Power Division

Simulation of Single-Phase Laser Diode Cooler

White Paper

Prepared By:
Michael Seibert, Ph.D.

November 2019

UNITED STATES ARMY FUTURES COMMAND (AFC), COMBAT CAPABILITIES DEVELOPMENT COMMAND (CCDC) C5ISR CENTER, COMMAND, POWER, & INTEGRATION (CP&I) DIRECTORATE, POWER DIVISION, 6631 CYBER ST, BUILDING 5100, ABERDEEN PROVING GROUND, MD 21005-1852

This page is intentionally left blank.

CONTENTS

REFERENCES:

1. Datta, M., Choi, H., "Microheat exchanger for cooling high power laser diodes," Applied Thermal Engineering 90 (2015) 266-273

INTRODUCTION

High powered laser weapons require high performance thermal management. This report examines the heat transfer and pressure drop characteristics of a water cooled laser diode array. It begins with a plane channel and examines how changes in geometry and flow affect the heat transfer and pressure drop performance of the cooler. The simulations show the capabilities of computational fluid dynamics (CFD) in single phase flow and the performance of water as a coolant (60 W/cm^2).

PURPOSE

The purpose of these simulations is to perform a baseline thermal performance characterization of water cooling laser diodes.

SIMULATION SETUP

An array of 12 laser diodes as fabricated by Datta and Choi [1] is considered. The pressure drop and heat transfer characteristics are simulated using ANSYS FLUENT. Diodes are 20 mm x 9 mm with 110 W heat load each. Diodes are mounted to a ceramic plate 1 mm apart. The overall plate dimensions are 160 mm x 50 mm. Water flows through an 11 mm channel matching the dimensions of the plate. Inlet and outlet tubes are 3/8" OD. The heat exchanger fabricated by Datta included complex channels within this volume. Initial simulations assess performance with a simpler empty channel geometry. Flow rate and channel thickness are varied. Pressure drop and maximum temperatures are evaluated.

The initial mesh is shown in Figure 1 with 1,918,028 cells. Meshing the copper blocks rather than set the diodes as a boundary condition on the plate adds 19,488 total cells, ~1% of total. The ceramic aluminum nitride layer is assigned properties of aluminum since the AlN properties were not available.

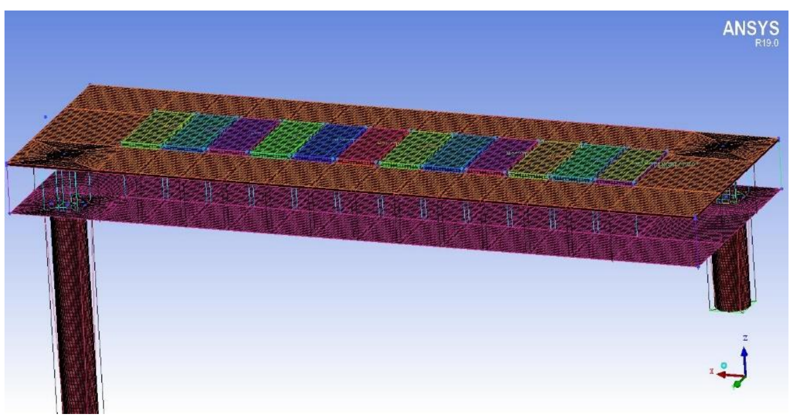

Figure 1. Computational mesh. 12 diode mounting blocks are visible on the top. The longer tube to the left is the water inlet and the water outlet is on the right.

Heat flux on the diodes is 110W/(9mmx20mm)=611,111 W/m^2 or 61.1 W/cm^2. This heat flux is lower than what researchers are targeting with two-phase micro heat sinks. However, as shown below, it is aggressive for single phase cooling. The challenge is not the heat flux as much as the heat flux with a small temperature difference.

Datta's requirements set max water flow rate at 8 L/min (0.133 kg/s) [1]. This was used for initial simulations with the baseline geometry. The inlet temperature was set to 27°C, a common ambient temperature default.

SIMULATION RESULTS

The temperature profile of the ceramic layer is shown in Figure 2. Diodes 1-8 were near 100°C at the ceramic bottom. Diodes 9-12, closer to the inlet were near 67°C. These temperatures are high for laser diodes and the variation across the array is unacceptable.

The outlet temperature, 29.4°C, is purely a function of mass flow rate and heat input from the diodes. It is independent of geometry configuration. The 2.4 °C temperature rise shows the mass flow rate is high enough to consider the water constant temperature for control purposes. I.e. temperature differences in the ceramic are not due to the water heating up.

Inlet pressure 7.86 kPa (1.14 psi) (Datta specified <8 psi, allowing modification for flow enhancement). A sharp pressure change was visible at the beginning of the outlet tube due to the sudden restriction in flow cross section.

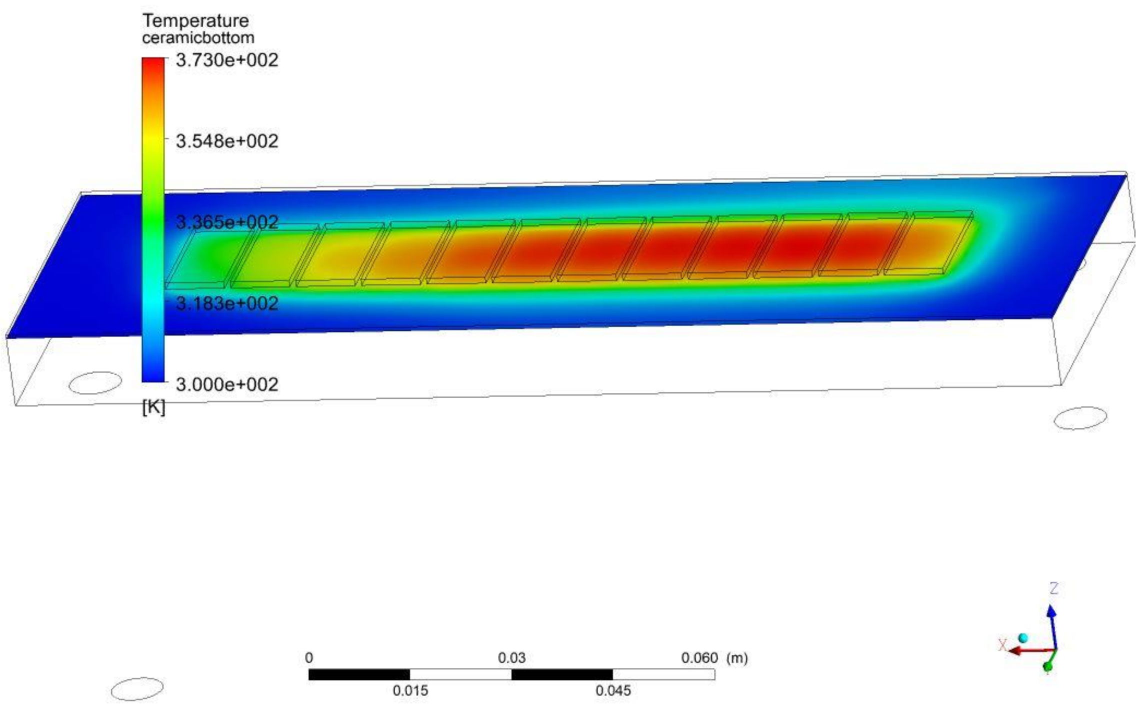

Figure 2. Temperature profile on the ceramic layer adjacent to the coolant. Inlet tube is on the left, outlet on the right

Geometry Modification and Discussion

The next step reduced the channel thickness. Cases with channel thickness 2 mm and 1 mm did not converge in steady state. Simulations with geometry modifications were based on the 5 mm thickness to make use of steady state simulations which are faster. It is possible that better convergence could be achieved using initial conditions more closely aligned to the expected flow, particularly the inlet and outlet tube flow directions.

Streamline plots of the 5 mm channel in Figure 3 show highest water velocity is not under the diodes. Water exiting the inlet tube spreads in 360° until it encounters an obstruction. Nothing directs flow directly toward the outlet or under the diodes. After encountering the wall, water follows the wall. 65% of the flow is outside the diodes. Low velocity under the diodes produces lower heat transfer. Aligning inlet and outlet with the diodes did not significantly change the diode temperature. Flow from the inlet tube hits the top wall and spreads in all directions. When it hits a side wall, it follows that wall, leading to higher flow velocities along the edges than in the center under the diodes. 0.7 m/s near edge compared to 0.5 m/s under the diodes.

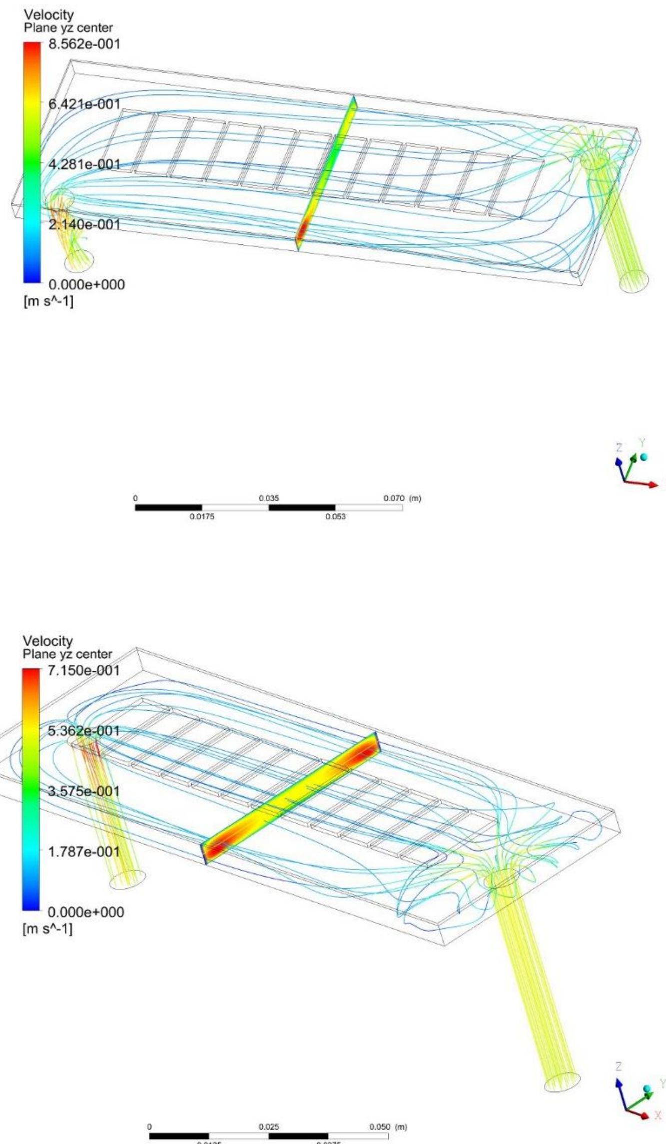

Figure 3 a&b. Streamlines of 5 mm thick channel with inlet and outlet in a) original locations and b) in line with the diodes. YZ plane in the center is colored by velocity, again showing high velocity is not under the diodes.

The temperature of the spreading plate is much lower in this outer region than under the diodes. It is not clear that the extra width is helping spread the heat. The next steps seek to focus the flow under the diodes. First, baffles are inserted after the second diode. The baffles are the full height of the channel and extend from the side walls to the edge of the diodes. They focus the flow through the middle of the heat exchanger while maintaining the spreader plate for analysis. More flow is directed under the diodes, reducing temperature. Four diodes downstream from the baffles, flow expands through the width of the channel increasing temperature for the last four diodes. The outer temperature doesn't rise substantially when the flow rate adjacent to it is reduced, showing it is not necessary.

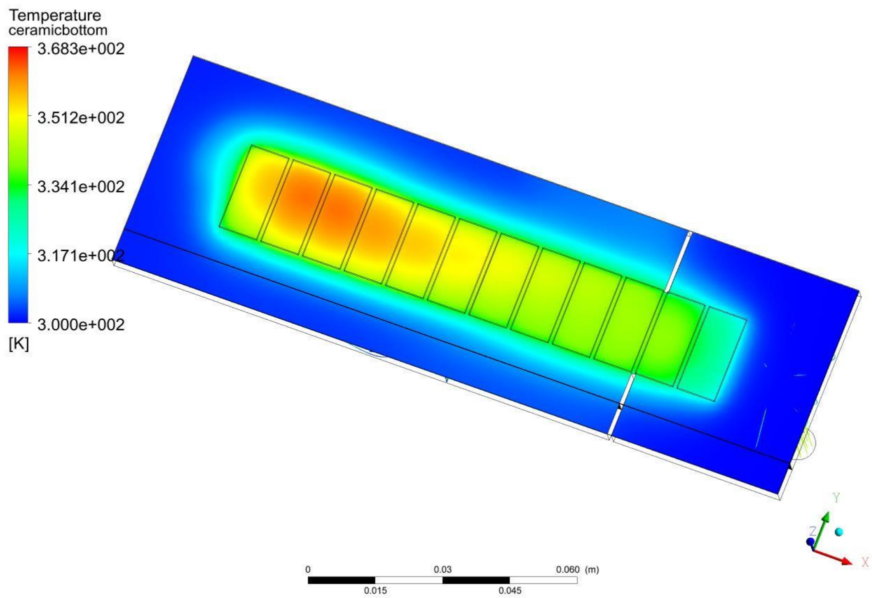

Figure 4. Temperature profile with two baffles. Baffles can be seen between the 2nd and 3rd diode from the right.

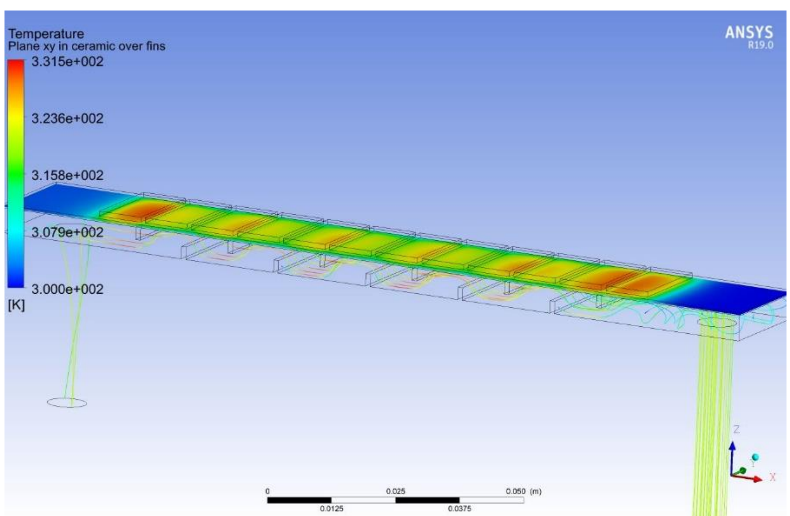

Figure 5. Temperature profile with width reduced and upper and lower fins.

Figure 5 shows the heat exchanger width reduced with alternating top and bottom fins. The upper fins are conductive, increasing the surface area for heat rejection. The lower fins are not conductive, but serve to direct flow to impinge the ceramic. The lowest ceramic temperatures are above the lower fins where the water impinges. The highest temperatures are in the wake of the upper fins where flow is directed away from the surface. This configuration reduces the maximum temperature to 58°C with pressure drop of 66 kPa (9.6 psi). This pressure drop is higher than the limit specified by Datta[1], but the results demonstrate possibilities with geometry changes.

These simulations demonstrate how 60 W/cm^2 can be accomplished using liquid coolant with geometric flow enhancements. The importance of fins or other obstructions to direct the flow to the hot surface is demonstrated. The low temperature change (2.4°C) in the cooling water indicates that higher heat flux will require novel geometry in addition to phase change.

OBSERVATIONS & CONCLUSION

Summary of Simulation Results:

	11 mm Channel	**5 mm Channel**	**5 mm Channel with Top and Bottom Fins**
Max Ceramic Temp	101°C	97 °C	58°C
Velocity Under Diodes	0.18 m/s	0.45 m/s	2.83 m/s
Convection Coefficient	0.96 W/cm^2-°C	1.01 W/cm^2-°C	2.15 W/cm^2-°C

Table 1 – Summary of performance characteristics for geometric configurations.

Overall Conclusions and Takeaways:

- Spreading the heat is less effective when the increased area reduces fluid velocity and thus heat transfer coefficient
- High flow water is effectively constant temperature.
- Flow across a flat plate has lower convection coefficient than flow disturbed by fins.
- Fluid goes where it is directed/guided.
- Constant mass flow rate gives heat transfer not affected by reduced flow area due to ~constant Reynolds number (within range of this study).

These simulations lay the groundwork for future analysis and design for laser diode cooling simulations and experiments. They set expectations for performance and provide an initial starting point for design. Next steps include further geometry enhancements and implementing boiling heat transfer in both simulation and experiments.