s

Figure 6: Temperature distribution over the copper tube

## 2 Analysis of Radiator Fins

The heated water carrying tubes pass through a mesh of fins. Heat transfer due to convection through fins is responsible for heat dissipation. The water finally cools down and recirculates in the loop.




(a) Radiator setup (b) a single fin of the radiator

Figure 7: Standard radiator model

$T_b$ = Base temperature of fins = The constant temperature of walls of water carrying tubes
Therefore, $T_b = 30\ °C$

To get a mathematical idea of working of fins, let's consider a compact model with N fins attached to the same base. Assuming temperature distribution over the entire base remains uniform throughout (i.e. conductive resistance between tube and fin base is assumed to be low). Rate of heat transfer in a finite fin with convective tip:

$$q = \sqrt{hPkA_c}\ .\theta_o\ .(\frac{tanh(mL) + (h/mk)}{1 + (h/mk).tanh(mL)})$$

$$m = \sqrt{\frac{hP}{kA_c}}$$

L = length of fin
h = convective heat transfer coefficient
k = thermal conductance
$A_c$ = area of cross section
$\theta_o = T_b - T_a$, $T_a$ = ambient temperature

Let the efficiency of a single din be $\eta_f$:

$$\eta_f = \frac{q}{q_{ideal}} = \frac{q}{h(PL)\theta_o}$$

$$\Rightarrow \eta_f = \frac{1}{mL}\ .(\frac{tanh(mL) + (h/mk)}{1 + (h/mk).tanh(mL)})$$

**So the overall efficiency of N fins with same base:**

$$\eta = 1 - \frac{NA_f}{A_t}(1 - \eta_f)$$

Likewise, efficiency of the whole setup can be calculated.

# 3 Discussion

- $T_m$ values calculated numerically can also be computed with the help of CHT simulation.

| Velocity(m/s) | $T_m$(from numerical calculation) | $T_m$(from CHT) |
|---|---|---|
| 0.001 | 303.59 K | 305.15 K |
| 0.01 | 314.82 K | 314.48 K |

- To maintain a laminar flow within the tube, velocity water and diameter of the tube can be manipulated to get better results.
- A defined thermal boundary layer can be seen in the results of simulations.
- Boundary condition of constant wall temperature is chosen (instead of constant wall heat flux) since it is practically more feasible to attain.
- After getting the Nusselt number, the convective heat transfer coefficient of water can be calculated.

$$h = Nu(k/D)$$

- The structure of fins can be manipulated in order to get better efficiency.

# References


[1] Cpu cooler: Liquid cooling vs air cooling.

[2] Dr. Y. Krishna Dr. V .V. Subba Rao T.Hari Krishna D.Bhanuchandrarao, M.Ashok chakravarthy. Cfd analysis and performance of parallel and counter flow in concentric tube heat exchangers. *International Journal of Engineering Research Technology (IJERT)*, Vol. 2(Issue 11), November - 2013.

[3] *Aree Ali Mhammad and **Karwan Hama Faraj. The cooling of computer radiator system by numerical simulation. *(KAJ) Kurdistan Academicians Journal*, 4(1) part A:159–164, March 2006.

[4] Emre Ozturk and Ilker Tari. Forced air cooling of cpus with heat sinks: a numerical study. *IEEE Transactions on Components and Packaging Technologies*, Vol. 31(No. 3):pp. 650–660, (2008).

[5] Gaikwad Mahesh Shamrao Prasannakeerti Prakash Magadum. Analysis optimization of processor cooling fins. *International Journal of Engineering and Innovative Technology (IJEIT)*, Volume 5(Issue 4), October 2015.