Computational Fluid Dynamics Study of an Atmospheric Water Generator

TEXAS STATE UNIVERSITY

The rising STAR of Texas

Deola Aina, Professor Mark Summers, Dr. Bahram Asiabanpour
Ingram School of Engineering, Texas State University

Introduction

The goal of this project is to use Computational Fluid Dynamics (CFD) to study an Atmospheric Water Generator (AWG). The purpose of this is to aid the development of an optimal AWG configuration. The AWG utilizes a thermoelectric cooler (or Peltier) which is a device that when powered, transfers heat from one surface of the device to the other. This results in a very hot side and a very cold side. Under the right atmospheric conditions (dry bulb temperature and humidity), the cooling of the Peltier device allows humid air to be converted into moisture.

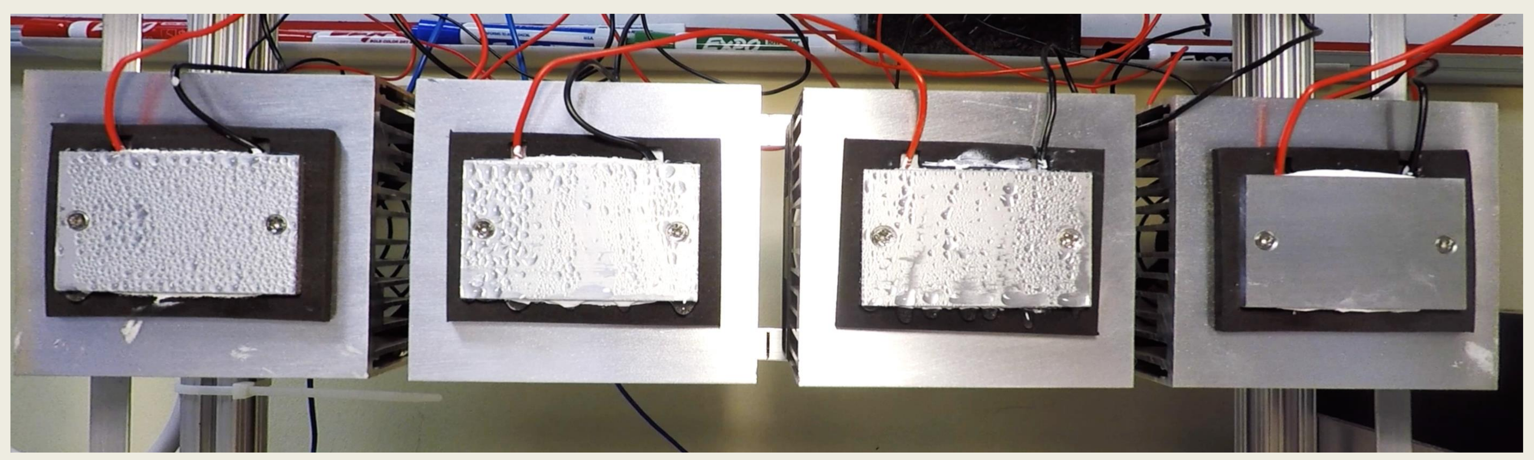

Figure 1: Assembly of AWG units being tested

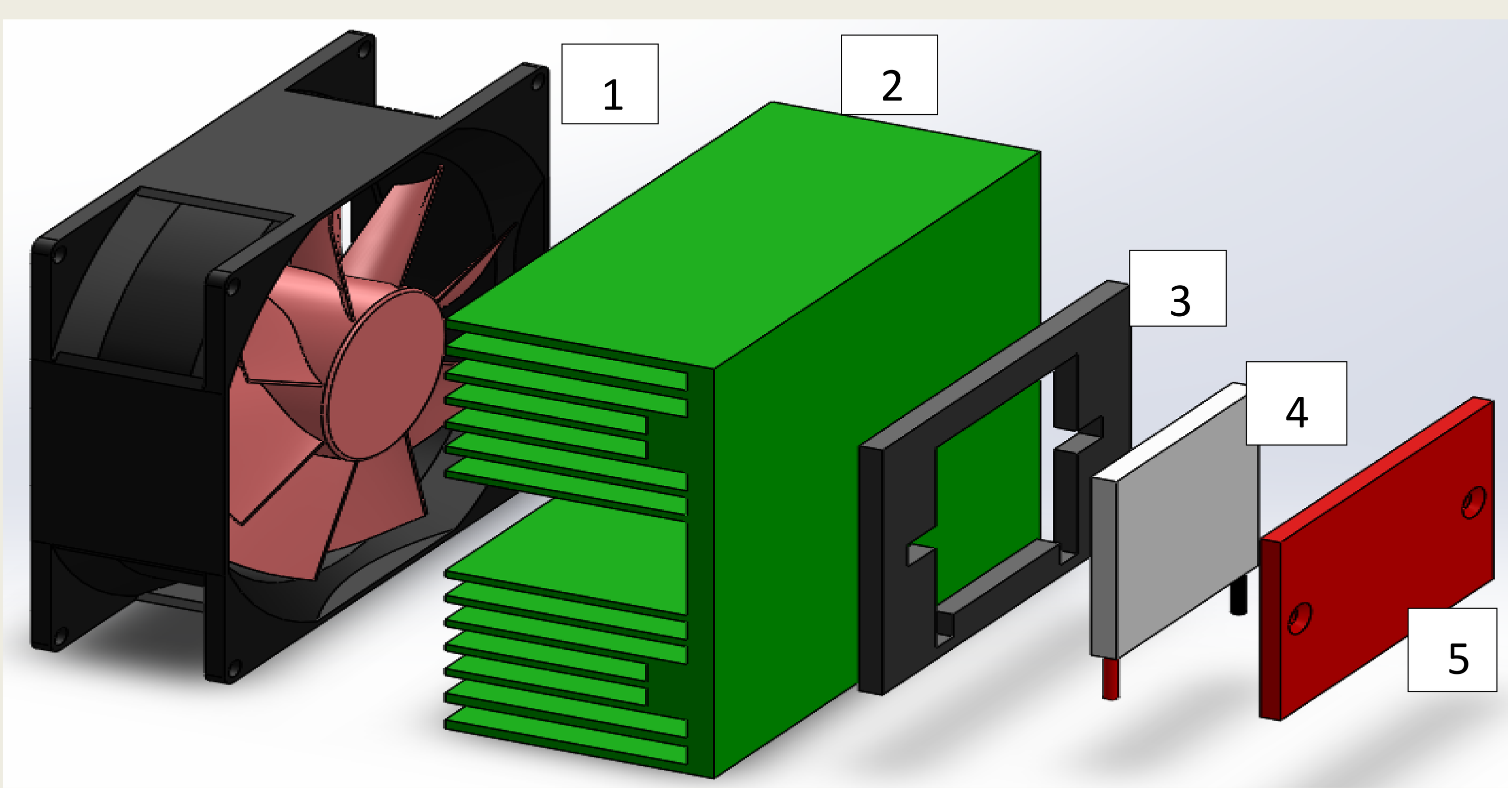

Figure 2: Exploded view of the AWG unit

ITEM NO.	PART NAME	FUNCTION
1	Fan	Cool the heat sink
2	Aluminum heat sink	Dissipate heat generated by the Peltier
3	Insulating foam	Insulate the Peltier from convective heat exchange
4	Peltier device	Provide cooling to the condensation plate
5	Condensation plate	Collect water condensation generated by the Peltier

Table 1: Parts identification from AWG unit

Method

Computational Fluid Dynamics is a branch of fluid mechanics that uses numerical analysis to solve for fluid flows with or without solid interaction. A CFD analysis examines fluid flow in accordance with its physical properties such as velocity, pressure, temperature, density and viscosity. In this project, it is used primarily to study the heat exchange between the solids in contact and between the solids and the ambient air conditions.

A base run of the study is conducted testing the Peltier at 1A, with the aim of studying the simultaneous heating and cooling effect of the Peltier. The images below are a comparison of the initial study with and without fan generated air flow.

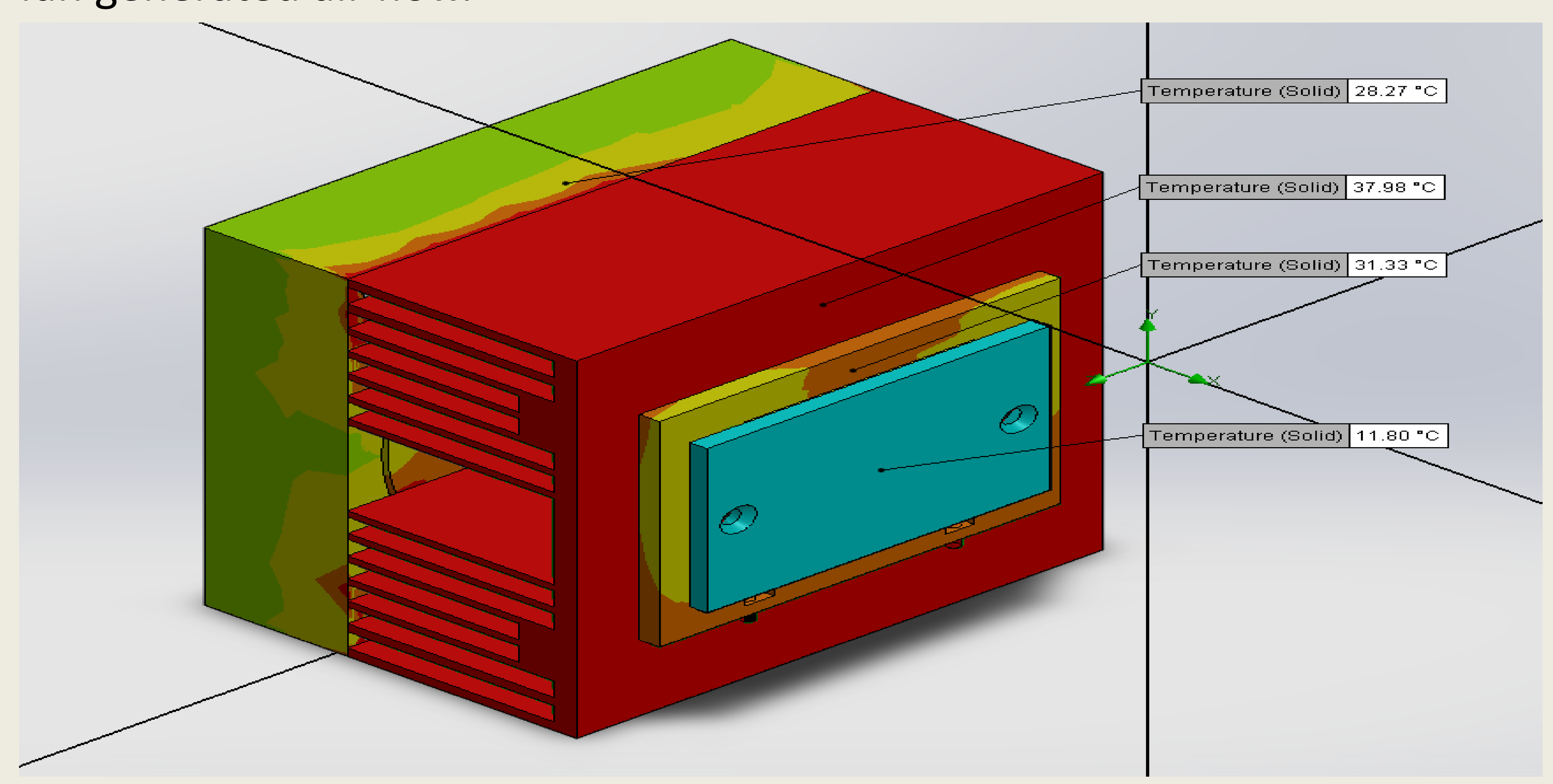

Figure 3: Initial study of the Peltier at 1A and no fan generated air flow (To = 21.11°C)

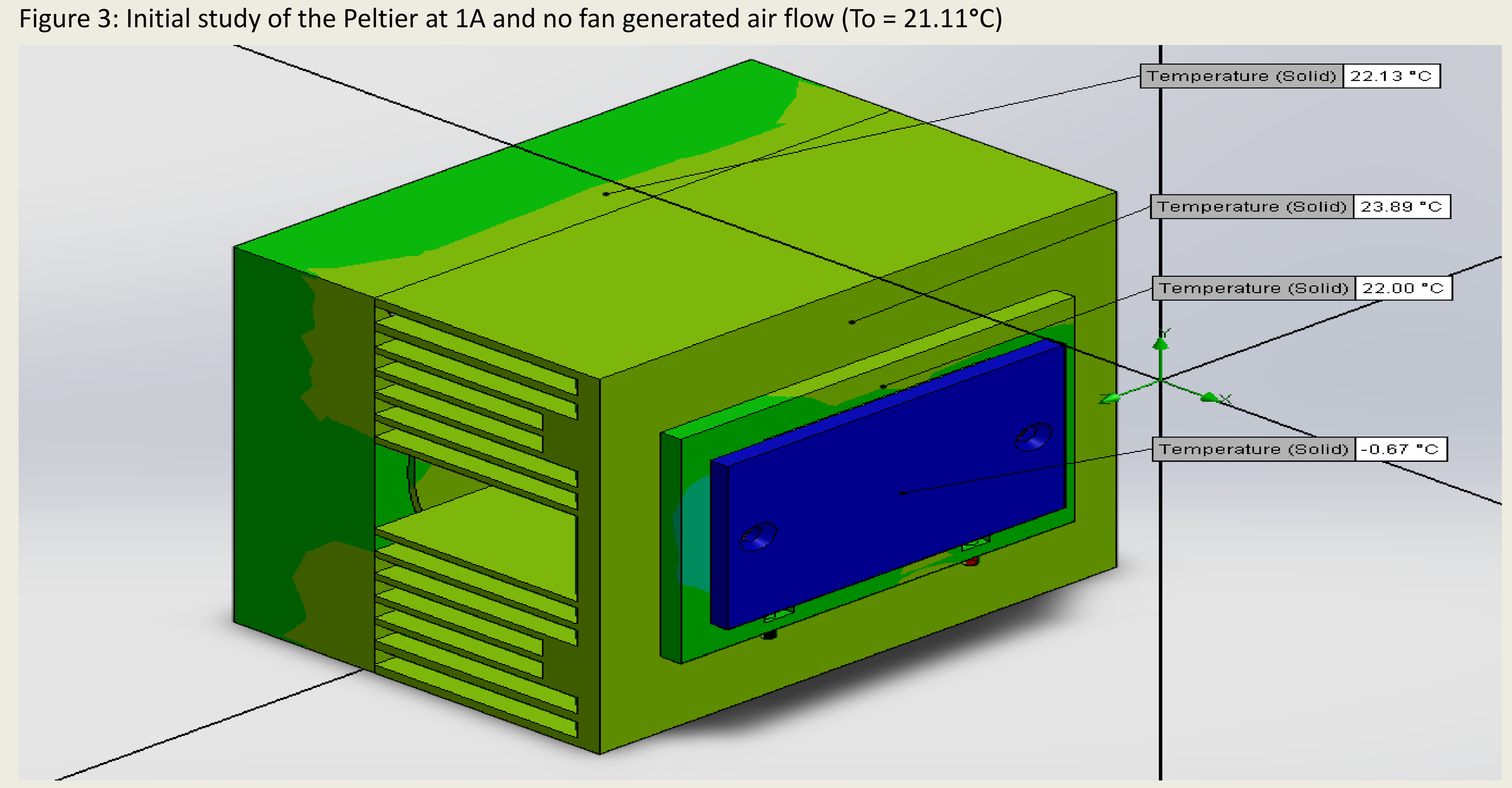

Figure 4: CFD study of the Peltier at 1A with fan generated air flow

Temperature Change	1A	1.5A	2A	2.5A	3A	Fan efficiency
With fan	-0.65°C	-6.97°C	-10.91°C	-12.78°C	-12.88°C	29.51%
Without fan	11.77°C	16.12°C	25.08°C	35.52°C	54.71°C	

Table 2: Results simulating the Peltier at 0.5A intervals

Noting that between 2.5A and 3A there is little change in the surface temperature of the condensation plate, it is concluded that there is no use in running the Peltier beyond 3A. At this point, a CFD study is conducted to examine how the heated air flows in a surrounding of more than one AWG unit.

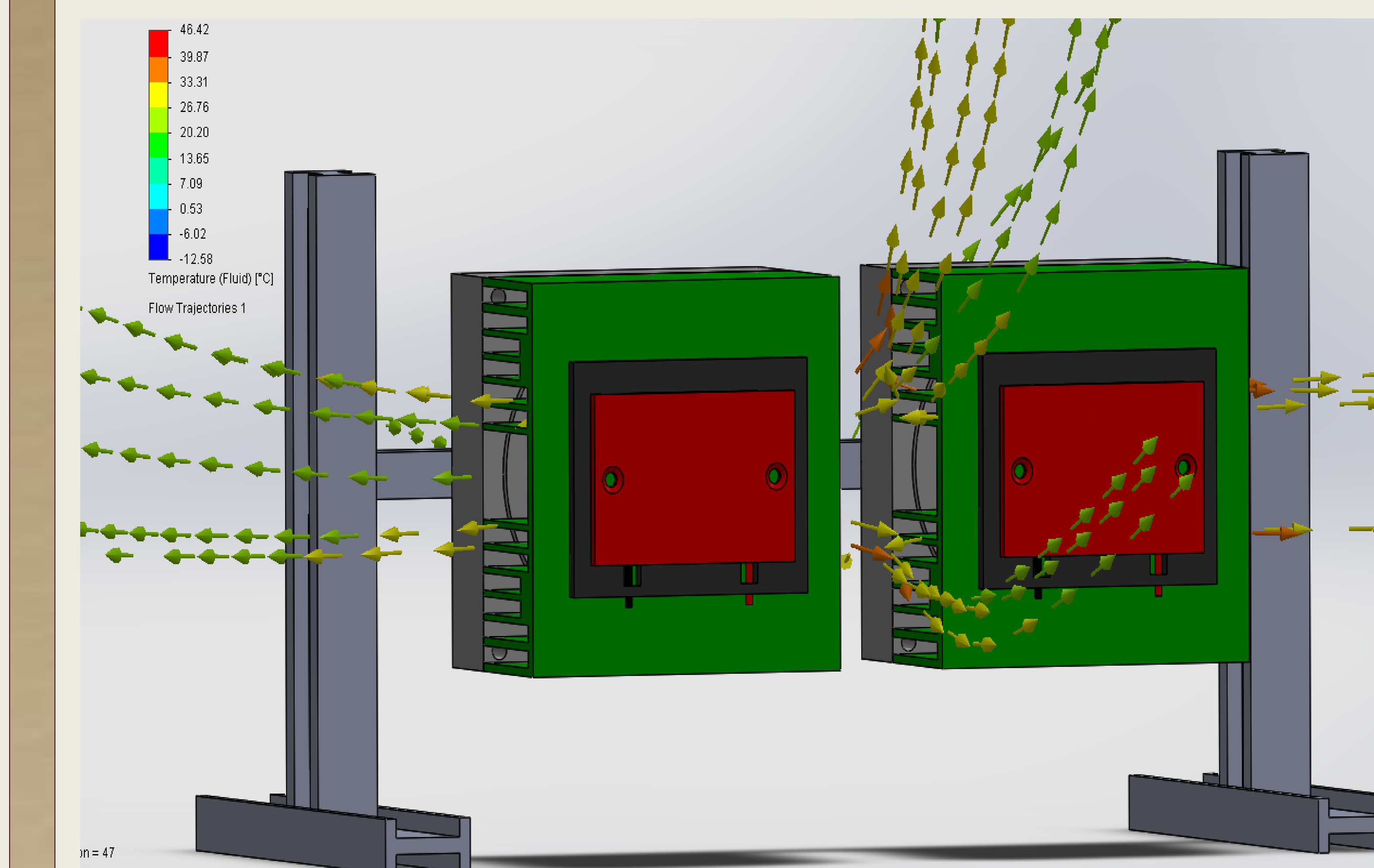

Figure 5: Flow trajectory of the convective heat exchange between heat sinks

Conclusion and Future Works

The initial conjecture was that the heated air flowing out of one heat sink would flow into the other. The CFD analysis has shown that the air in this scenario behaves as an incompressible fluid and would never flow into a heat sink that also has heated air flowing out of it. Future works involves using CFD to find an optimal method of cooling an assembly of multiple AWG units. The study would compare the difference in cooling using small fans for individual AWG units versus using a large fan to cool up to four AWG units. It will also be used to check different orientations that might affect the cooling so as to optimize the efficiency of the Peltier.

Acknowledgments

This work has been completed with funding from the US Department of Agriculture (grant # 2016-38422-25540) and Department of Education-MSEIP program (grant #P120A140065). The authors would like to thank the USDA, ED, and Texas State University for providing funding and access to infrastructure and laboratories. Sponsors are not responsible for the content and accuracy of this article.

References

"Specification of Thermoelectric Module-TEC1-12706". *Thermonamic Module*