Heat transfer in conduction Report on Heat Sink Design

Yang Yang[1], Yuchen Yang[2], Minmin Yu[2], Zhijie Wu[2], Xinyu Zhu[2]
1. China Ship Science Research Center, Wuxi, Jiangsu 214000
2.Taihu Laboratory of Deepsea Technological Science, Wuxi, Jiangsu 214000

Abstract: The purpose of the project is to design a heat sink with limited information given and make sure it reaches certain requirements. Design and optimization process will be done in the beginning, 2-D analytical, and 2-D numerical solution will be generated and used to check the result. Also, a flow simulation will be made by using SOLIDWORKS. In the end, result will be compared, different between each result will be analyzed.

Keyword: Heat transfer; Heat Sink; Simulation; Solid works

1. TNTERDUCTION

The goal of this project is to design two heat sinks that will be work on a 1-inch by 1-inch chip under forced convection (FC) and natural convection (NC). Only boundary condition given is air speed equal to 1 m/s under FC. In the end, both heat sink design should have a maximum source to air thermal resistance of 1 ℃/W. Final design should optimize on weight, size and pressure drop.

2. Design and Optimization

Design and optimization process were done by using spreadsheet optimization tool, to star the design, some factors need to be considered first. Then, in order to optimize for thermal resistance, size, weight, pressure drop, we need to define the geometries of the heat sink. In the end, to have a simple and clear view of the result. Overall material cost and all related resistance will be used to compare with other alternatives. We also consulted to a PDF from Simon Fraser University, and other source, but since Rfh in NC can be neglect, we here by choose the solution in Grankoplis' book and ignored the calculation of resistance from heating fluid.

General Property Used

Fluid Approach velocity: 1 m/s

Material chosen: Pure aluminum

Aluminum conductivity: 237 W/m*K

Ambient temperature: 25 ℃

Air conductivity at 25℃: 0.02551 W/m*K

Air density at 25℃: 1.184 kg/m^3

Air dynamic viscosity at 25℃: 1.849*10^-5 kg/m*s

Assume source maximum temperature: 100 ℃

3. FC Geometry & NC Geometry

Boundary condition: Forced Convection with air speed 1m/s.

Variable	Definition	Unit	Value
L	Fin Height	m	0.0370
W	Heat Sink Width	m	0.0355
D	Fin Depth	m	0.0370
t fin	Fin Thickness	m	0.0005
Wch	Channel Width	m	0.0020
n	Number of Fins	-	15.0000

Boundary condition: Natural Convection without any air flow.

Variable	Definition	Unit	Value
L	Fin Height	m	0.0620
W	Heat Sink Width	m	0.0610
D	Fin Depth	m	0.0650
t fin	Fin Thickness	m	0.0015
Wch	Channel Width	m	0.0020
n	Number of Fins	-	18.0000

4. 2D Analytical Result for FC

As we neglect the heat transfer end by convection, we remain with

$$\frac{\theta}{\theta_b} = \frac{T(x) - T_\infty}{T_b - T_\infty} = \frac{\cosh[m(L-x)]}{\cosh(mL)}$$

X	T(x) (Kelvin)	X	T(x) (Kelvin)
0.002	333	0.009	327.59
0.003	329.82	0.012	326.54
0.006	328.69	0.015	325.52

By considering the different values of x, we can find the temperature distribution in the fins. Following table contains the considerations for different points across the fin length.

5. 2D Analytical Result for NC

By considering the different values of x, we can find the temperature distribution in the fins. Following table contains the considerations for different points across the fin length.

X	T(x) (Kelvin)
0	333
0.005	332.98
0.010	331.96
0.015	330.95
0.020	329.94
0.025	328.94

6. 2D Numerical Solution

2D numerical solutions are obtained by using optimum geometry parameters obtained during general design section in finite difference equation developed from Taylor's Series expansion. Temperature of the internal points:

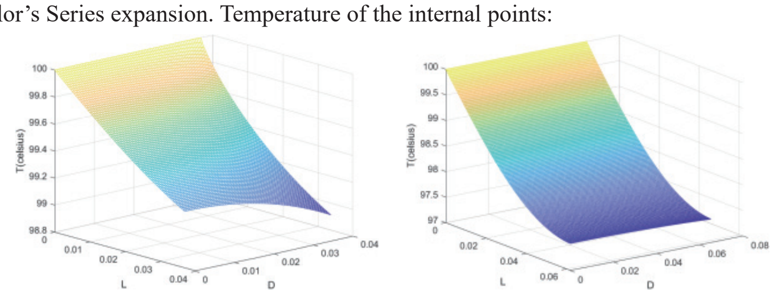

Figure 1. Numerical FC and NC heat sink temperature distribute on fin.
For FC: Q=7.6047 W/m^2; R all_fin = 0.6575 °C/W
For NC: Q=4.4011 W/m^2; R all_fin = 0.9467 °C/W

7. FC CFD Simulation Results

After the calculation is finished, we can call up forms and graphs to check the results according to the needs. As shown in figures.

The temperature distribution of the fluid flowing through the plane of the heat sink. We can also see the temperature distribution of the fluid in the side profile. As shown in figure 2.

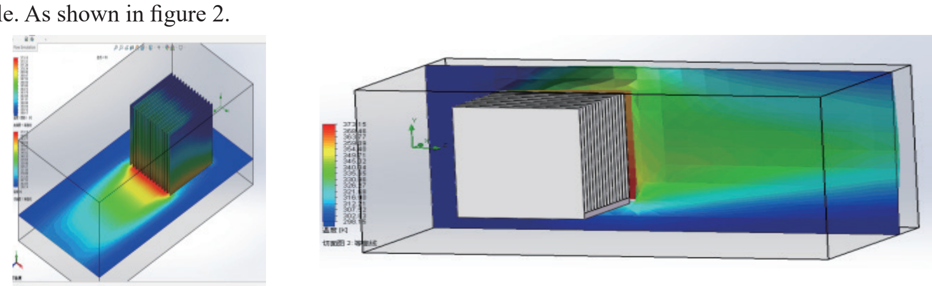

Figure 2. The temperature distribution through the plane of the heat sink and the side profile.

Because in this simulation case, the air fluid has a velocity of 1 m/s in the positive direction of the z-axis, we can load the flow trajectory simulation of the fluid. At the same time, the temperature distribution can be loaded on the simulated fluid flow trajectory. We can also just simulate the heat flux distribution of the heat sink by software, as shown in the figure 3.

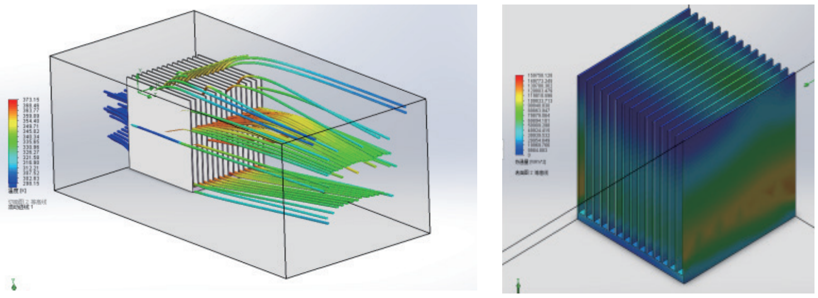

Figure 3. Load temperature distribution on the simulated fluid flow trajectory. And the heat flux distribution of the heat sink.

At the same time, we can get a graph of various parameters of each side of the heat sink, such as the curve of heat flux, And the temperature distribution curve of each side of the heat sink, such as figure 4.

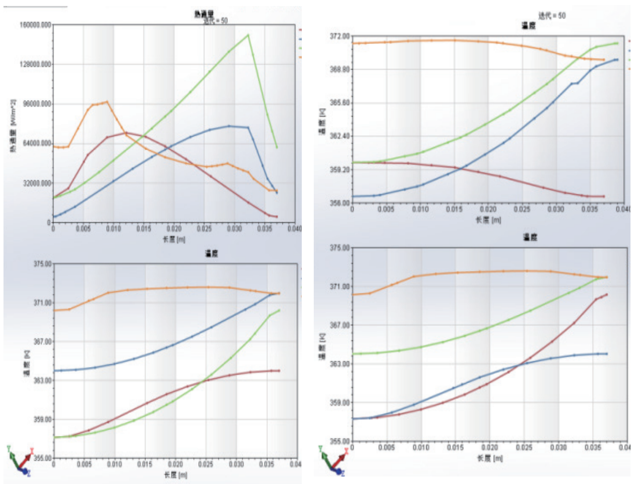

Figure 4. The curve of heat flux and temperature of each side

8. NC CFD Simulation Results

Since the fluid is stationary, the result of heat dissipation is manifested in a form that diffuses from the center to the periphery, shown in the figures 5.

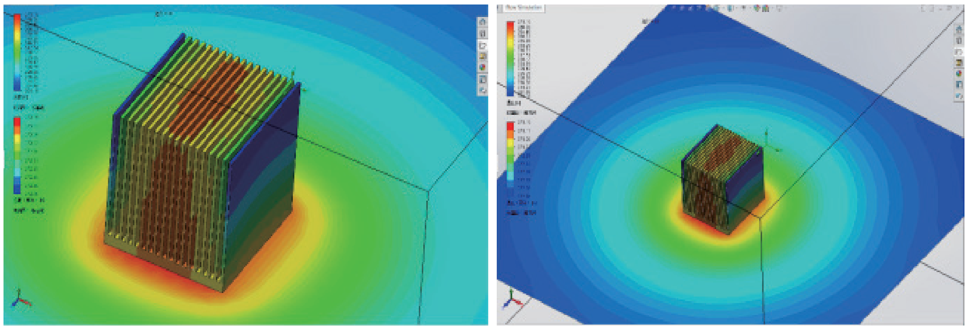

Figure 5. The temperature distribution of the obtained fins and the temperature distribution of the fluid passing through the plane of the bottom surface of the fins

The temperature distribution of the fluid flowing through the plane in the middle of the fin is shown in the figures 6.

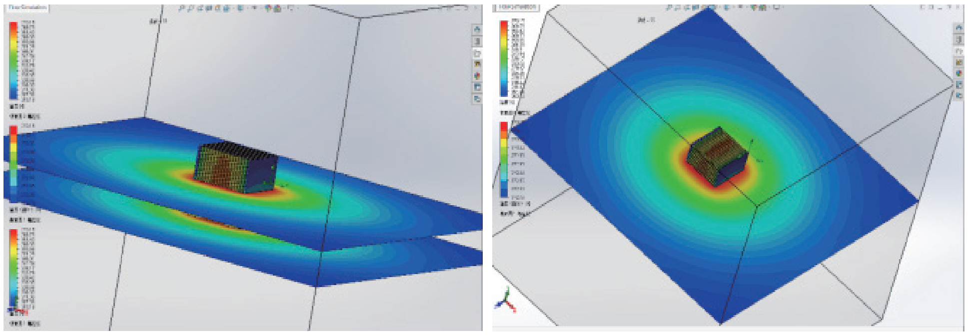

Figure 6. The temperature distribution of the fluid flowing through the plane of the heat sink

From the side profile, the temperature distribution of the fluid is shown in the figure 7.

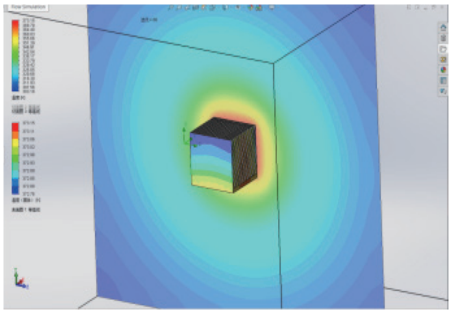

Figure 7 The temperature distribution of the fluid in the side profile

Since the velocity of the fluid air is zero in this simulation, no flow trajectory map can be given. But if we need a three-dimensional temperature distribution to better perform the next data analysis, we can also load a 3D temperature distribution of the heat sink, as shown in the figure 8.

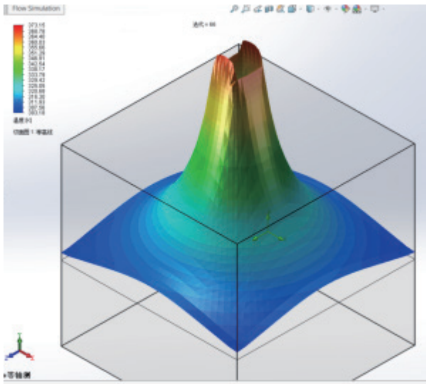

Figure 8 3D temperature distribution of the heat sink

At the same time, we can get a graph of the heat flux of each side of the heat sink, There is also a temperature distribution curve of each side of the heat sink, as shown in the figure 9.

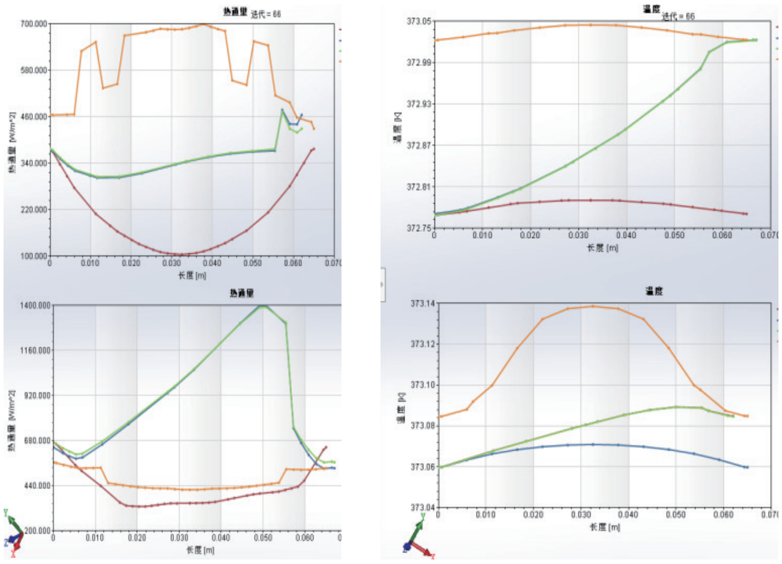

Figure 9 The curve of heat flux and temperature of each side

9. Conclusion and Recommendation

After comparing the result of our solutions, it comes really clear that we can separate the solutions into two categories. First one would be the 2D solutions lead by the spreadsheet, sure thing the temperature distribution map and thermal resistance of each solution is going to be slightly different than each other but they still fall in a close area. However, as you can see, the CFD solution offers a much different result by provided big temperature distribution different than previous solutions. Temperature difference (or we can say – the ability to dissipate heat) on each fin plate of the heat sink is different than each other, the ones in the middle is much hotter than the once on the side.

Due to CFD's ability on examine the problem globally, it will for sure consider more issues and add more ways the heat that can be dissipated out of the heat sink. In previous solution, the heat sink was built by satisfying over all thermal resistance smaller than 1 ℃/W, during a lot of the situation, only fin resistance was considered, this will of course lead to a heat sink that perform better in real life. After CFD calculation, the heat transfer coefficient for the force convection is 73.627W/m^2*K which is significantly higher than calculated number 49.51W/m^2*K. In natural convection the heat transfer coefficient is 7.344W/m^2*K verses calculated 7.423W/m^2*K.

From this simple comparison, we can guess wildly that problem is from the fluid, the way to calculate thermal resistance from heating fluid is the key to refine the calculation but unfortunately Solid works do not explain the results very carefully, not to mention how the calculations were done.

Conclusion is CFD solution will provide more realistic result but if we are using only fin resistance to design the heat sink, it will sure satisfy the design requirement, only down side about it is the design will not be optimum.

References

[1] "Majid Bahrami - Teaching - ENSC 388." https://www.sfu.ca/~mbahrami/ensc388.html (accessed Mar. 07, 2023).

[2] "(PDF) Transport Processes and Separation Process Principles Fifth Edition CHRISTIE JOHN GEANKOPLIS A. ALLEN HERSEL DANIEL H. LEPEK | Mahmoud Abumazin - Academia.edu." https://www.academia.edu/49073469/Transport_Processes_and_Separation_Process_Principles_Fifth_Edition_CHRISTIE_JOHN_GEANKOPLIS_A_ALLEN_HERSEL_DANIEL_H_LEPEK (accessed Mar. 07, 2023).

[3] "Staedy Conduction Heat Transfer.pdf." Accessed: Mar. 07, 2023. [Online]. Available: https://www.sfu.ca/~mbahrami/ENSC%20388/Notes/Staedy%20Conduction%20Heat%20Transfer.pdf

[4] "Scrap Metal Prices in Canada." https://www.priceofscrapmetals.com/canada/ (accessed Mar. 07, 2023).

About the author: Yang Yang (1990 --), male, Han nationality, born in Nanyang City, Henan Province, Grade 1 engineer, Master of Engineering, research direction of Marine equipment machinery, hydraulic direction