ASTFE American Society of Thermal and Fluids Engineers

9th Thermal and Fluids Engineering Conference (TFEC)
April 21-24, 2024
Partially Online Virtual and at Oregon State University, OR Conference

TFEC-2024-50521

LASER-PROCESSED CONDENSING HEAT EXCHANGER FOR SPACE APPLICATIONS

Tyler R. Hatch[1*], Kyle W. Monaghan[1], Ezra O. McNichols[1], Cody W. Farinacci[1], John Sanders[2], Mike Izenson[3]

[1]NASA Glenn Research Center, 21000 Brookpark Rd., Cleveland, OH 44135, USA
[2]Edare LLC, 48 Lafayette St., Lebanon, NH 03766 USA
[3]Creare LLC, 16 Great Hollow Rd., Hanover, NH 30755 USA

ABSTRACT

In sizing a condensing heat exchanger for space life support systems, there is a limit to allowable pressure drop along with a required vapor recuperation rate. This sizing is performed by using known heat transfer and pressure drop correlations in a numerical model. Dimpling the surface of an air-liquid channeled heat exchanger increases the convective heat transfer rate, however, this creates difficulties within the numerical model as the flat plate correlations are no longer accurate. To increase the fidelity of the numerical model used for sizing the heat exchanger, Computational Fluid Dynamics (CFD) with an SST k-ω turbulence model was performed on a periodic slice portion of the rectangular heat exchanger. Through CFD modeling a heat flux and pressure drop was calculated across the heat exchanger channel and this information was used to create a localized Nusselt number and incorporated into the numerical model along with a more accurate pressure drop due to the dimpled surface. Additionally, through modeling the heat exchanger, there was an increase in the measurement accuracy of geometric features via Computer Aided Drafting (CAD) software which was incorporated into the numerical model for further increase in fidelity

KEY WORDS: Computational methods, Heat exchanger, spacecraft, turbulent flow, pressure drop

NOMENCLATURE

A	Area	(m^2)	Nu	Nusselt Number	(-)
Cp	Specific heat capacity at constant pressure	(J/kg/Kg)	$\overline{Nu}$	Average Nusselt Number	(-)
D_h	Hydraulic Diameter	(m)	q''	Heat Flux	(W/m^2)
h	Heat Transfer Coefficient	($W/m^2/K$)	ρ	Density	(kg/m^3)
$\bar{h}$	Average Heat Transfer Coefficient	($W/m^2/K$)	T_{ref}	Reference Temperature	(K)
k	Thermal Conductivity	(W/m/K)	T_w	Wall Temperature	(K)
L	Length	(m)	U	Velocity Vector	(m/s)
n	Normal vector to wall	(m)	u	X-direction Velocity	(m/s)

1. INTRODUCTION

One of NASA's goals to enable long-duration space exploration requires a robust condensing heat exchanger (CHX) for life support systems. Currently, the condensing heat exchangers used for space systems rely on coatings to achieve proper wetting properties. However, these coatings have been known to prematurely lose their hydrophilicity, leading to water carryover and ultimately premature CHX refurbishment. NASA's 2020 Taxonomy Roadmap cites the need for development of a robust condensing heat exchanger as a "Tier 1 Gap", meaning the agency has a high priority need for development of this technology. Specifically, CHX is cited as Gap 06-50 in the document [1]. Recently, advancements in the manufacturing process for electroplating silver and stainless steel has shown to be feasible for CHX development. There has not been a successful way to bond these two metals sufficiently to allow the necessary heat transfer for condensation/cooling and exhibit long performance in the relevant environment (e.g., high humidity). Prior

*Corresponding Author: tyler.r.hatch@nasa.gov

work has been performed using a steel/silver packet for condensation in a CHX unit [2]. Laser processing of the packet with a femtosecond laser has shown to modify the surface properties such that the wetting behavior for condensation is enhanced. This also reduces dependency on coating applications, as the geometry is permanently altered by the laser processing. NASA’s Laser Processed Condensing Heat Exchanger (LP-CHX) project sought to investigate this manufacturing process, ultimately hoping to improve the cost savings and lead time significantly compared to current state-of-the-art. In parallel, a preexisting pressure drop thermal model was analyzed and updated to reflect changes in the design for a more accurate picture of the expected performance.

In this work, we describe the initial efforts to model a CHX using this packet design in a thermal-fluid model. As the design was updated based on lessons learned from laboratory testing, new additions such as dimples were added to increase performance. However, the original model used to calculate heat transfer data did not include the dimpling pattern. They were not included in the original model due to lack of data to modify the flat-plate analysis. The existing simulation was updated to incorporate these new parameters and was also run alongside a new computational fluid dynamics (CFD) simulation to validate the original results.

2. CONDENSING HEAT EXCHANGER OVERVIEW

The condensing heat exchanger is comprised of a series of fluid-air heat exchangers stacked upon each other, forming a cassette as seen below in Fig. 1. In order for water to condense out of the air, the plates are chilled with a cooling loop flowing along the upper section. Air passes between the fluid-air heat exchangers through small channels, where the air condenses onto the plate of the heat exchanger and the condensate is collected. The height of these channels can be increased to mitigate pressure drop, but this modification comes at the cost of reduced heat transfer efficiency and, consequently, a decrease in the condensation rate. An optimal channel height was found to be 1.0668 mm (0.042”). A dimpled surface was applied to the air side of the fluid-air heat exchangers as seen in Fig. 2. Dimples were convex 0.508mm (0.02”), and spaced 5.7404mm (0.226”) apart in both the X and Y direction They obstruct air flow and encourage circulation and condensation.

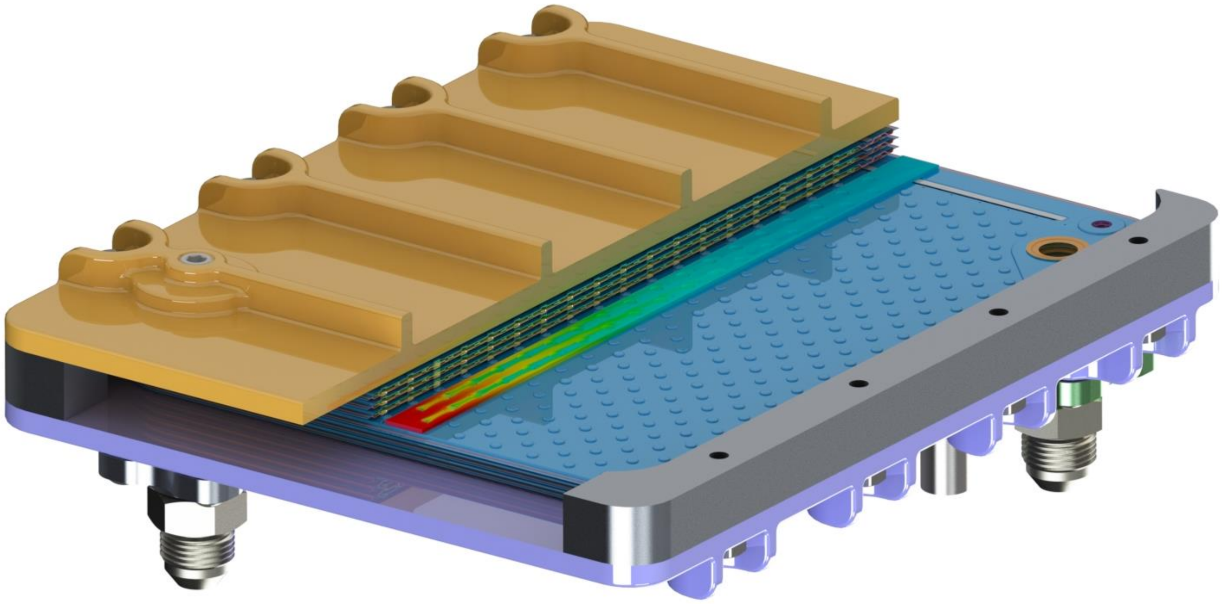

Fig. 1 Cut-away rendering of LP-CHX with Computational Fluid Dynamics domain superimposed.

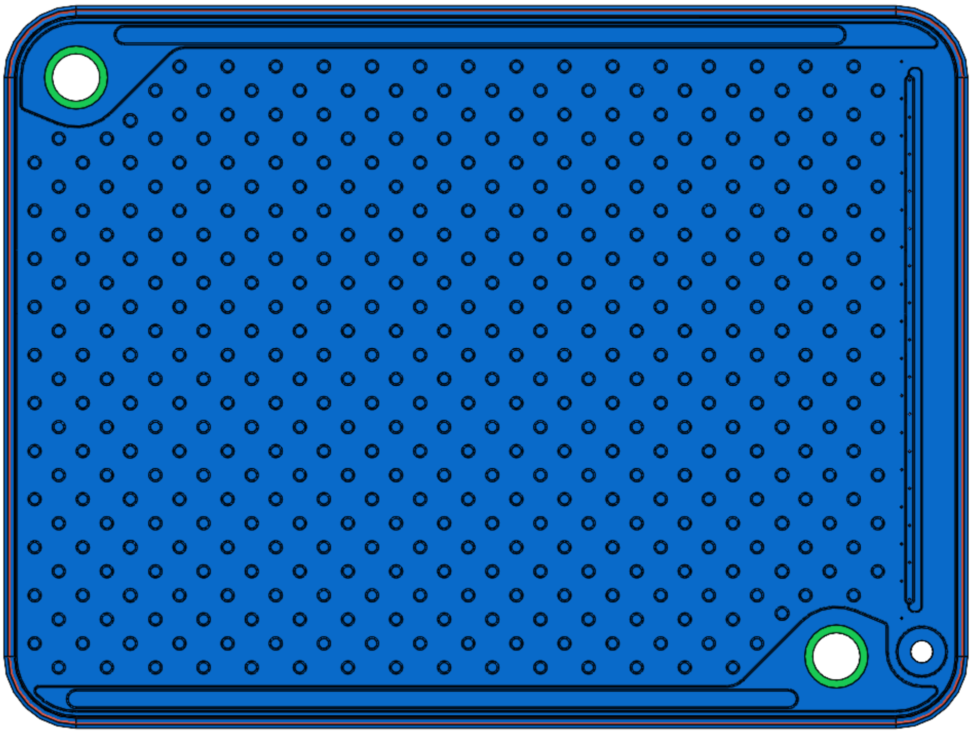

Fig. 2 Condensing Heat Exchanger surface with convex dimples shown.

3. NUMERICAL MODEL

A numerical model was created to size and analyse the performance of a condensing heat exchanger. Initially the model utilized two different methods, one where the fluid condensation on the heat exchanger created a thick film which impedes heat transfer as the air condenses, and another that used a thin film which does not impede heat transfer. Through experimental analysis it was determined that there was a low enough condensation rate that a thin film model would be more reflective of the heat exchanger. In the numerical model, an individual channel is divided into 1-inche wide cells, and the model steps through calculating an initial temperature by satisfying the enthalpy balance and from this a pressure drop, heat transfer rate, and condensation rate is calculated. An assumption made within the numerical model was to assume flat plate correlations for pressure drop and heat transfer across the channel. During experimental analysis recorded pressure drops increased from the predicted model. To increase the fidelity of these calculations and incorporate true geometric features, computational fluid dynamics was performed to determine a more accurate pressure drop and Nusselt number which was then re-incorporated into the numerical model.

Additionally, the condensing heat exchanger unit was planned to undergo laser processing of the dimpled surface to increase wettability and enhance the condensation process. As a result of the laser processing, the surface height and thermal resistance changed slightly. Thus, a second coating was applied in the numerical model to account for this thermal resistance based on the known properties of the processing method [3].

4. COMPUTATIONAL FLUID DYNAMICS

To decrease computational time and the requirement of resolving dimples within a mesh, a periodic portion of a single channel was modelled. Two models were meshed with the software Pointwise, a non-dimpled channel and a dimpled surface channel. Meshes were created with a mesh density to resolve the dimpled surfaces of 14,901,375 nodes, and 2,521,950 nodes for the undimpled surface. Meshes were exported with boundary surfaces to Fluent. Computational Fluid Dynamics (CFD) was performed using Ansys Fluent 2021. To resolve flow through a narrow passageway as recommended via turbulence model documentation, an SST k-ω turbulence model was chosen [4]. Inlet boundary condition was set as a velocity inlet, the fluid velocity was provided through the numerical model. The outlet boundary condition was defined as a pressure-outlet with a target mass flow. The mass flow information was also provided by the numerical model. A convergence metric of decreasing residuals by an order of three in magnitude of initial condition was used. To decrease computational time, a middle portion of a single channel of the heat exchanger was modelled using periodic boundaries on its sides (Figure 3). Then pressure drop and heat transfer was extrapolated for the entire condensing heat exchanger, shown in Figure 4.

To determine mesh independence to verify that CFD results are independent of the mesh, a secondary coarser mesh was created. The increase in mesh size resulted in 2,101,545 cells, a large reduction in cells from the utilized extra fine mesh. CFD was ran until a similar convergence criterion was met. The coarser mesh resulted in an average heat flux difference of 3.3%, a 5.1% change in Nusselt number, and a difference in total pressure drop from inlet to exit of 1.1%. Due to the closeness in results generally being within 5%, it was determined that the results from the extra fine mesh are independent of the mesh itself. Furthermore, we had confidence in the mesh sizing since CFD analysis matched with textbook calculations for a flat channel [5]. Additionally, computational time did not warrant reduction in node count for efficiency. The increased node count between meshes was to account for meshing the dimples.

Table 1. Mesh Independence Study

Key Performance Indicators (KPIs) Change	
# of Mesh Cells	14% of extra fine amount
Average Heat Flux	3.3%
Nusselt Number	5.1%
Total Pressure Drop	1.1%

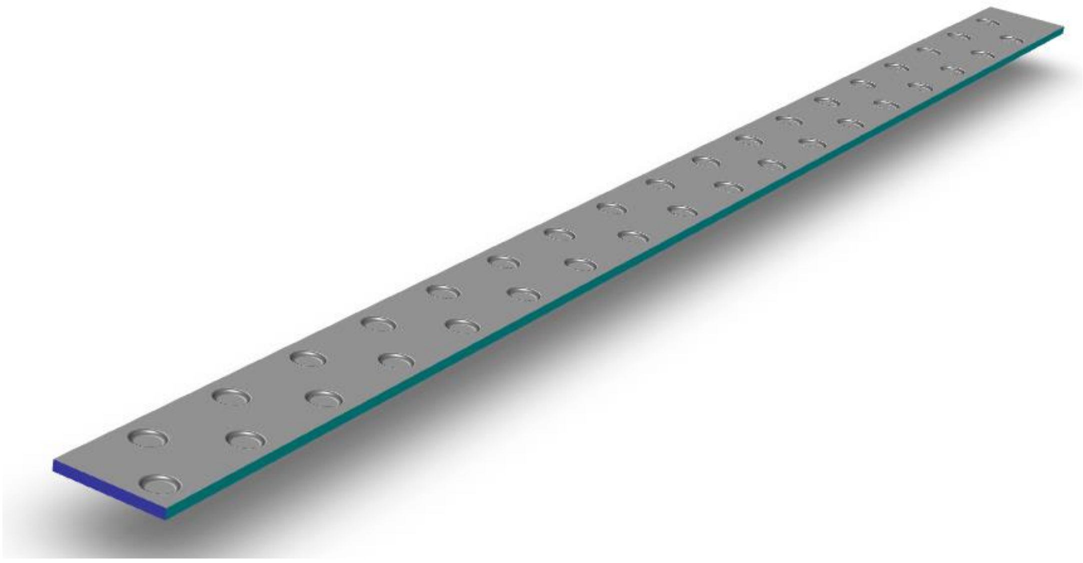

Fig. 3 Sliced view of dimpled surface used for modeling.

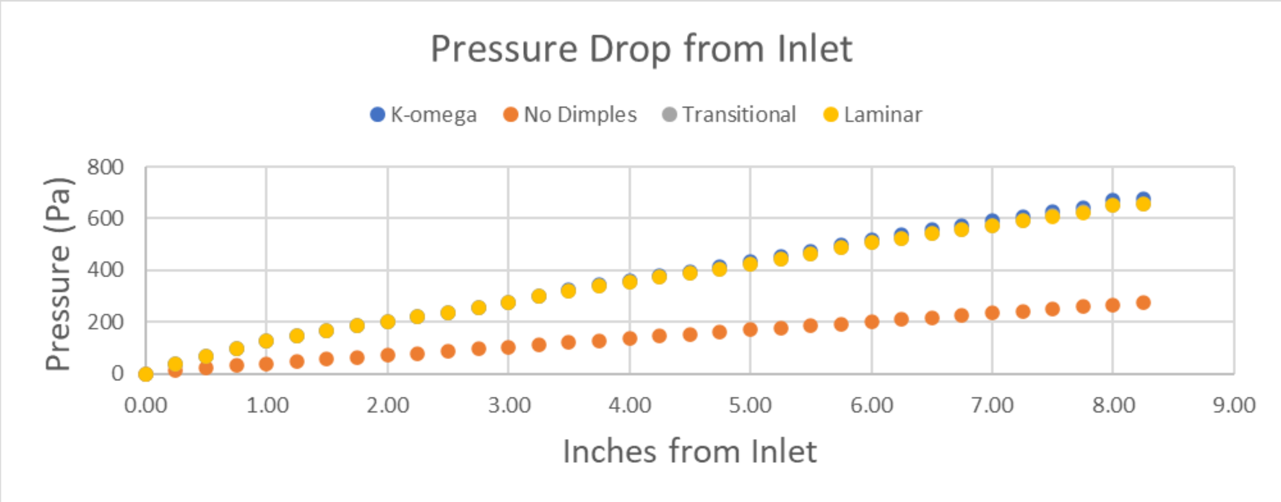

Fig. 4 CHX Pressure drop versus distance for various models. The SST k-ω and laminar models showed good agreement here with prior data, indicating that either would be suitable for representing the flow for pressure drop calculations. The geometry without dimples was also run as a control case.

5. FLOW CHARACTERIZATION

The Reynolds number was just above the criteria for internal flow to be characterized as laminar. As such, the heat exchanger was analysed using a laminar model, and transitional model, and a few turbulence models, shown in Figure 5 below. Figure 6 shows the total heat exchanged versus total pressure drop across the whole channel for four case scenarios.

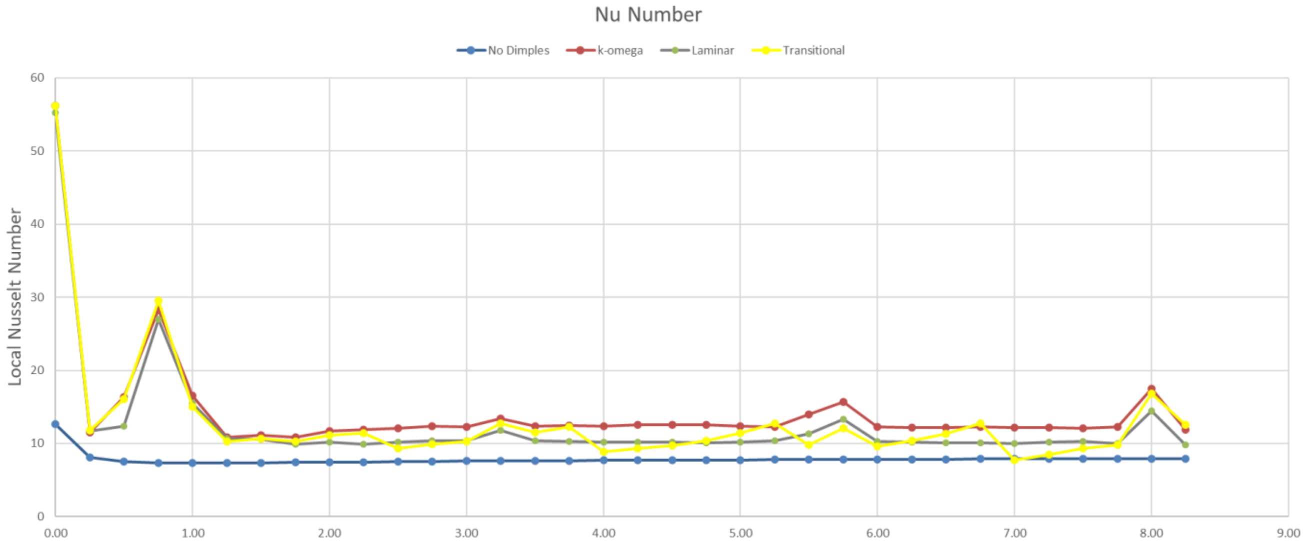

Fig. 5 CHX Nusselt Number for various models. Again, the SST k-ω and laminar models showed good agreement here, however the laminar flow model slightly underestimated Nu compared to the turbulent case. Prior work without the dimpled pattern is also shown for comparison.

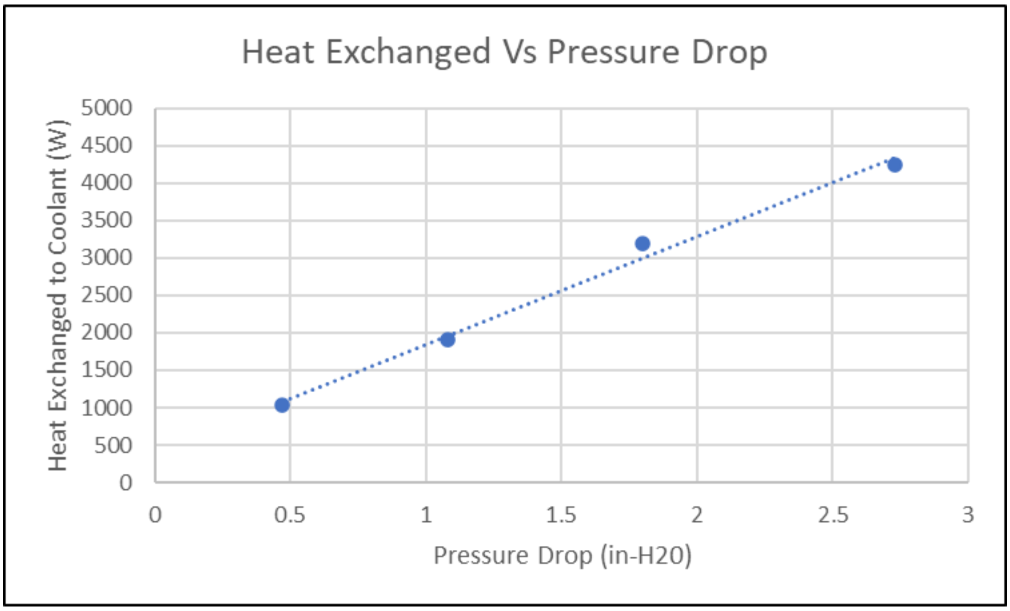

Fig. 6 CHX total heat exchanged versus total pressure drop for four different loading cases.

3.1 Nusselt Number Calculations

The Nusselt Number was calculated here as a dimensionless number representing the ratio of convective to conductive heat transfer at a wall boundary in the fluid. Both advection (fluid motion) and diffusion (conduction) were included in the value, chosen to represent the overall performance profile of the system. The heat transfer coefficient, *h*, was calculated using Equation 1 below. The variable T_{ref} can be chosen arbitrarily but must be reported for repeatability and consistency, shown in Equation 2. In Ansys, this variable defaults to a constant value. However, for this study T_{ref} was calculated as the bulk fluid temperature along the length of the heat exchanger.

$$h = \frac{q''}{T_w - T_{ref}} = \frac{-k\frac{\partial T}{\partial n}\Big|_{at\ wall}}{T_w - T_{ref}} \tag{1}$$

Where

$$T_{ref} = \frac{\iint \rho C_p u T dA}{\rho C_p UA} \tag{2}$$

The average Nusselt number, $\overline{Nu}$, was calculated using Equation 3, which includes the average heat transfer coefficient $\bar{h}$ determined via integration in Equation 4.

$$\overline{Nu} = \frac{\bar{h} D_h}{k} \tag{3}$$

Where

$$\bar{h} = \frac{1}{L}\int_0^L h(x)\, dx \tag{4}$$

Ansys Fluent was used to solve these equations along the length of the slice.

6. CONCLUSIONS

CHX's are a prominent technology in use for various NASA lift support systems such as humidity control of the ISS. As NASA is preparing for longer duration missions to the moon and beyond, it is paramount that these systems can operate for long periods with minimal downtime for repairs. The LP-CHX work is a frontrunner as a potential successor for current ISS/Shuttle era heritage CHX technology that could be used in missions to the moon or Mars. This effort for LP-CHX builds on prior work in establishing laser-processed sheets for enhanced heat transfer and longevity in CHX applications. Although prior quarter-scale units were developed, the design has evolved so that the original models used to validate that work are in need of updates. This project updated much of the CAD files and modeling parameters for the LP-CHX project to use as a standard for future hardware development of the new design. After updating the Nusselt Number in the excel analysis, reported total heat transfer increased 21.3%. Now that a window of possible values for the Nusselt number and pressure drop have been determined, future work in assembling the full-scale unit can use these values as benchmarks for predicting performance of the system.

ACKNOWLEDGMENTS

The authors would like to thank the NASA Polaris Program for funding this effort and providing guidance to the team throughout the project. Additionally, we would like to acknowledge NASA Contract 80NSSC22P0096 for allowing both Creare and Edare to support this effort.

REFERENCES

[1] National Aeronautics and Space Administration, (2020). *2020 NASA Taxonomy.* Retrieved November 11, 2023, from https://www.nasa.gov/otps/2020-nasa-technology-taxonomy/

[2] Hansen, S., Castro-Wallace, S., Hamilton, T., Zuhlke, C., Alexander, D., & Fischer, B. (2018, July). Continued Laser Processed Condensing Heat Exchanger Technology Development. 48th International Conference on Environmental Systems.

[3] Roth, N., Zuhlke, C., Peng, E., Hansen, S., Shield, J. E., & Alexander, D. (2018). Creation of micro/nano surface structures on silver using collinear double femtosecond laser pulses with different pulse separation. Multiscale and Multidisciplinary Modeling, Experiments and Design, 1, 145-153.

[4] Comsol, (2017). *Which Turbulence Model Should I Choose for My CFD Application?* Retrieved 11 November 2023 from https://www.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/

[5] Ansys, (2021). *Forced Convection Over a Flat Plate*. Retrieved 12 Nov, 2023, from https://courses.ansys.com/wp-content/uploads/2021/02/LT4_C2_L3-Handout-v2.pdf