



NAVAL POSTGRADUATE SCHOOL

MONTEREY, CALIFORNIA

THESIS

**AN ASSESSMENT OF GYROID TRIPLY PERIODIC
MINIMAL SURFACES FOR ADDITIVELY
MANUFACTURED HEAT EXCHANGERS**

by

Derrick J. Thompson

December 2023

Thesis Advisor:

Co-Advisor:

Second Reader:

Walter C. Smith

Sean O'Brien (NNL)

Garth V. Hobson

Approved for public release. Distribution is unlimited.

THIS PAGE INTENTIONALLY LEFT BLANK

REPORT DOCUMENTATION PAGE			<i>Form Approved OMB No. 0704-0188</i>	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instruction, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188) Washington, DC, 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE December 2023	3. REPORT TYPE AND DATES COVERED Master's thesis	
4. TITLE AND SUBTITLE AN ASSESSMENT OF GYROID TRIPLY PERIODIC MINIMAL SURFACES FOR ADDITIVELY MANUFACTURED HEAT EXCHANGERS			5. FUNDING NUMBERS	
6. AUTHOR(S) Derrick J. Thompson				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Postgraduate School Monterey, CA 93943-5000			8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) N/A			10. SPONSORING / MONITORING AGENCY REPORT NUMBER	
11. SUPPLEMENTARY NOTES The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government.				
12a. DISTRIBUTION / AVAILABILITY STATEMENT Approved for public release. Distribution is unlimited.			12b. DISTRIBUTION CODE A	
13. ABSTRACT (maximum 200 words) With the Navy integrating new technologies and capabilities into its fleet, there is a corresponding increase in the power demand. However, this increased power generation also results in higher heat production, which poses a risk of failures. To address these challenges, the U.S. Navy (USN) is interested in exploring the potential of additive manufacturing processes for producing heat exchangers. The objective of this project is to leverage additive manufacturing to create smaller, more cost-effective, and highly efficient heat exchangers. Sponsored by the USN, this research focuses on enhancing the operational readiness, cost-effectiveness, and efficiency of power generation systems. The outcomes of this research will hold substantial value for the undersea warfare (USW) community, as this will contribute to improved operational readiness, cost reduction, and increased efficiency of power generation systems in submarines. This research used triply periodic minimal surfaces to improve the performance of heat exchangers by more than 400% and decrease size by almost 84%.				
14. SUBJECT TERMS heat exchanger, additive manufacturing, heat coefficient, fin, powder bed fusion			15. NUMBER OF PAGES 103	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UU	

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18

THIS PAGE INTENTIONALLY LEFT BLANK

Approved for public release. Distribution is unlimited.

**AN ASSESSMENT OF GYROID TRIPLY PERIODIC MINIMAL SURFACES
FOR ADDITIVELY MANUFACTURED HEAT EXCHANGERS**

Derrick J. Thompson
Ensign, United States Navy
BS, United States Naval Academy, 2022

Submitted in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

from the

**NAVAL POSTGRADUATE SCHOOL
December 2023**

Approved by: Walter C. Smith
Advisor

Sean O'Brien
Co-Advisor

Garth V. Hobson
Second Reader

Brian S. Bingham
Chair, Department of Mechanical and Aerospace Engineering

THIS PAGE INTENTIONALLY LEFT BLANK

ABSTRACT

With the Navy integrating new technologies and capabilities into its fleet, there is a corresponding increase in the power demand. However, this increased power generation also results in higher heat production, which poses a risk of failures. To address these challenges, the U.S. Navy (USN) is interested in exploring the potential of additive manufacturing processes for producing heat exchangers. The objective of this project is to leverage additive manufacturing to create smaller, more cost-effective, and highly efficient heat exchangers. Sponsored by the USN, this research focuses on enhancing the operational readiness, cost-effectiveness, and efficiency of power generation systems. The outcomes of this research will hold substantial value for the undersea warfare (USW) community, as this will contribute to improved operational readiness, cost reduction, and increased efficiency of power generation systems in submarines. This research used triply periodic minimal surfaces to improve the performance of heat exchangers by more than 400% and decrease size by almost 84%.

THIS PAGE INTENTIONALLY LEFT BLANK

TABLE OF CONTENTS

I.	MOTIVATION	1
II.	LITERATURE REVIEW	3
A.	HEAT EXCHANGER BASICS.....	3
B.	HEAT EXCHANGER DESIGN.....	4
C.	STATE-OF-THE-ART HEAT EXCHANGER DESIGN	6
D.	ADDITIVELY MANUFACTURING HEAT EXCHANGERS.....	9
E.	TRIPLY PERIODIC MINIMAL SURFACE	11
F.	COST ANALYSIS	16
G.	COMPUTATIONAL FLUID DYNAMICS.....	21
III.	RESEARCH DESIGN	27
A.	DESIGN PARAMETERS AND METHODS	27
B.	INITIAL CALCULATIONS	28
IV.	UNIT CELL.....	31
A.	UNIT-CELL MESH SENSITIVITY STUDY	33
B.	SETUP.....	40
V.	RESULTS	43
VI.	CONCLUSION	59
A.	FUTURE WORK.....	60
	APPENDIX A. MATLAB CODE FOR INITIAL CALCULATIONS	61
	APPENDIX B: TWO FLUID DOMAIN SETUP.....	63
	APPENDIX C: LARGE SCALE MODEL.....	65
	APPENDIX D: MATLAB CODE TO POST-PROCESS THE DATA.....	67
	LIST OF REFERENCES.....	75
	INITIAL DISTRIBUTION LIST	81

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF FIGURES

Figure 1.	Traditional Shell-and-Tube Heat Exchanger. Source: [6]	2
Figure 2.	Additively Manufactured Gyroid-Lattice Heat Exchanger with Two Fluid Domains Indicated by the Red and Green. Source: [7]	2
Figure 3.	Schematic of a Power Plant. Source: [11]	4
Figure 4.	Flow Paths for Shell-and-Tube Heat Exchangers. Source: [13]	6
Figure 5.	Schematic of Rankine Cycle. Source: [15]	7
Figure 6.	State-of-the-Art Heat Exchanger Design. Adapted from [14]	8
Figure 7.	Schematic of Powder Bed Fusion. Source: [20]	10
Figure 8.	Importance of Support Structures. Source: [24]	11
Figure 9.	Bioinspired TPMS Unit-Cells. Source: [25]	11
Figure 10.	Gyroid TPMS Unit Cell in Ntopology	13
Figure 11.	Moody Diagram. Source: [28]	14
Figure 12.	Area Averaged Nu Number of (a) Hot Channel (b) Cold Channel. Source: [25]	15
Figure 13.	Area Averaged Friction Factor in (a) Hot Channel (b) Cold Channel. Source: [25]	15
Figure 14.	Additively Manufactured TPMS Heat Exchanger. Source: [7]	16
Figure 15.	Flow Chart of Manufacturing Process of a Shell-and-Tube Heat Exchanger. Source: [31]	17
Figure 16.	Cost of Shell-and-tube Heat Exchanger in Relation to Shell Diameter. Source: [30]	18
Figure 17.	Concept Laser X Line 2000R Printer. Source: [32]	20
Figure 18.	Estimate Cost \$/lb of AM Parts. Source: [35]	21
Figure 19.	Beta Function of Producing CAD Bodies From Implicit Bodies in Ntopology	32
Figure 20.	SolidWorks of Unit Cell	32

Figure 21.	Workflow for Unit-Cell Study	33
Figure 22.	Histogram of Y^+ Values for Coarse Mesh	34
Figure 23.	CFD Periodic Setup	36
Figure 24.	Residuals for Mesh Sensitivity Study	37
Figure 25.	X-Velocity Monitor Point	38
Figure 26.	Shear Stress in Mesh Sensitivity	38
Figure 27.	Velocity Magnitude Vector Field of Each Mesh (m/s).....	39
Figure 28.	Coarse Mesh with Inflation Layers.....	40
Figure 29.	Fluctuation of Heat Transfer Coefficients Representing Unsteadiness of the Flow	41
Figure 30.	Residuals of Laminar Flow	44
Figure 31.	Turbulence Intensity	45
Figure 32.	Residuals from Turbulent Model Runs	46
Figure 33.	Heat Transfer Coefficient vs. Reynolds Number.....	47
Figure 34.	Nu Number Relationship to Heat Transfer Coefficient	48
Figure 35.	Nusselt Number vs. Reynolds Number.....	49
Figure 36.	Overall Heat Transfer Coefficient	51
Figure 37.	Heat Transfer Area vs. Reynolds Number.....	52
Figure 38.	Pressure Drop per Unit Length	53
Figure 39.	Heat Exchanger Length.....	54
Figure 40.	Large-Scale Model Conservative Pressure Drop.....	55
Figure 41.	Friction Factor vs. Reynolds Number	55
Figure 42.	Surface Roughness Heat Transfer Coefficient.....	56
Figure 43.	Surface Roughness Friction Factor	57
Figure 44.	Surface Roughness Pressure Drop	58

Figure 45.	Mesh of Two-Fluid Domain Setup	63
Figure 46.	Setup for Two-Fluid Domain in Fluent	64
Figure 47.	Pathlines of Failed Two-Fluid Domain Setup	64
Figure 48.	Implicit Model of TPMS Heat Exchanger	65
Figure 49.	Fault-Tolerant Mesh without Inflation Layers.....	66

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF TABLES

Table 1.	Limiting Parameters for Heat Exchangers. Adapted from [14].....	8
Table 2.	Cost of Materials. Adapted from [31].....	18
Table 3.	Cost For Tubing. Adapted from [30].....	19
Table 4.	Factors for Type of Construction. Adapted from [30].....	19
Table 5.	Factor for Baffle Orientation. Adapted from [30]	19
Table 6.	Design Parameters	27
Table 7.	Water Properties Used in Calculations. Adapted from [40]	28
Table 8.	Metal Properties Used in Calculations. Adapted from [41, [42]	28
Table 9.	Initial Calculations for Model Using TPMS	29
Table 10.	Initial Calculations for TPMS and Shell-and-Tube Heat Exchanger.....	29
Table 11.	Mesh Sensitivity Study	34
Table 12.	Range of Flow Parameters	43
Table 13.	Heat Transfer Area and Overall Heat Transfer Coefficient.....	50

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF ACRONYMS AND ABBREVIATIONS

AM	additive manufacturing
CFD	computational fluid dynamics
HTC	heat transfer coefficient
LMTD	log mean temperature difference
PBF	powder bed fusion
PCHE	printed circuit heat exchanger
SST	shear stress transport
TPMS	triply periodic minimal surface

THIS PAGE INTENTIONALLY LEFT BLANK

NOMENCLATURE

A_s	wetted surface area
D_h	hydraulic diameter
D_ω	cross-diffusion term
E	energy
F	force
f	friction factor
G	generation of energy
g	gravitational acceleration
h	heat transfer coefficient
I	unit tensor
J	diffusion flux
K_s	sand grain equivalent roughness
k	kinetic energy
k_t	thermal conductivity
L	length
Nu	Nusselt number
η	thermal efficiency
σ	turbulent Prandtl number
P	pressure
ρ	density
Q	heat
Re	Reynolds number

T	temperature
TI	turbulence intensity
t	thickness
τ	shear stress
Γ	effective diffusivity
U	overall heat transfer coefficient
u	velocity
μ	absolute viscosity
ν	kinematic viscosity
V	volume
W	work
ω	specific rate of dissipation of kinetic energy
Y	dissipation

ACKNOWLEDGMENTS

I would like to acknowledge the help and wisdom that Dr. Smith and Sean O'Brien provided me with throughout the whole process of writing this thesis. They collectively spent many hours helping design, improve Matlab code, and overcome the many barriers ran into during this research because of the novelty of the field.

THIS PAGE INTENTIONALLY LEFT BLANK

I. MOTIVATION

The use of additive manufacturing (AM) has the potential to benefit heat exchanger design. By implementing complex geometries through AM, it becomes possible to reduce the size of heat exchangers while simultaneously improving their performance, efficiency, and cost-effectiveness. AM technology enables the production of geometries that were previously unattainable through traditional manufacturing methods. The U.S. Navy is keen on leveraging this AM technology to enhance the capabilities of its growing fleet. This study focuses on the application of AM in manufacturing heat exchangers with triply periodic minimal surfaces (TPMS). The aim is to investigate the advantages of utilizing TPMS structures in heat exchangers, specifically in terms of heat transfer and fluid dynamics.

Shell-and-tube heat exchangers, shown in Figure 1, are the most common heat exchanger type found across all industries [1]. Traditionally, two main methods are employed to manufacture shell-and-tube heat exchangers: tube drawing and casting. Tube drawing involves extruding a pipe through a die to modify its outer and inner diameter [2]. Casting is a process in which a liquefied material is poured into a specially designed cavity and allowed to cool to achieve the desired shape [3]. In the case of shell-and-tube heat exchangers, the tubes produced through drawing and the outer shell created through casting need to be joined using a combination method, typically welding. Welding is a fabrication technique that utilizes heat to fuse two materials together [4].

The continuous advancement of AM technology presents promising opportunities to implement complex geometries through AM. One of the key advantages of AM is its ability to produce intricate internal geometries with minimal post-processing requirements. A notable example is the gyroid-lattice heat exchanger produced using AM which “exhibits a 55% increase in effectiveness in comparison to a thermodynamically equivalent, most-efficient, counter-flow heat exchanger at 1/10th of its size” [5]. An example of the gyroid-lattice heat exchanger is shown in Figure 2.

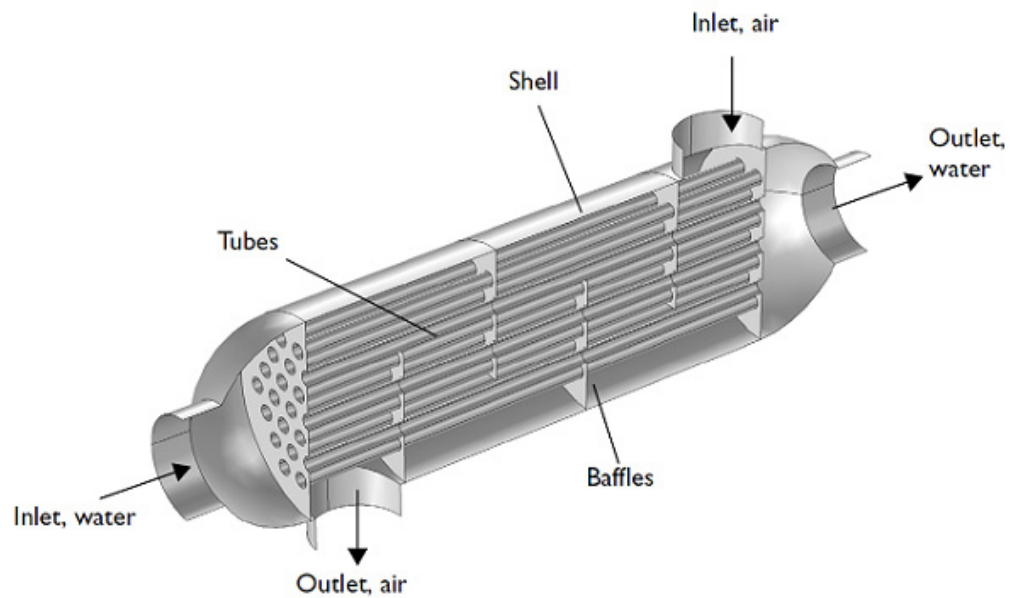


Figure 1. Traditional Shell-and-Tube Heat Exchanger. Source: [6]

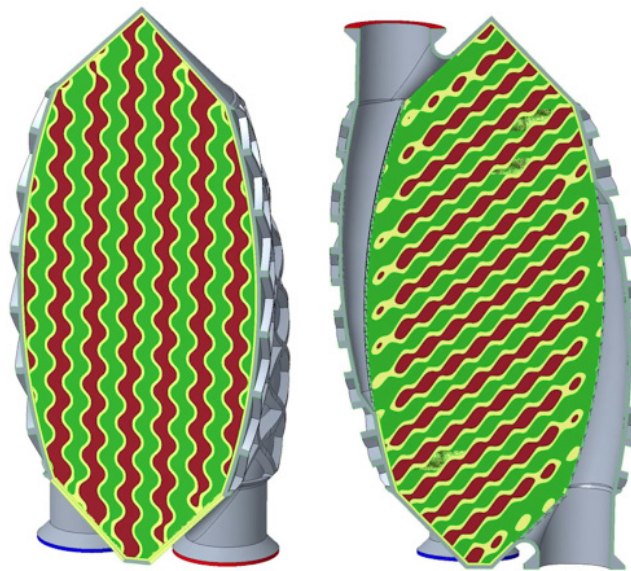


Figure 2. Additively Manufactured Gyroid-Lattice Heat Exchanger with Two Fluid Domains Indicated by the Red and Green. Source: [7]

II. LITERATURE REVIEW

A. HEAT EXCHANGER BASICS

The first widely recognized heat exchanger was the plate heat exchanger by Albrecht Dracke in 1878 [8]. Heat exchangers have continued to evolve as technology developed as they are one of the limiting factors in performance when it comes to any system that produces heat. Heat removal depends on the thermal efficiency of power-plants,

$$\eta = \frac{W_{out}}{Q_{in}}, \quad (1.1)$$

and current state-of-the art nuclear power plant Westinghouse AP1000 has a thermal efficiency of 32% [9]. Where η is the thermal efficiency, W_{out} is the output work and Q_{in} is the heat put into the system. This ratio signifies the fraction of the heat input converted to useful work. Currently, power plants operate with a thermal efficiency of 32%, meaning that only 32% of the heat supplied to the system is effectively converted into usable work or power. The remaining 68% is considered waste heat, which must be removed to ensure the power plant operates optimally. Figure 3 illustrates the schematic of a power plant. In the diagram the heat generation occurs at the red tube originating at the core. The waste heat is indicated at the condenser, which is dissipated from the turbine. Consequences of not removing the heat can lead to the plant overheating, potentially leading to damage to the machinery [10].

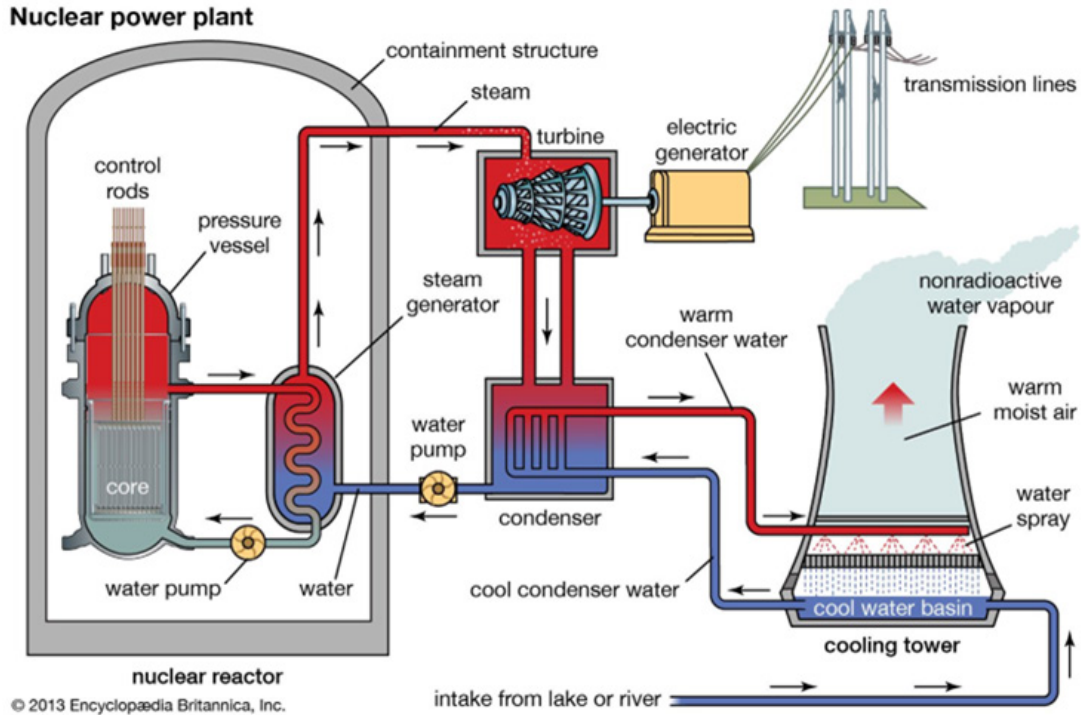


Figure 3. Schematic of a Power Plant. Source: [11]

B. HEAT EXCHANGER DESIGN

Designing an efficient, high-performing, and cost-effective heat exchanger involves considering various factors. In this design review, the focus will be on single-phase heat exchangers, particularly the widely used shell-and-tube configuration. According to Ramesh K. Shah and Dušan P. Sekulić the first step in design is the problem criteria [12]. The problem criteria includes working fluid, inlet pressure (mode of force), inlet temperature, mass flow rate and allowed pressure drop across each fluid [12].

Additionally, the environment should be considered for corrosion and fouling purposes. When all the parameters are defined the orientation of the flow must be determined. Shell-and-tube heat exchangers offer various orientations to choose from, as depicted in Figure 4. Flow arrangement depended on available packaging space and the required heat exchanger effectiveness [12].

Once the flow arrangement has been decided, the number of passes and baffles are chosen. The number of passes and baffles dictate pressure drop and most importantly heat

exchanger performance. One commonly used method to assess the performance of the heat exchanger is the log-mean temperature difference method, known as LMTD, where the total heat rejection is given by

$$Q_{out} = UA_s \Delta T_m \quad (1.2)$$

where U is the overall heat transfer coefficient measured in units of (kJ/m²K), A_s is the heat transfer area (m²), and ΔT_m is the log-mean temperature difference (K). Where ΔT_m is calculated by

$$\Delta T_m = \frac{(T_{i,t} - T_{o,s}) - (T_{o,t} - T_{i,s})}{\ln \frac{(T_{i,t} - T_{o,s})}{(T_{o,t} - T_{i,s})}} \quad (1.3)$$

where $T_{i,t}$ is the inlet tube temperature, $T_{o,t}$ is the outlet tube temperature, $T_{i,s}$ is the shell inlet temperature and $T_{o,s}$ is the shell outlet temperature. The overall heat transfer coefficient is a useful quantity to analyze how effective a dual-fluid domain heat exchanger is. U is a culmination of each domain's effective heat transfer coefficient, h_s . Where each domain's h_s can be calculated by

$$h_s = \frac{Nu_{D_h} k}{D_h} \quad (1.4)$$

where Nu is Nusselt's number, k is the thermal conductivity of the fluid and D_h is the hydraulic diameter, nominally the diameter of the tube in a shell and tube heat exchanger otherwise the characteristic length for other shapes other than circular tubes. Using each domain's h_s the U for the heat exchanger can be calculated using

$$U = \frac{1}{\frac{1}{h_i} + \frac{t}{k_t} + \frac{1}{h_o}} \quad (1.5)$$

where h_i and h_o are the two fluid domain effective heat transfer coefficients and t is the thickness of the solid interface [12].

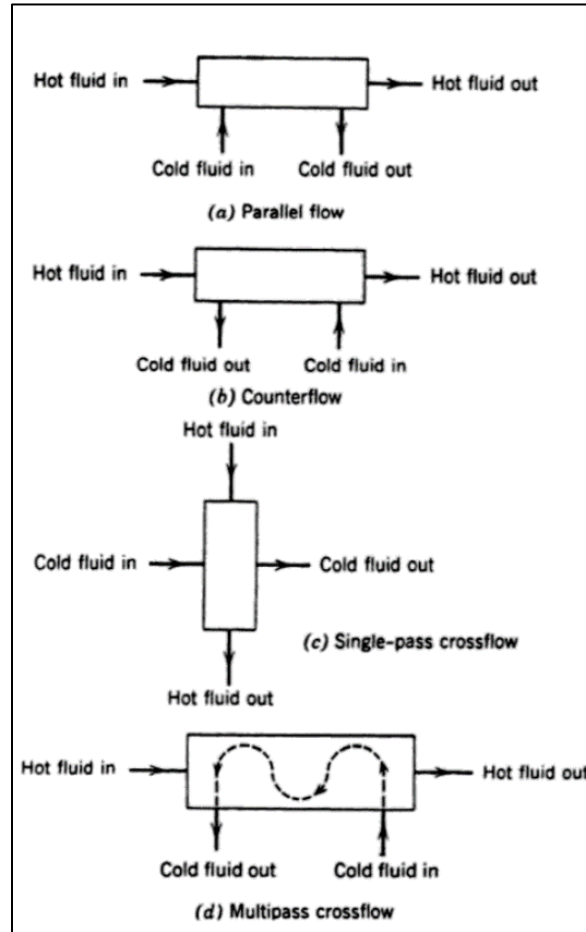


Figure 4. Flow Paths for Shell-and-Tube Heat Exchangers. Source: [13]

C. STATE-OF-THE-ART HEAT EXCHANGER DESIGN

In thermodynamic cycles such as the Rankine Cycle, the removal of waste heat occurs at the condenser, as illustrated by Q_2 in Figure 5. The growing power demand leads to an increase in heat production, which in turn poses a challenge for the current state-of-the-art heat exchanger designs. State-of-the-art heat exchangers include shell-and-tube, plate-and-frame, plate-fin, and printed circuit heat exchanger [14].

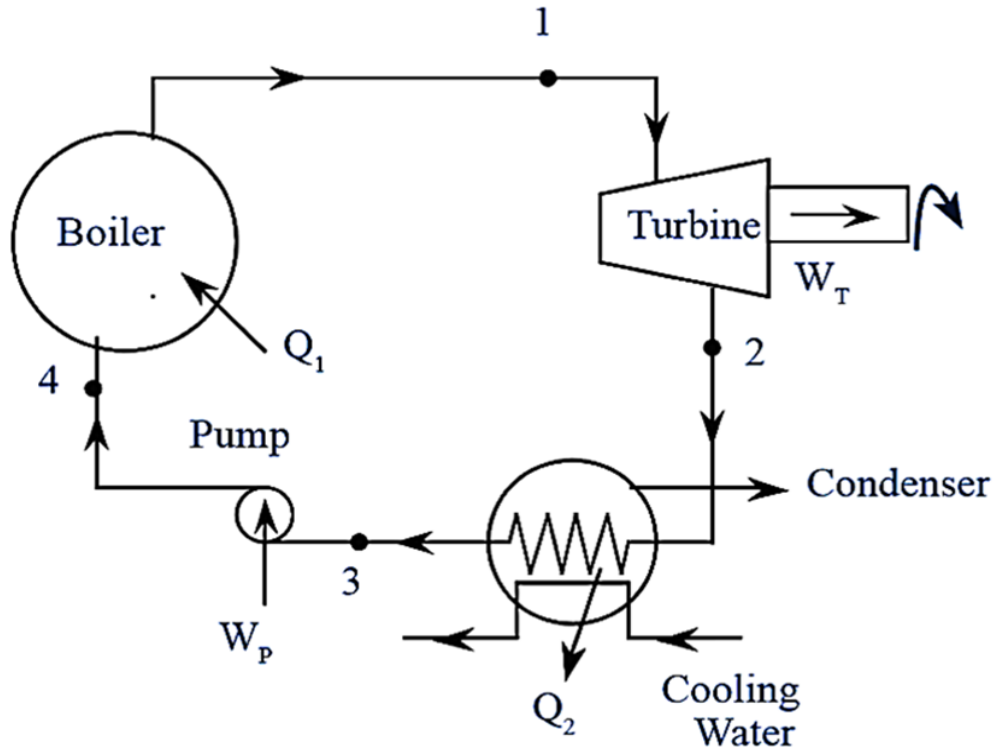
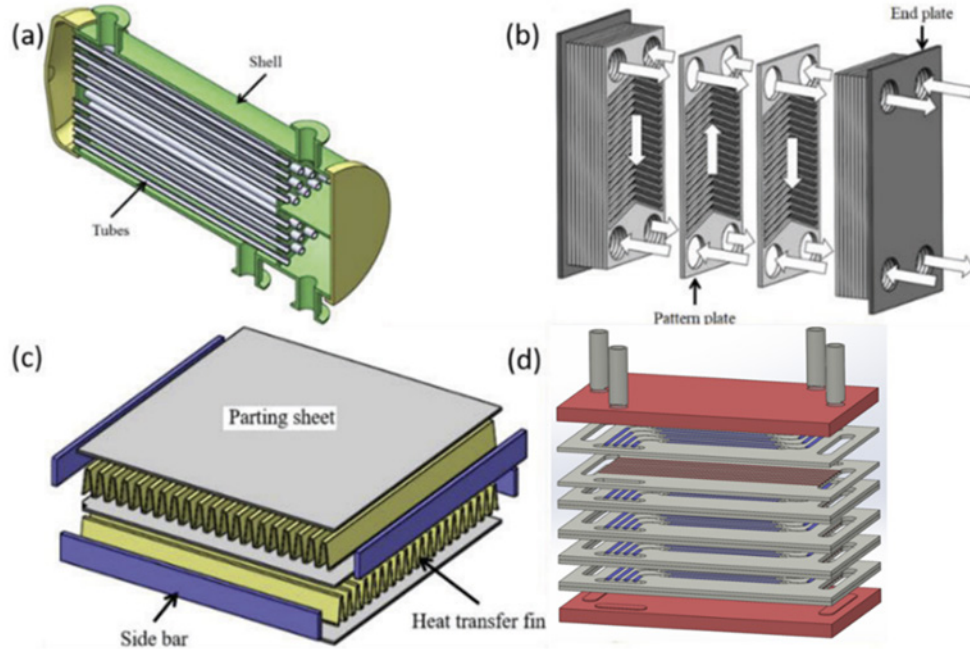


Figure 5. Schematic of Rankine Cycle. Source: [15]

Shell-and-tube heat exchangers, depicted in Figure 6, are the most common type used across all industries. These heat exchangers are comprised of tubes containing one fluid, which are placed inside a shell filled with another fluid. The key advantage of shell-and-tube heat exchangers is their ability to operate under extreme conditions, including high pressures and temperatures. However, due to their low surface area to volume ratio, these heat exchangers tend to be bulky and heavy. Plate-and-frame heat exchangers utilize rigid metal frames to support corrugated, fin acting, plates in between channels. Instead of a rigid frame, plate-fin heat exchangers use sheets to enclose the liquid and with fins inside. While these designs offer a greater surface area to volume ratio, enabling enhanced heat transfer, they are limited in their capacity to handle high temperatures and pressures. They are also prone to significant pressure drops and fouling issues. Printed circuit heat exchangers (PCHE) consist of plates that are etched with grooves to direct flow. PCHEs can operate at high temperatures; however, the fine grooves contribute to high pressure

drops. The process of etching each plate can be quite expensive [14]. The advantages and disadvantages of these four heat exchangers are summarized in Table 1.



(a) Shell-and-tube Heat Exchanger, (b) plate-and-frame heat exchanger, (c) plate-fin heat exchanger, (d) printed circuit heat exchanger

Figure 6. State-of-the-Art Heat Exchanger Design. Adapted from [14]

Table 1. Limiting Parameters for Heat Exchangers. Adapted from [14]

	Area/V (m^2/m^3)	T_{max} ($^{\circ}\text{C}$)	Pressure (bar)
Shell-and-tube heat exchanger	50-100	1100	1000
Plate-and-frame heat exchanger	120-660	815	200
Plate-fin heat exchanger	800-1500	800	200
Printed circuit heat exchanger	200-2500	980	900

D. ADDITIVELY MANUFACTURING HEAT EXCHANGERS

Additive manufacturing (AM) is an emerging technology that has been used for rapid prototyping since the 1980s. Despite its extensive use by doctors, students, artists, and market researchers, the adoption of AM in the manufacturing sector was limited until 2012 [16]. A goal of the implementation of AM is to reduce production time as well as cost [16]. A concern for AM is the presence of residual stresses and anisotropic material properties. These factors can weaken the structural integrity of as-printed objects, making them unsuitable for production purposes. It was not until 2021 when specific standards for 3D printed products began to emerge [17]. Efforts to achieve mechanical properties in AM metal parts that are comparable to those produced through traditional manufacturing methods have proven to be highly costly, which contradicts the original purpose of the implementation of AM.

Common materials used in heat exchangers include titanium, stainless steel, carbon, and brass [18]. Titanium is corrosion resistant, and most applicable for seawater heat exchangers. The titanium alloy commonly found in AM is Ti-6Al-4V, which can be printed by powder-bed-fusion (PBF). PBF is a process in which powder is sintered or fused by the application of any form of a laser. The position of the part being printed is commonly fixed in the x-y direction and a piston drives a build plate in the z-direction [16]. The PBF process is shown in Figure 7. PBF provides an avenue of producing complex geometries with high accuracy and surface finish [19].

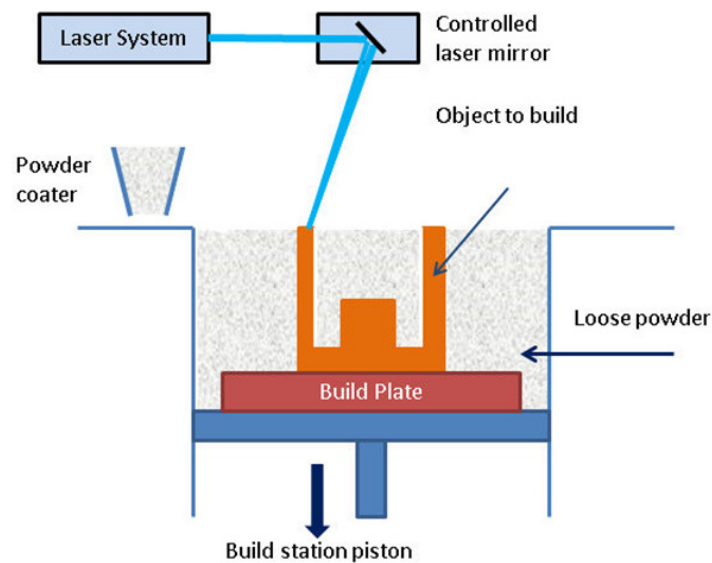


Figure 7. Schematic of Powder Bed Fusion. Source: [20]

AM is a rapidly advancing field, but it also has its limitations. Part accuracy is influenced by the height of each layer of powder, which is dictated by the powder size. Finer powder results in higher printing costs. Powder manufacturing includes solid-state reduction, atomization, electrolysis, and chemical processes. Electrolysis creates the smallest and most uniform powder of any method, while chemical processes are the most common, it is unreliable for particle size and quality [21]. Another limitation of PBF is the build size and rate. Part size is constrained to the size of the machine, with the current industry standard being 250mm x 250mm x 250mm [22]. Printing larger parts can take days, and typical print rates range between 10–50 cc/hr for most materials. Unlike some other AM processes, PBF does not allow for additional features on existing parts [23]. A major limitation for AM using the PBF method is the need for support structures. Every layer is fused layer by layer, but the powder needs to be supported from underneath to be fused properly. Support structures are needed for overhanging features that are less than 45 degrees. Figure 8 demonstrates the result of not adding support structures.

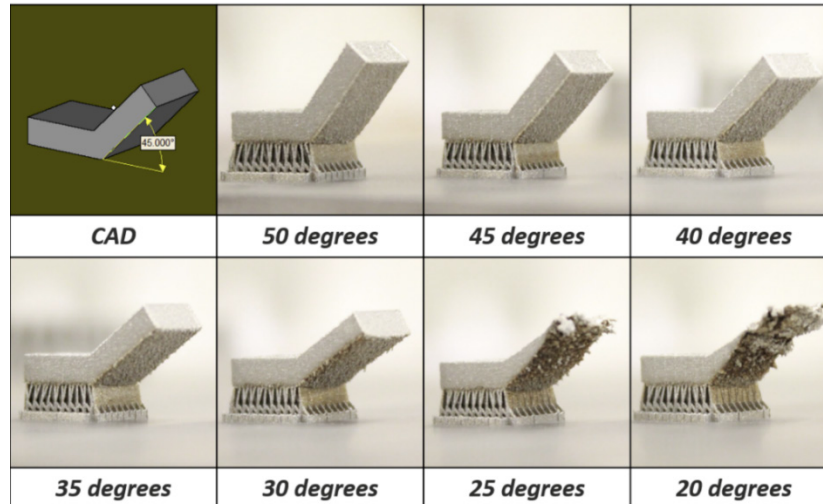


Figure 8. Importance of Support Structures. Source: [24]

E. TRIPLY PERIODIC MINIMAL SURFACE

As technology advances, the threshold for performance of heat exchangers has increased dramatically. AM has allowed for designing lightweight and compact heat exchangers. Triply periodic minimal surfaces (TPMS) have been found in nature like skeletons and structures. Figure 9 shows how the wing of the butterfly is constructed by a lattice of gyroids, and the shell of weevil L. Augustus beetles consist of a lattice with the Schwartz-D architecture [25]. Weihong Li, Guopeng Yu and Zhibin Yu researched have explored the potential benefits of incorporating TPMS into heat exchanger design [25].

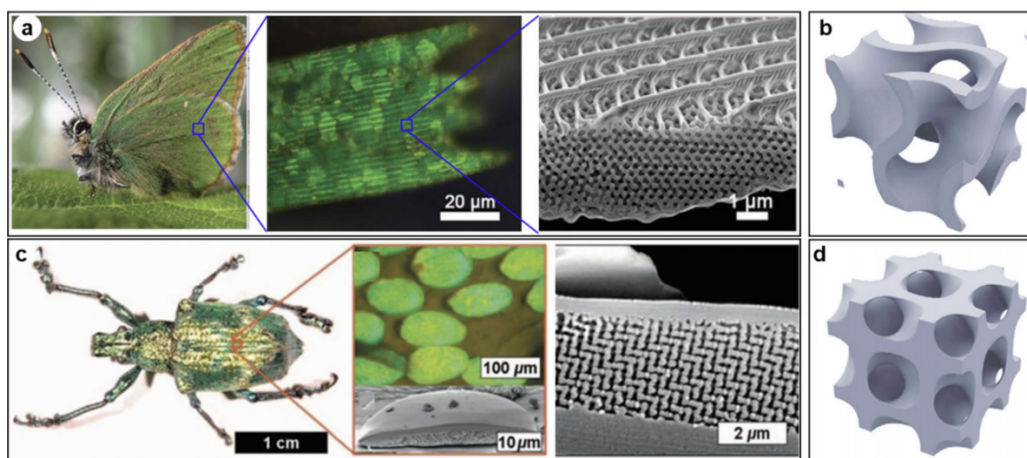


Figure 9. Bioinspired TPMS Unit-Cells. Source: [25]

TPMS are complex mathematical surfaces exhibiting periodic patterns in three dimensions. These surfaces have zero mean curvature, resulting in smooth curvatures without sharp corners or edges. This makes TPMS ideal for heat exchangers, as they offer a high surface area to volume ratio, leading to lightweight and high-performance designs. TPMS topologies can be divided into two non-intersecting yet intertwining domains, which can be utilized for two-fluid heat exchangers. The intertwining channels with their absence of sharp edges enhance fluid dynamics, promoting gradual enlargement or contraction of the fluid and reducing pressure drops. The curved nature of the surfaces promotes more turbulent flow, which increases the heat transfer rate [25]. Turbulence can be characterized through computational fluid dynamics (CFD) using

$$TI = \frac{\sqrt{\bar{u}'^2}}{u_{avg}} \quad (1.6)$$

where $\bar{u}'$ is the average velocity functions and u_{avg} is the average flow velocity.

Slaughter was one of the first who took advantage of the properties of TPMS by introducing these surfaces in heat transfer devices in 2003 [26]. AM has made it accessible to create these surfaces as they are all self-supporting requiring no supports to print. TPMSs are mathematically derived equations, such as

$$Gyroid = \sin(x) \cos(y) + \sin(y) \cos(z) + \sin(z) \cos(x), \quad (1.7)$$

which is the equation for a gyroid TPMS.

The unit cell that (1.6) produces is shown in Figure 10, and the software used to create the unit cell and the full model is Ntopology. Ntopology is a new software created to optimize designing for AM. There are many capabilities of Ntopology including designing TPMS lattices.

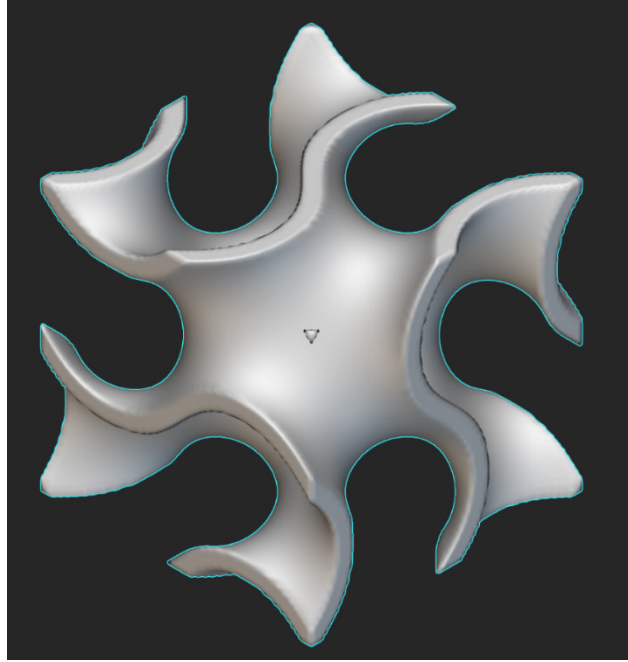


Figure 10. Gyroid TPMS Unit Cell in Ntopology

A dimensionless number used to categorize the effects of viscosity on fluid systems is the Reynolds number [27]

$$\text{Re} = \frac{\rho D_h u_{inlet}}{\mu}. \quad (1.8)$$

Equation (1.8) calculates the Reynolds number of the flow, where ρ is the fluid's density, and μ is the dynamic viscosity. Because TPMSs are not common shapes, to determine the hydraulic diameter

$$D_h = \frac{4V}{A_s} \quad (1.9)$$

the volume of the fluid domain, V , and the wetted heat transfer surface area, A_s are used. The hydraulic diameter resembles any cross-section channel flow like flow through a pipe. It is an effective diameter that closely relates what the flow would look like going through

a circular pipe for non-circular pipes [27]. With the hydraulic diameter heat transfer properties can be calculated for the system. Pressure drop is calculated by

$$\Delta P = \frac{2f\rho u_{avg}^2 L}{D_h} \quad (1.10)$$

where f is the friction factor, and L is the length from the entrance, and ΔP is the plane-averaged pressure difference at the inlet and outlet [25]. Whereas the friction factor through pipes is nominally found from a Moody Diagram, Figure 11.

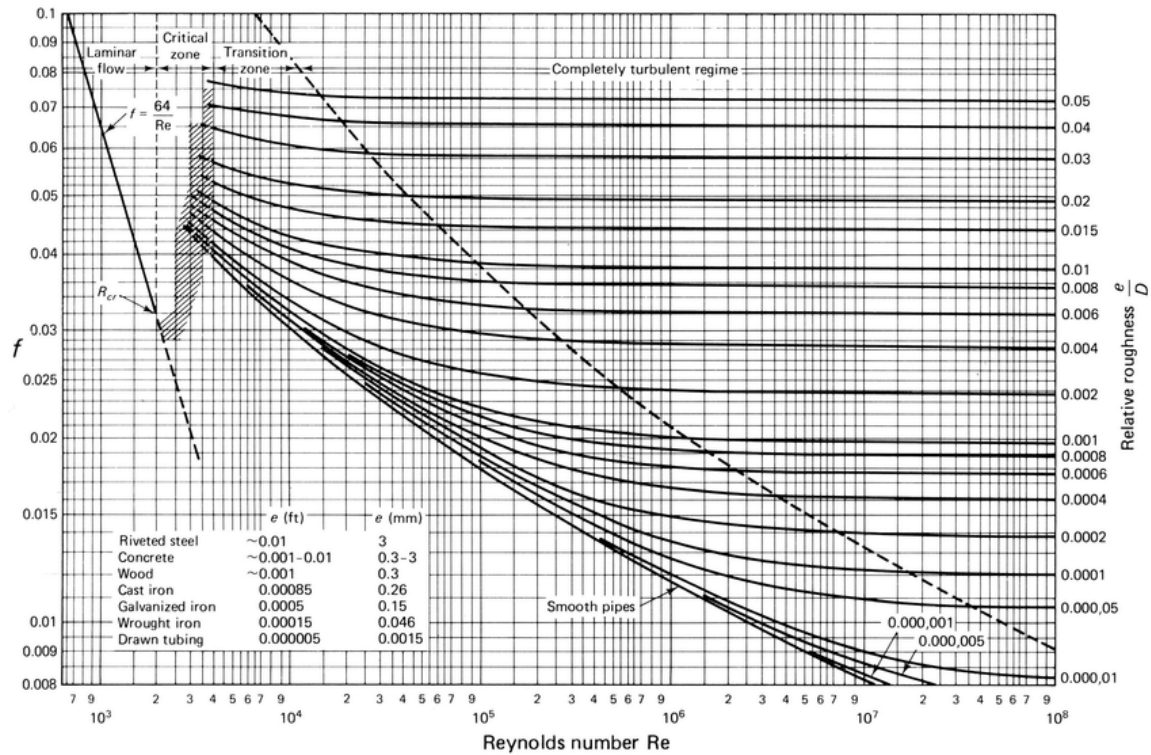


Figure 11. Moody Diagram. Source: [28]

One previous study investigated turbulent flow through a TPMS. The working fluid was carbon dioxide. The result of this study is presented in Figure 12 and Figure 13. PCHE was mentioned earlier part of the current state-of-the-art heat exchangers. PCHE has shown promise to outperform shell and tube heat exchangers. Figure 12 and Figure 13 show that TPMS geometries outperform PCHE.

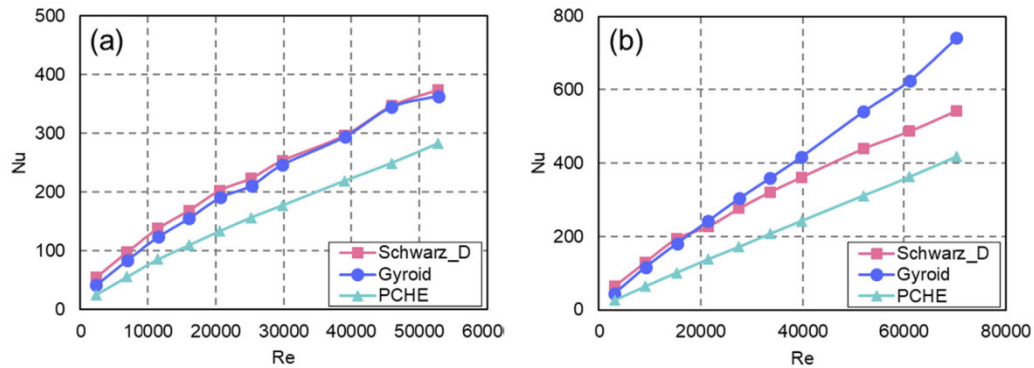


Figure 12. Area Averaged Nu Number of (a) Hot Channel (b) Cold Channel. Source: [25]

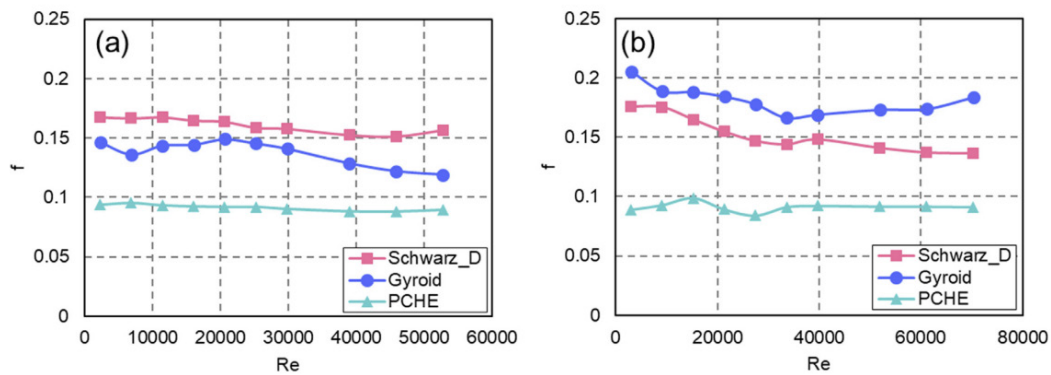


Figure 13. Area Averaged Friction Factor in (a) Hot Channel (b) Cold Channel. Source: [25]

W. Li conducted a computational analysis of turbulent CO₂ flow through TPMS. Advanced Engineering Solutions along with the U.S. Army investigated utilizing gyroid lattices in a gearbox oil heat exchanger for a helicopter, which could only be created through AM [29]. This gyroid heat exchanger resulted in four times the cooling power and half the size compared to conventional designs. A visual representation of the gyroid heat exchanger is in Figure 14. The heat exchanger was designed for laminar flow, with fuel and oil as the working fluid.

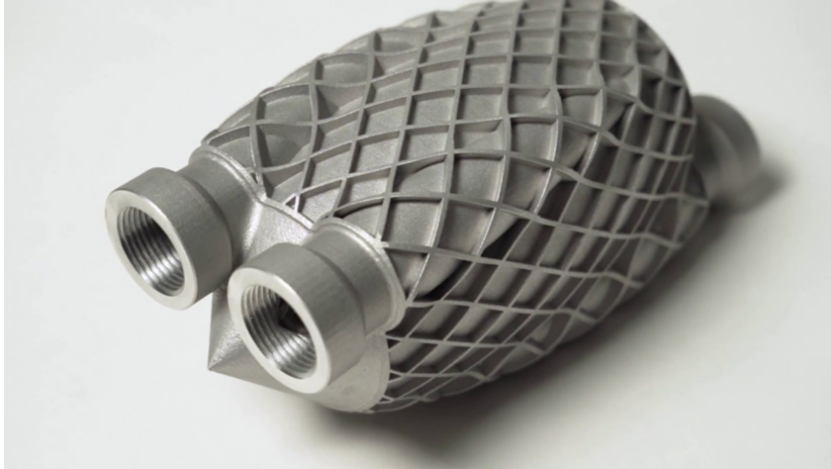


Figure 14. Additively Manufactured TPMS Heat Exchanger. Source: [7]

F. COST ANALYSIS

When it comes to manufacturing industrial parts, a big concern is cost. The cost analysis of a shell-and-tube heat exchanger will go step by step to determine the final cost. A flow chart is shown in Figure 15 to illustrate the manufacturing process of a shell-and-tube heat exchanger. The cost break-down of the shell side depends on the material, diameter, number of passes and length. Additionally, the complexity of the heat exchanger and the number of baffles also impact the cost, and these weighting factors are shown in tables 2–5. Table 2 is the factor for materials, Table 3 is the factor for tubing, Table 4 is the factor for complexity of construction, and Table 5 is the factor for baffle orientation. All weighting cost factors are applied to the basic heat exchanger cost with the following parameters: Type AES TEMA R all-steel A214 19.05-mm tubes, 25.4-mm square pitch, two-pass. The overall cost relative to the size of the exchanger is shown in Figure 16.

The cost of the shell side of a heat exchanger is complicated, the main cost driver for a shell-and-tube heat exchanger is the tube side shown by the calculations performed in the Heat Exchanger Design Handbook [30].

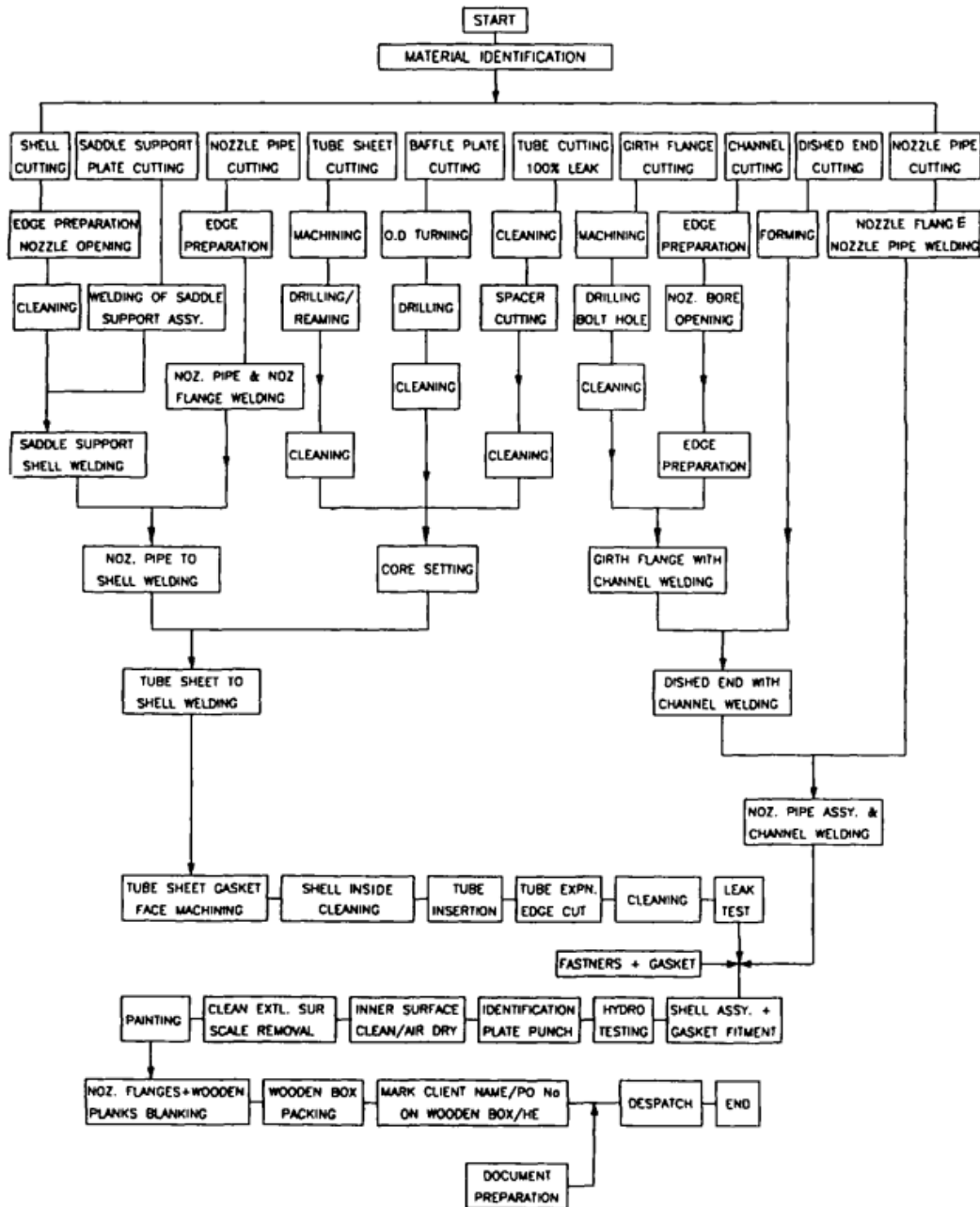


Figure 15. Flow Chart of Manufacturing Process of a Shell-and-Tube Heat Exchanger. Source: [31]

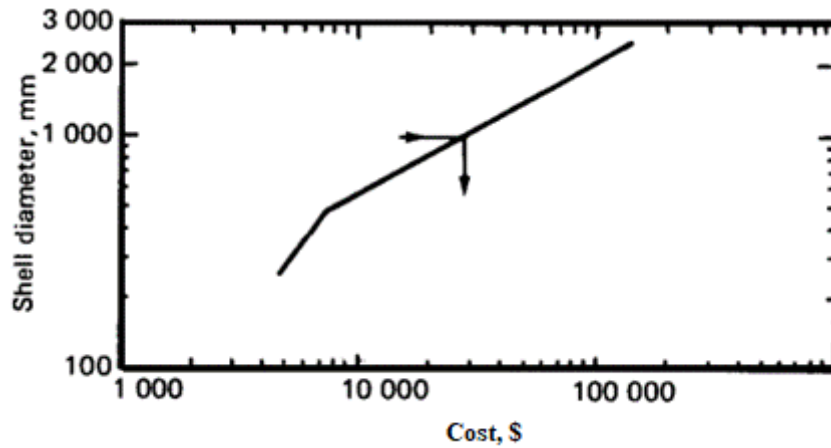


Figure 16. Cost of Shell-and-tube Heat Exchanger in Relation to Shell Diameter. Source: [30]

Table 2. Cost of Materials. Adapted from [31]

	Shell ID		
Material	560mm	560-762mm	More than 762mm
Carbon Steel	1	1	1
18/8 shell and bundle	3.4	4	4.5
18/8 bundle CS shell	3	3.5	4
Aluminum Bundle	1.2	1.3	1.35
Copper Tubes	1.2	1.3	1.4
70/30 Copper-nickel tubes	1.4	1.55	1.7
90/10 Copper-nickel tubes	1.4	1.5	1.65
304 T/S CS Shell	2.2	2.2	2.4
304 Tubes CS remainder	1.65	1.7	1.7
5CR T/S CS Shell	1.21	1.25	1.31
C-1/2 Mo T/S CS Shell	1.2	1.25	1.31
5CR S/S CS Shell	1.4	1.5	1.8
Based on 6.096m long shell			

Table 3. Cost For Tubing. Adapted from [30]

Tube	Price, \$/m	Tube gauge, BWG
A214 steel	1.12	14
A179 steel	1.43	14
Aluminum Brass	3.2	16
A213/304 S/steel	2.92	16
A213/321 S/steel	3.87	16
A213/316 S/steel	4.76	16
70/30 Cu-Ni	4.38	16
90/10 Cu-Ni	3.93	16

Table 4. Factors for Type of Construction. Adapted from [30]

Type	Factor Comment
Floating-head AES	1
Pull-through floating-head AET	1.045
Fixed tube sheet BEM	0.9
U-tube BEU	0.85

Table 5. Factor for Baffle Orientation. Adapted from [30]

Baffle Ratio = $\frac{\text{baffle pitch}}{\text{shell diam.}}$	Factor
0.25 or greater of shell diam.	1
0.2 of shell diam.	1.01
0.167 of shell diam.	1.02
0.143 of shell diam.	1.03
0.1 of shell diam.	1.05

Compared to the well-established manufacturing processes for traditional methods, the cost analysis for AM production is relatively new. As mentioned before, one limitation of AM is its size constraint. The biggest laser powder bed fusion printer today is the General Electric Concept Laser X Line 2000R, shown in Figure 17, that has a build volume

of 800x400x500 mm (x,y,z) [32]. The starting price for this machine is \$1,000,000 without including the price of the powder. The price of Ti-6Al-4V is \$250/kg [33]. The price of powder depends on the quality of powder which is dependent on the methods mentioned earlier of producing the powder, electrolysis being the most expensive but creates the most quality parts.



Figure 17. Concept Laser X Line 2000R Printer. Source: [32]

As mentioned earlier, Ti-6Al-4V produced through AM requires post-processing to achieve similar physical properties to traditionally manufactured parts. Post-processing methods may include heat treatment, hot isostatic pressing, machining and surface treatments. Heat treatment can vary from \$500-2000 depending on material and part size. While the other post processing methods are optional [34].

Figure 18 provides a cost breakdown for manufacturing an AM part per pound, including pre- and post-processing costs, as well as machine costs per hour. The cost breakdown was obtained using the EOS M290 machine, which has a build rate of $4.2 \text{ mm}^3/\text{sec}$. Inconel 718 powder, priced at \$120 per kilogram, was used as the material. The operating cost of the EOS machine is \$50/hr. The graph illustrates a linear relationship

between cost and part size. The same principle can be applied to any material, as the operating cost and post-processing cost do not fluctuate as much as the cost of powder for each specific material [35].

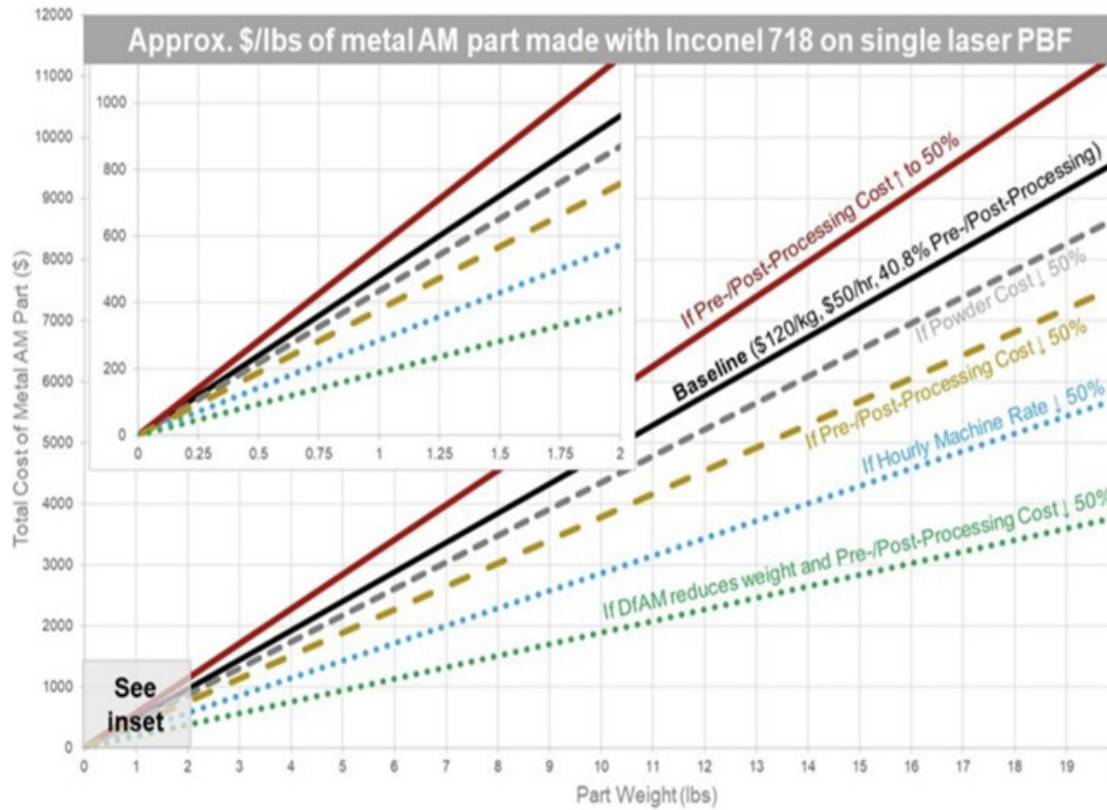


Figure 18. Estimate Cost \$/lb of AM Parts. Source: [35]

G. COMPUTATIONAL FLUID DYNAMICS

Commercial software package Ntopology is used to create TPMS surfaces, and ANSYS Fluent 2022 R1 is used to solve the compressible and steady form of continuity, momentum. The governing equation for continuity is

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (1.11)$$

and momentum is

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla(\rho \vec{v} \vec{v}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (1.12)$$

where $\bar{\bar{\tau}}$ is the stress tensor

$$\bar{\bar{\tau}} = \mu[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \vec{v} I]. \quad (1.13)$$

Where $\vec{v}$ is the velocity vector, $\rho \vec{g}$ is gravitational body force, $\vec{F}$ is the external forces and I is unit tensor. When dealing with turbulent flow, additional equations are employed to solve the characteristics of the turbulence. Fluent utilizes Reynolds-Averaged Navier-Stokes (RANS) turbulence models. These models simplify the original seven-equation Navier-Stokes equation into two transport equations. Among the various RANS models, the most commonly used ones are the two-equation models. These models involve deriving two transport equations to represent two turbulence properties: kinetic turbulence energy (k), dissipation of turbulence kinetic energy (ϵ), or the specific dissipation rate (ω) [36].

The $k - \epsilon$ approach is the most widely used turbulent model as it simulates wall-bounded flow very well with limited adverse pressure gradient. Whereas $k - \omega$ was specifically developed to enhance the accuracy of flow prediction near the wall. Both models, $k - \epsilon$ and $k - \omega$, possess their own strength and weaknesses. To combine the advantages of these two models, the Shear Stress Transport (SST) model was introduced. The SST model smoothly transitions from $k - \omega$ near the wall to $k - \epsilon$ model in the far field flow [37]. In the simulation of turbulence effects using Fluent, the $k - \omega$ SST model is used, with a dimensionless wall distance (y^+) of less than 1 to ensure proper wall resolution. From the ANSYS Fluent Theory Guide the $k - \omega$ SST turbulence model is a blended function that combines the robust and accurate formulation of the $k - \omega$ model near the wall with the freestream independence of the $k - \epsilon$ model in the far field [38]. The $k - \omega$ SST model incorporates various refinements, such as multiplying and adding the $k - \epsilon$ and $k - \omega$ models by a blending function, introducing a cross-diffusion derivative term in the ω equation, and employing a different definition for turbulent viscosity [38]. Fluent uses (1.14)–(1.18) to solve the $k - \omega$ SST model where the kinetic energy

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}(\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k + S_k, \quad (1.14)$$

specific dissipation rate

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j}(\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega, \quad (1.15)$$

the effective diffusivity

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad (1.16)$$

and

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega}, \quad (1.17)$$

and turbulent viscosity

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max[\frac{1}{a^*}, \frac{SF_2}{a_1 \omega}]} \quad (1.18)$$

are all solved. Where G_k is the generation of kinetic energy due to mean velocity gradients. G_ω is the generation of ω . Γ_k and Γ_ω are the effective diffusivity of k and ω . Y_k and Y_ω represent the dissipation of k and ω due to turbulence. The S_k and S_ω are the source terms defined by the user in the initialization. Lastly, D_ω represents the cross-diffusion term. For further explanation of how each term is obtained refer to the ANSYS Fluent theory Guide [38].

CFD is not only useful for predicting flow behaviors, but also for solving conjugate heat transfer problems. Fluent does this by resolving the energy equation

$$\frac{\partial}{\partial t}(\rho E) + \nabla(\vec{v}(\rho E + p)) = \nabla(k_t \nabla T - \sum_j h_j \vec{J}_j + \overline{(\tau_{eff} \vec{v})}) + S_h \quad (1.19)$$

where E is evaluated by

$$E = h - \frac{p}{\rho} + \frac{u^2}{2}. \quad (1.20)$$

Where T is temperature, $\vec{J}_j$ is the diffusion flux of species j . For further explanation see the Fluent Theory Guide [38]. The energy equation is used to solve the heat transfer coefficient, which fluent defines the surface heat transfer coefficient as

$$h_s = \frac{Q}{T_{wall} - T_{ref}}. \quad (1.21)$$

Fluent has the option to model roughness on walls, which is particularly relevant for additively manufactured parts that inherently exhibit higher surface roughness. In the case of as-built PBF Ti-4Al-6V, the surface roughness ranges from 7.867–29.94 μm , whereas traditional Ti-4Al-6V ranges from 0.32–0.874 μm [19], [39]. To model roughness in Fluent, the roughness height, K_s and roughness constant C_s are used. For additively manufactured parts K_s is found using

$$K_s = 18R_a - 0.005R_a. \quad (1.22)$$

Even though there is no specific guidance for C_s , Fluent models a C_s of 0.5 as perfectly uniform and 1 is non-uniform along the surface.

In W. Li computational analysis of turbulent carbon dioxide (CO₂) flow through TPMS, the computational method employed was SST [25]. According to W. Li, SST is known for its superior accuracy in capturing flow characteristics near walls compared to other turbulence models such as the $k - \epsilon$ models. The mesh used in the analysis was fully wall-resolved, as indicated by the fact that all y -plus, y^+ , values were all below one. Periodic boundary conditions were applied to all sides of the fluid and a coupled boundary condition was imposed at the solid-fluid interface. Inflation layers were used to better resolve the flow near the wall, consisting of eleven layers where the first layer thickness

was 2.5×10^{-4} mm. A mesh sensitivity study was conducted to ensure the mesh had no significant influence on the results as well as to save computational power.

THIS PAGE INTENTIONALLY LEFT BLANK

III. RESEARCH DESIGN

A. DESIGN PARAMETERS AND METHODS

The objective of this study is to create an AM heat exchanger to replace the conventional shell-and-tube heat exchanger. The initial stage in the design process involves determining the crucial parameters that restrict the design space. The key parameters of the heat exchanger that is in question of improving upon this research are in Table 6.

Table 6. Design Parameters

	Fresh Water Side	Sea-Water Side
Temperature Inlet Range	82C	-2C -> 32C
Maximum Pressure	1MPa	10MPa
Flow Rates	$0.38 \text{ m}^3/\text{s}$	$0.23, 0.19, 0.13 \text{ m}^3/\text{s}$
Pressure Drop	$10\text{kPa @ } 0.145 \text{ m}^3/\text{s}$	$6.9\text{kPa @ } 0.063 \text{ m}^3/\text{s}$
Cost	\$5.5 Million total for two heat exchangers	
Maximum Size (LxWxH)	3 m x 0.75 m x 0.75 m	
Maximum Size for Easy Removal	1.8m x 0.75m	
Heat Removal	5.861MW	

These parameters were considered when performing initial calculations. When performing the calculations, the following water properties in Table 7 and metal properties in Table 8 were used.

Table 7. Water Properties Used in Calculations. Adapted from [40]

	Fresh Water	Seawater
Density (kg/m^3)	997	1021
Thermal Conductivity ($\frac{W}{m^2 \cdot K}$)	0.6	0.6181
Dynamic Viscosity ($\frac{Ns}{m^2}$)	3.453e-04	8.2491e-04
Specific Heat Capacity ($\frac{J}{kg \cdot K}$)	4187	4005.8

Table 8. Metal Properties Used in Calculations. Adapted from [41, [42]

	As-printed Ti-4Al-6V
Density ($\frac{kg}{m^3}$)	4420
Specific Heat Capacity (J/kg-K)	556.25
Thermal Conductivity (W/m-k)	5.62
Surface Roughness (μm) Default Print	10.5

B. INITIAL CALCULATIONS

The water properties from Table 7 were found on ThermExcel [43] and seawater properties were found using code from MIT [40]. The properties were implemented in Appendix A. Matlab code for Initial Calculations. The initial calculations produced results shown in Table 9 and Table 10. These calculations were conducted to establish a baseline understanding of the expected outcomes from the CFD simulations. The model calculations generated the results displayed in Table 9. These results served as the basis for designing a complete heat exchanger using TPMS, considering the limiting pressure and the required heat exchange area, as shown in Table 10. Entrance and exit length effects were not taken into consideration when performing these initial calculations. When dealing with a large-scale model the inlet and exit conditions can change the heat transfer coefficient and pressure drop.

Table 9. Initial Calculations for Model Using TPMS

	Fresh Water Side	Seawater Side
Heat Transfer Coefficient $\left(\frac{W}{m^2K}\right)$	1.602e+05	1.452e+05
Inlet Velocity (m/s)	0.1426	0.2578
Heat Transfer Surface Area (m^2)	0.1474	0.1474
Hydraulic Diameter $D_h(m)$	9.362e-04	9.362e-04
Reynolds Number (Re)	30880.8	23928.3

Table 10. Initial Calculations for TPMS and Shell-and-Tube Heat Exchanger

	Fresh Water Side	Seawater Side
Heat Transfer Surface Area (m^2)	0.8408	0.9275
Length of Heat Exchanger (m) (Pressure Drop Driven)	1.039	0.3833
Length of Heat Exchanger (m) (Heat Exchange Area Driven)	0.135	0.15
Number of Unit Cells (For limiting Length)	1.39e+04	5.11e+03

THIS PAGE INTENTIONALLY LEFT BLANK

IV. UNIT CELL

The unit cell analysis was chosen because it was computationally less demanding. Research of turbulent flow through TPMS is so novel that obtaining a baseline characterization of flow properties and heat transfer properties through a TPMS is vital in the initial design for the TPMS heat exchanger.

The workflow of this unit cell analysis is shown in Figure 21. Starting with the unit cell created in Ntopology, Figure 10, it was expanded to a 5x5 lattice. The unit cell was a 15 mm cube, creating a 5x5 lattice that made it 75 mm wide. The ultimate objective was to fabricate the TPMS lattice using the EOS M290 PBF machine, which has a maximum build size limit of 250 mm width [44]. The 5x5 of the 15 mm unit cell allowed for inlet and outlet tubes to be added to the large-scale design and still fit within the size constraints of the EOS M290 machine.

Moving from Ntopology to SolidWorks, the lattice structure was exported as a .step file. It is important to note that the process of creating a CAD body from the implicit body in Ntopology was still in beta and served as a means to quickly wrap the implicit body with minimal faces for export to SolidWorks. However, the faces generated by Ntopology didn't accurately represent the intended geometry. In Figure 19, it can be observed that not all edges and faces were well defined. Having well-defined edges and faces is crucial for creating accurate boundary conditions in Fluent, the software used for the subsequent simulation. To resolve this issue, SolidWorks was employed to perform an extruded cut operation on the 5x5 lattice. This operation resulted in a clearly defined single 15 mm unit cell with well-defined faces, as shown in Figure 20. The presence of these well-defined faces was essential for accurately defining the boundaries in Fluent, enabling the subsequent simulation process.

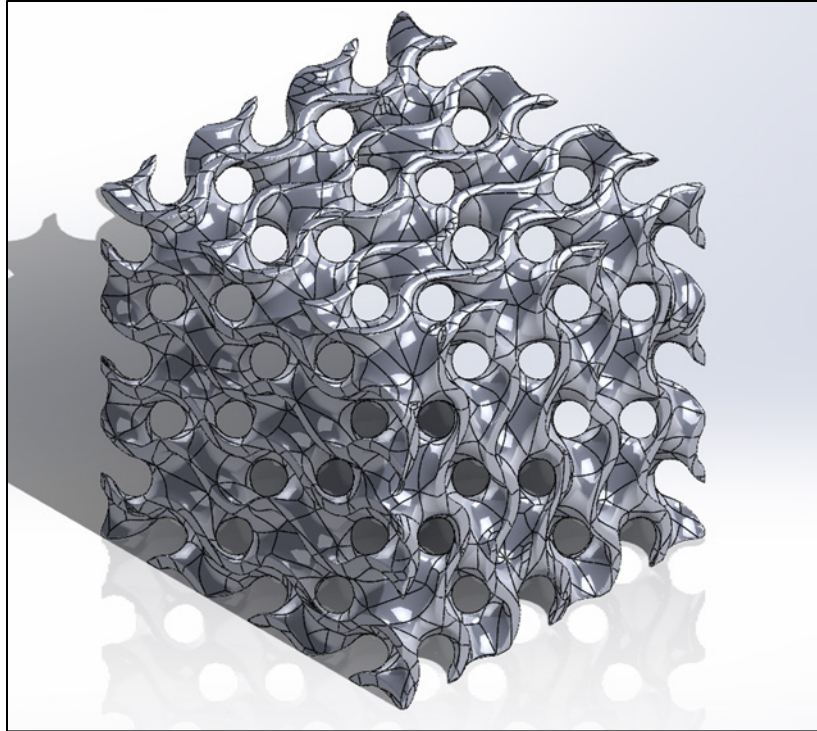


Figure 19. Beta Function of Producing CAD Bodies From Implicit Bodies in Ntopology

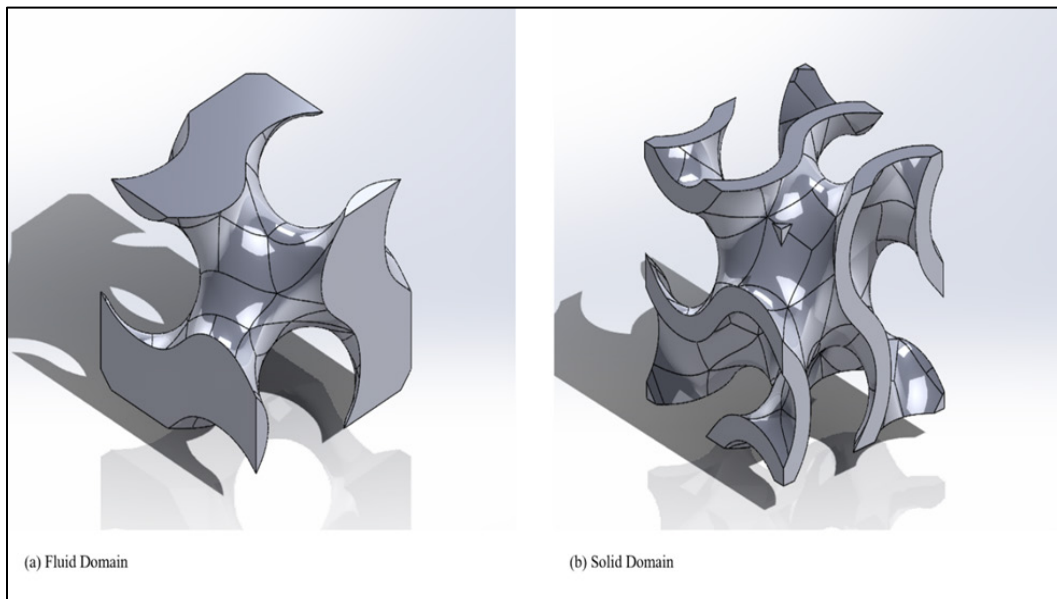


Figure 20. SolidWorks of Unit Cell



Figure 21. Workflow for Unit-Cell Study

A. UNIT-CELL MESH SENSITIVITY STUDY

Meshing plays a crucial role in CFD simulations as it impacts both accuracy and computational time. A mesh sensitivity study aims to determine the smallest mesh size that yields independent results, meaning that the mesh has no influence on the simulation outcomes [45].

The first step in the mesh sensitivity study was to find the initial layer height of the inflation layers. This parameter will affect the y^+ values, which determine whether or not the model is wall functioning or wall resolved. The $k - \omega SST$ turbulence was chosen to solve this problem, because there is expected to be vortex shedding, and from literature this model is the best at resolving flow near the wall. Previous studies with turbulent flow through TPMS have also used the $k - \omega SST$ model [25].

To ensure accuracy of the $k - \omega SST$ model, it is necessary to have y^+ values less than one. Y^+ values less than one ensure that the model is wall-resolved rather than wall-functioning. A wall-resolved model provides more accurate predictions of flow near the walls compared to wall-functioning models. Since the flow through TPMS exhibits unsteadiness due to vortex shedding, it is necessary to accurately capture this unsteadiness by using a wall-resolved system. The most significant factor influencing y^+ values is the initial layer height. For circular pipes, there are well-defined relationships between initial layer height and y^+ values. However, the flow through TPMS is different from regular circular pipes, so determining the appropriate initial layer height required trial and error. Eventually, an initial layer height of 2.5×10^{-4} m was found to be suitable. Based on this initial layer height, three meshes were produced: coarse, medium, and fine. The parameters

of the three meshes are found in Table 11. Curvature was turned on to capture the natural shape of the TPMS and was set to 12 degrees. Proximity was turned on to create finer meshes closer to boundaries to capture the fluid characteristics more accurately at the outlet of the unit cell. The default proximity sizing was turned on which is set to the global minimum size. The coarse mesh produced 90% of the y^+ values below one, shown in Figure 22. This confirms that the chosen initial layer height successfully achieves a wall-resolved model.

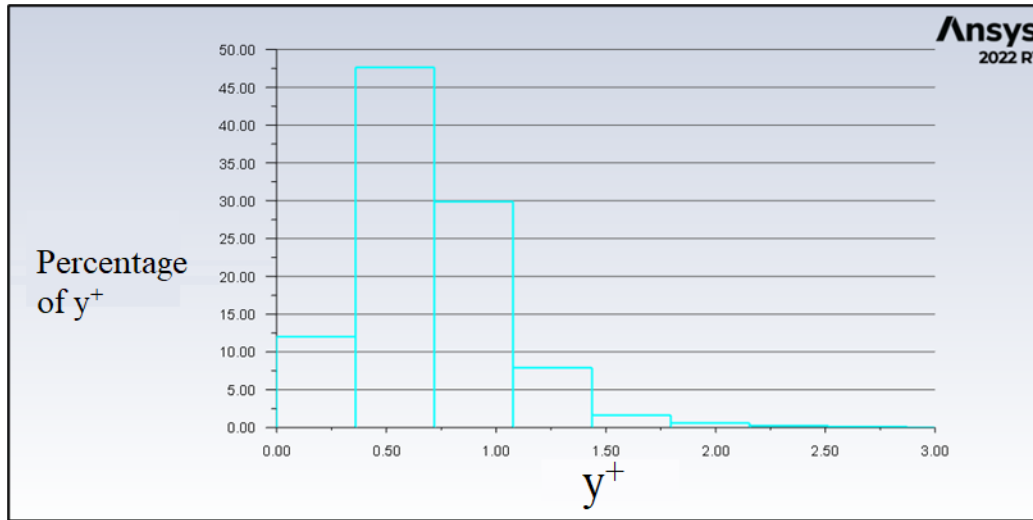


Figure 22. Histogram of Y^+ Values for Coarse Mesh

Table 11. Mesh Sensitivity Study

	Mesh #1	Mesh #2	Mesh #3
Bulk Mesh Size (m)	5e-3	5e-4	1e-4
Initial Layer Height (m)	5e-4	5e-4	5e-4
Number of Inflation Layers	14	14	14
Face Sizing (m)	1e-3	1e-4	5e-5
Number of Elements	300,000	3,000,000	10,000,000
Shear Stress (Pa)	5.21	5.61	5.06

The Fluent setup shown in Figure 23 consist of all periodic faces. Periodic boundary conditions are implemented when there is an expected repeating pattern of flow [38]. With periodic boundary conditions, the outlet conditions are fed back into the inlet along the same flow axis, effectively recycling the flow within the system. It is assumed that the flow through one unit cell can represent the flow through multiple unit cells. The periodic boundary conditions are applied to all faces to ensure fully developed flow within the 15 mm unit cell. The Fluent setup in Figure 23, illustrates the periodic boundary conditions by the yellow faces. It is important to note that the setup only consists of one fluid domain because the full unit cell with both fluid domains and the solid was causing errors with defining the periodic boundaries. The setup of the two fluid domain setup can be seen in Appendix B: Two Fluid Domain Setup. The periodic boundaries only worked when every boundary was defined with the flow direction from the low face to high face. One fluid is sufficient to study because the fluid domains are identical therefore the results found from one mesh sensitivity study on one fluid can be applied to the other fluid.

Fluent provides two options for driving periodic boundary conditions: pressure gradient and mass flow. Both methods were tested, and it was found that using mass flow as the driving parameter for the periodic boundary conditions produced better results in terms of convergence and flow behavior.

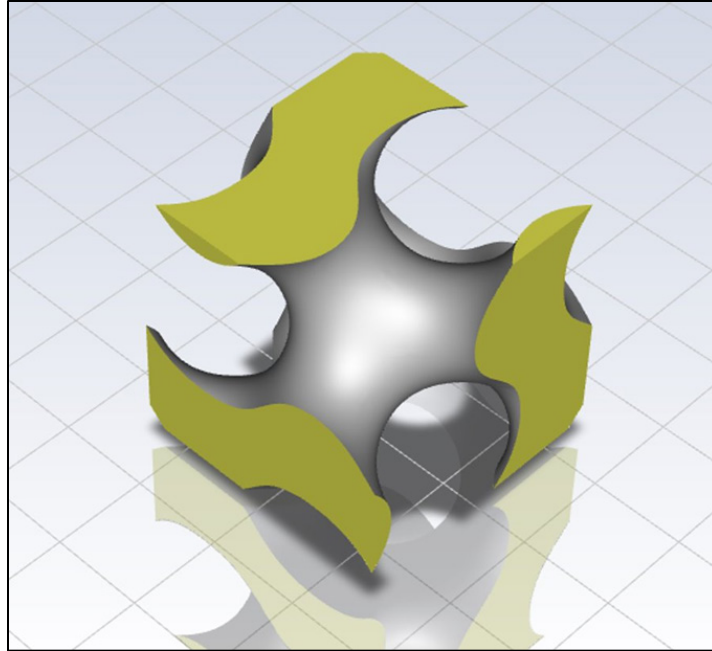


Figure 23. CFD Periodic Setup

In the remaining CFD runs the mass flow driven periodic condition was used. The mass flow rate was determined based on the velocity obtained from the initial calculations in Appendix A. Matlab code for Initial Calculations, 0.025 kg/s. The results from the individual fluid studies will produce values such as heat transfer coefficient and pressure drop, that will help model a full-scale heat exchanger. By using a unit cell, it reduces computing power and allows for more runs to explore trials and tribulations.

All three meshes presented convergence based on the residuals output by fluent and shown in Figure 24. The residuals plot serves as an initial indicator of convergence and stability of the system. Even though the residuals converge, this is not an indication of the steadiness of the flow. The velocity field at the x-low and x-high faces of the unit-cell were investigated to see if the flow pattern changed mesh to mesh.

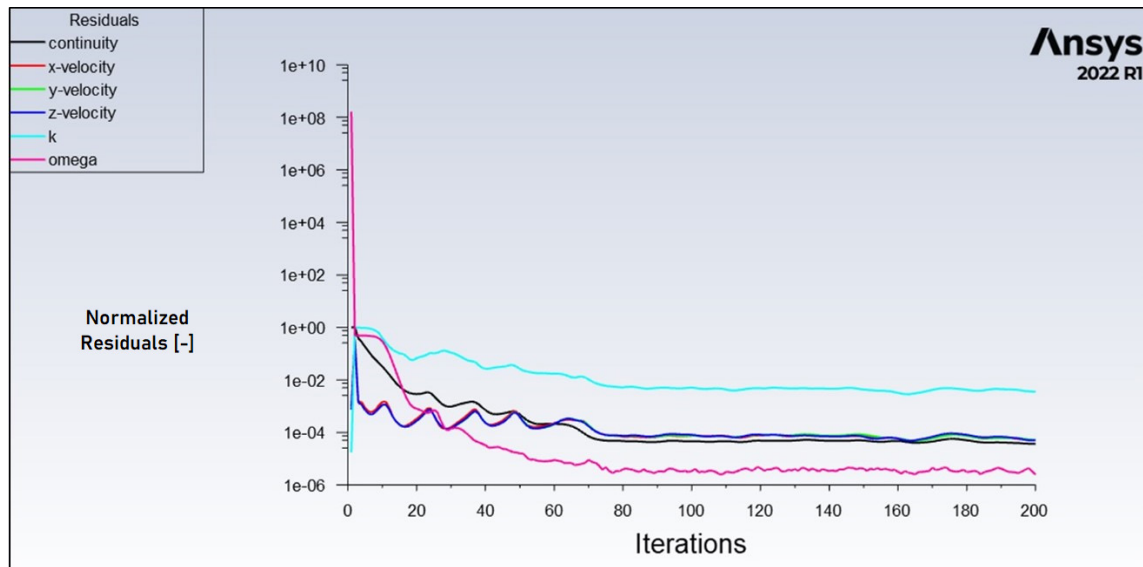


Figure 24. Residuals for Mesh Sensitivity Study

The velocity fields for all three meshes are shown in Figure 27. Each mesh produced the same flow pattern with vortices forming in the same location, the top right of each simulation. The vortices forming in the same location show that the results are mesh independent. This is supported by the convergence of velocity at a monitor point, as shown in Figure 25, where the velocity values stabilize to a single value across all meshes. Although the velocity convergence implies a steady flow, the results of the mesh sensitivity study indicate otherwise. While the vortices consistently formed in the same location, there was variation between meshes in terms of the number of vortices formed. This discrepancy may be attributed to the inherent unsteadiness of the flow caused by vortex shedding. The vortices observed in Figure 27 represent mere snapshots of the flow field possibilities over time, as vortex shedding can induce the formation of additional vortices. Similarly in Figure 26, the shear stress plot shows fluctuation revealing unsteady behavior. All three meshes yielded shear stress values within the same range despite therefore it was reasonable to utilize the coarse mesh, as depicted in Figure 28, in any steady-state solution.

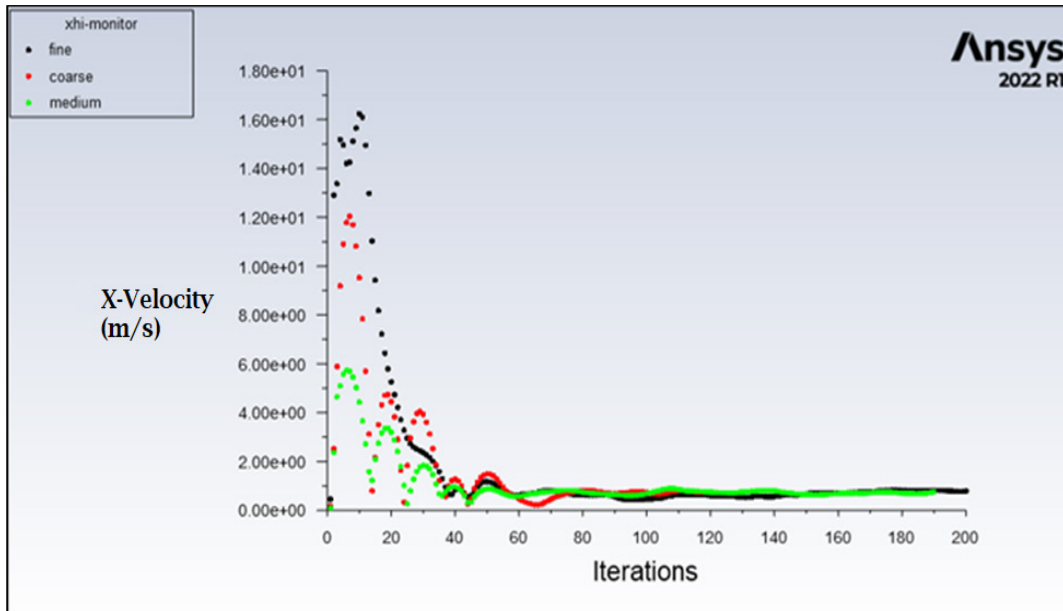


Figure 25. X-Velocity Monitor Point

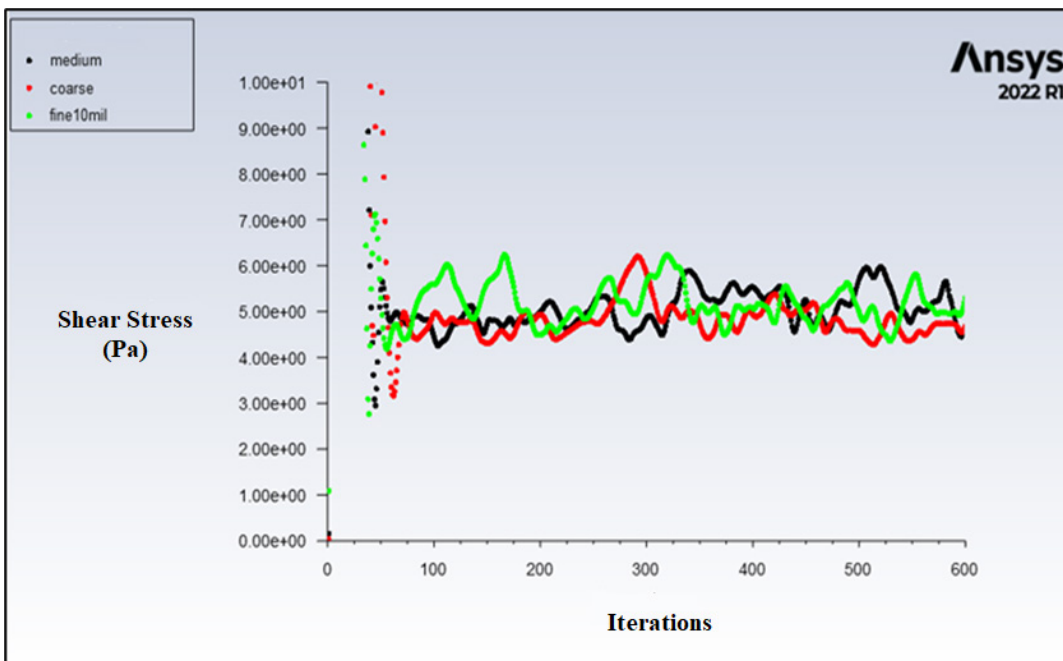
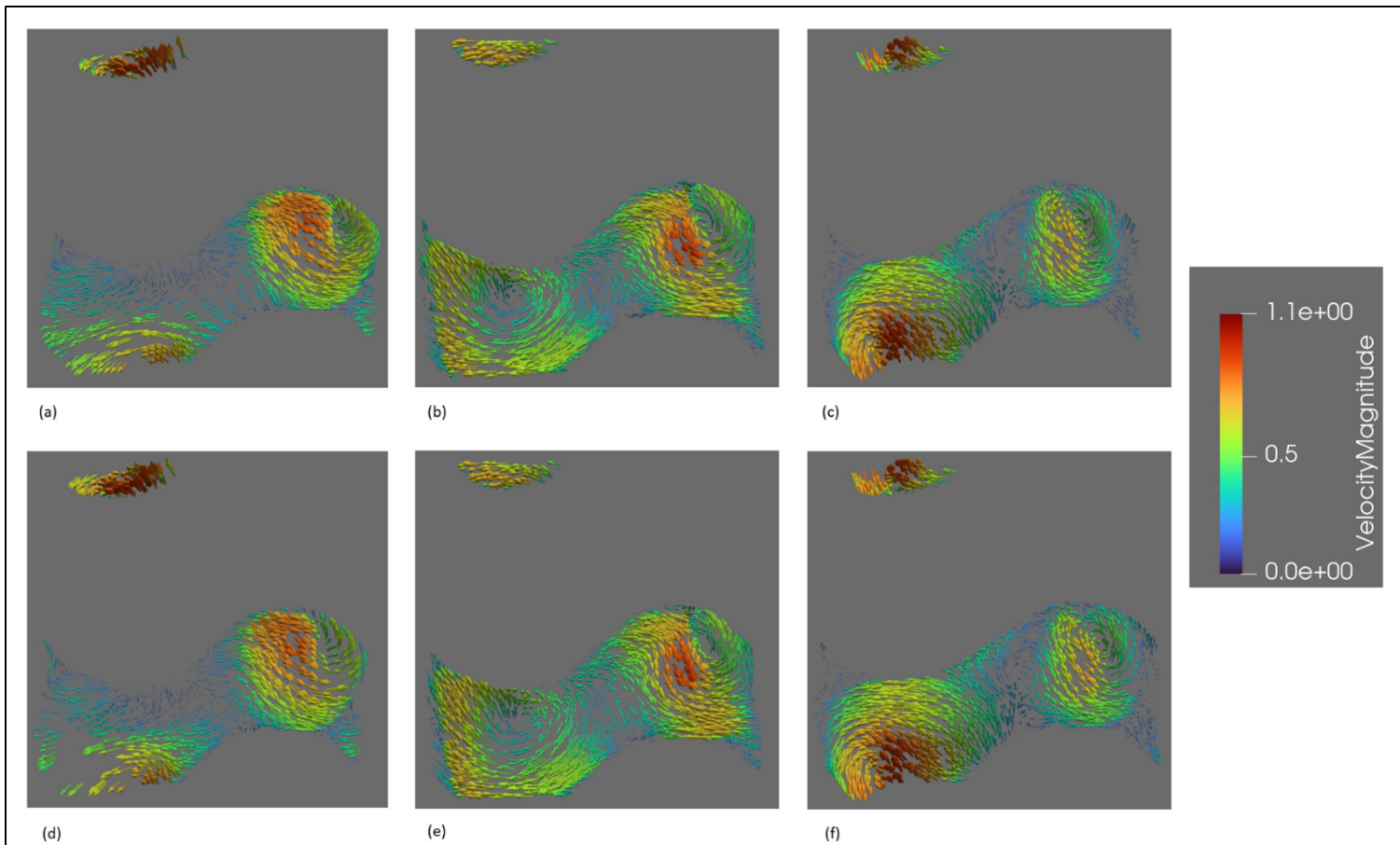


Figure 26. Shear Stress in Mesh Sensitivity



(a) Mesh Coarse Positive X-axis Face Velocity Flow (b) Mesh Coarse Positive Y-axis Face Velocity Flow (c) Mesh Coarse Positive Z-axis Face Velocity Flow (d) Mesh Coarse Negative X-axis Face Velocity Flow (e) Mesh Coarse Negative Y-axis Face Velocity Flow (f) Mesh Coarse Negative Z-axis Face Velocity Flow

Figure 27. Velocity Magnitude Vector Field of Each Mesh (m/s)

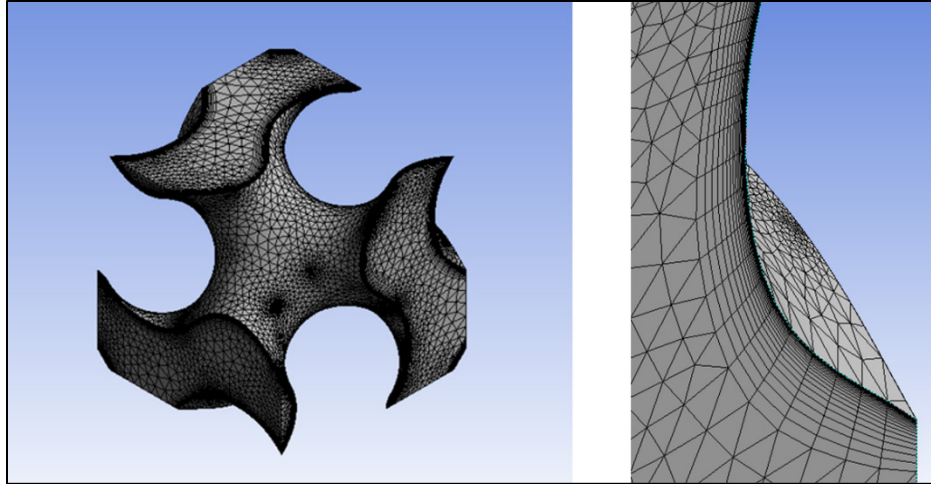


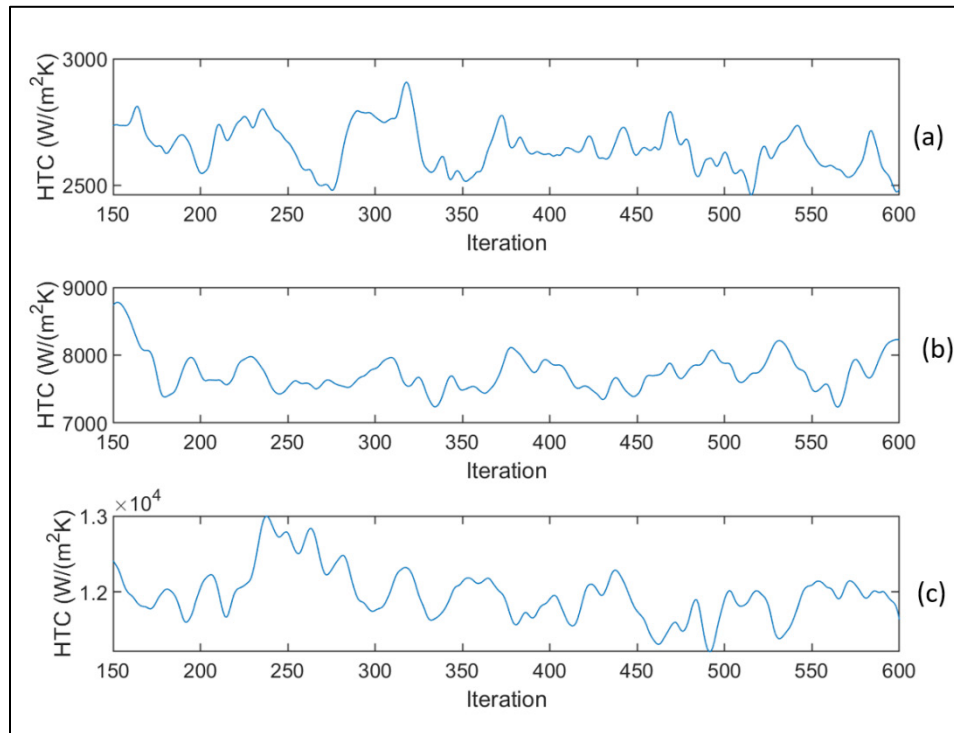
Figure 28. Coarse Mesh with Inflation Layers

B. SETUP

The goal of the heat transfer unit-cell model is to characterize heat transfer properties as Reynolds number varies to provide a baseline estimation of a heat exchanger design using TPMS. The coarse mesh was chosen for all subsequent steady-state conjugate heat transfer simulations. To ensure y^+ values remained under one, the highest Reynolds number case was chosen as the limiting case and defined the initial layer height for all the conjugate heat transfer runs, 1×10^{-4} m. All settings from the mesh sensitivity study were kept the same, with the addition of enabling the energy equation. The energy equation required an upstream temperature that was defined as 355.15 K, which is the inlet temperature for the fresh water from Table 6.

As observed in the mesh sensitivity study, the flow exhibits expected unsteadiness. Although this study assumes a steady-state solution, it is more appropriate to employ a transient solution to accurately capture the unsteady behavior of the flow. However, conducting transient simulations can be time-consuming. To overcome this, monitor points were incorporated to track variations in heat transfer coefficient, pressure drop, turbulence intensity, Nusselt's number, and friction factor within a steady-state solution. These monitor points exhibit fluctuations throughout the solution, providing mean, maximum, and minimum values for each property, which are utilized for further analysis. Having the minimum and maximum values allows for conservative estimations of the system's heat

transfer properties in comparison to a transient simulation. Figure 29 illustrates the fluctuation of these properties throughout the simulation due to the unsteadiness of the flow. The heat transfer problem was run with twenty different mass-flow rates to capture laminar and turbulent behavior of flow through TPMS.



(a) Monitor point of heat transfer coefficient in $Re = 300$ simulation (b) Monitor point of heat transfer coefficient in $Re = 1800$ simulation (c) Monitor point of heat transfer coefficient in $Re = 4500$ simulation

Figure 29. Fluctuation of Heat Transfer Coefficients Representing Unsteadiness of the Flow

In addition to the varying mass-flow rate, surface roughness was modeled. The goal of this research is to manufacture a heat exchanger using AM. It is known from literature that the surface roughness from AM parts is greater than wrought material. Equation (1.22) was used to find the sand grain equivalent roughness, roughness height, for three different roughness's. For all runs, C_s was set to 0.75 to model the randomness of the roughness along the surface.

The unit cell analysis was used as a design tool to provide a baseline for the heat transfer properties on both sides of the heat exchanger. Another way to provide a more accurate depiction of the heat transfer properties is to model the flow through a larger-scale model. Due to time-limitations, this study mainly focused on unit cell analysis. The large-scale model was looked at briefly to see the effects of the entrance and exit lengths, but results were never closely examined. The setup of the large-scale model is shown in Appendix C: Large Scale Model.

V. RESULTS

The goal of the unit cell analysis was to find the correlation between heat transfer properties and Reynolds number. Developing these relationships will be important for designing a large-scale heat exchanger with TPMS infill. Based on the results of the mesh sensitivity study, the coarse mesh was selected for further investigation of the heat conjugate problem. To characterize the heat transfer properties and flow behavior through the TPMS, a range of Reynolds numbers, found using (1.8), was considered, spanning from 300 to 40 000. The hydraulic diameter for one fluid domain was held constant throughout all simulations at 3.7 mm. The specific Reynolds numbers are shown in Table 12.

Table 12. Range of Flow Parameters

Inlet Velocity	Mass Flow Rate	Reynolds Number
0.0288	0.0025	306.62
0.2015	0.0176	1800.5
0.4318	0.0378	4597.1
0.5470	0.0478	5823.2
0.6909	0.0604	7355.7
0.8348	0.0730	8888.1
0.9788	0.0856	10421
1.1227	0.0982	11953
1.2667	0.1108	13485
1.4106	0.1234	15018
1.5545	0.1360	16550
1.6985	0.1485	18083
1.8424	0.1611	19615
1.9864	0.1737	21148
2.1303	0.1863	22680
2.2742	0.1989	24212
2.4182	0.2115	25745
2.5621	0.2241	27277
2.7061	0.2367	28810
2.8500	0.2492	30342

In addition, because the TPMS surfaces will eventually be printed, the surface roughness was also modeled to see the effect on heat transfer properties. The properties analyzed in this section include h , dp/dx , U , A_s , Nu , f and turbulence intensity. All properties were calculated using the code in Appendix D: Matlab Code to Post-Process the Data.

The laminar region is commonly defined by Reynolds number below 2000. The first two simulations are in this laminar region, and the laminar model was enabled for the simulation. The residuals are shown in Figure 30.

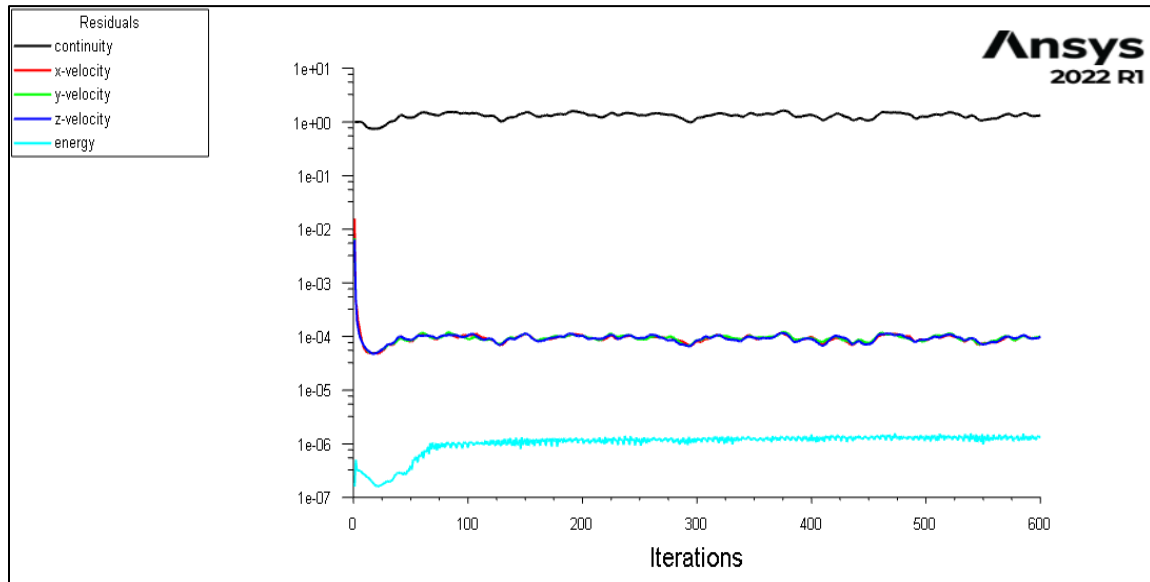


Figure 30. Residuals of Laminar Flow

In Figure 30, the residuals indicate that the continuity equation fails to converge. The continuity equation, represented by the black line in Figure 30 and (1.11), should equal zero at steady state. CFD aims to solve equations by minimizing the error towards zero, which is considered convergence. Typically, convergence is achieved when the residuals decrease by four orders of magnitude [38]. However, in Figure 30 the continuity equation remains above one and does not decrease in magnitude, indicating lack of converge. This suggests that the solution data may be inaccurate if the residuals fail to converge. The presence of inherent unsteadiness within the TPMS geometry, caused by its curvature, is

suspected to be the reason why the residuals for the laminar flow region did not resolve. To resolve this unsteadiness, it can be modelled as a steady turbulent fluctuation. To accommodate this, the $K - \omega$ SST model was enabled for all Reynolds numbers.

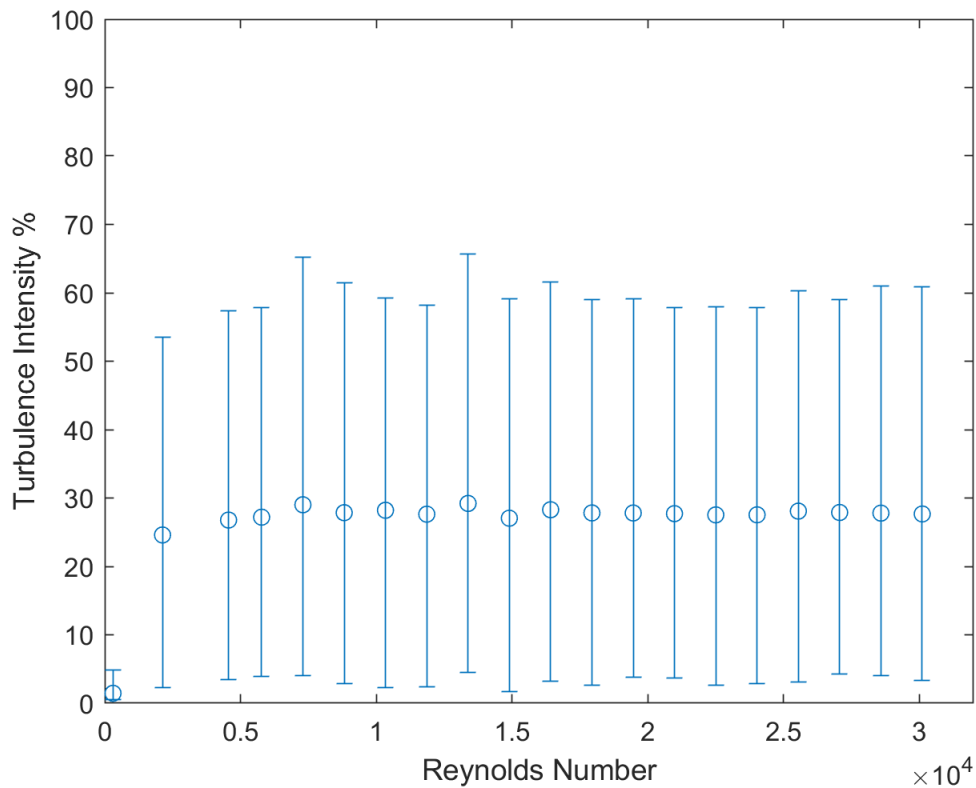


Figure 31. Turbulence Intensity

Figure 31 shows the turbulence intensity from (1.6). The turbulence intensity provides information about the level of turbulence in the flow and helps determine whether turbulence models should be utilized. According to the fluent modelling guide, turbulent models should be employed when turbulent intensity is above 10% [38]. In the case of the laminar run at 1800 Reynolds number, the turbulence intensity exceeds 10%, indicating it is appropriate to employ the turbulence model. The 300 Reynolds number simulation has little turbulence intensity; it can be concluded that the unsteadiness of the flow due to suspected vortex shedding hampered the continuity equation from resolving in Figure 30.

The residuals for the suspected laminar runs using the turbulence model are shown in Figure 32.

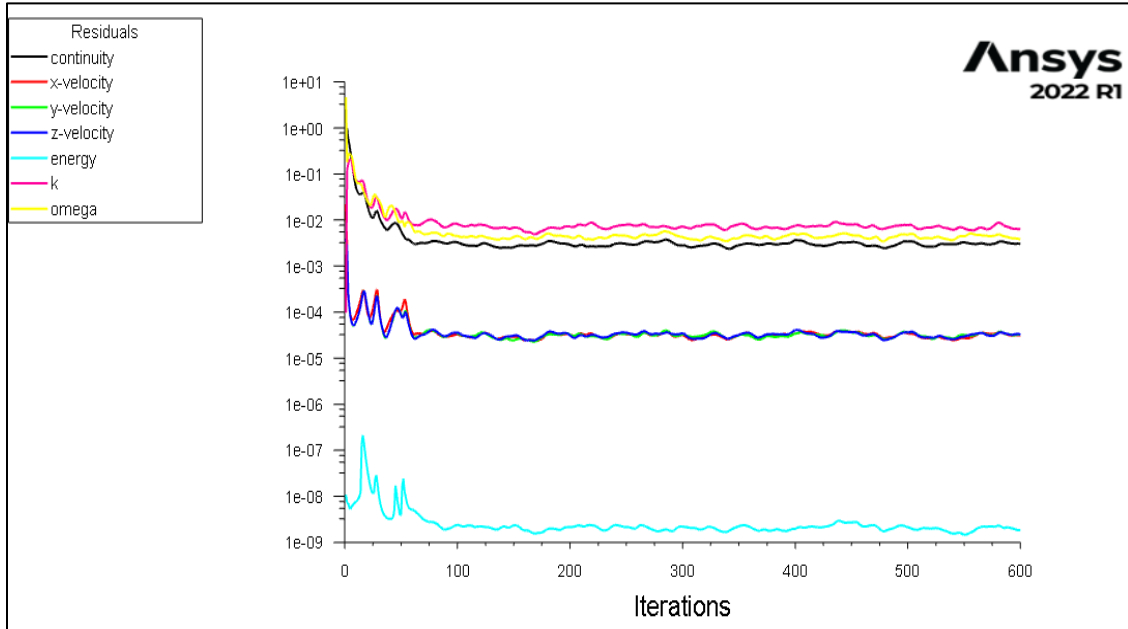


Figure 32. Residuals from Turbulent Model Runs

The residuals in Figure 32 are representative of all simulations, as there was very little variance among residual plots. The residual plot of Figure 32 shows the periodic boundary conditions solving the cyclic nature of the flow by the oscillations reaching a minimum steady value around iteration 150. As seen before, the residuals may converge, but the flow may still be unsteady because the flow stops evolving after 150 iterations all calculations using the mean, minimum, and maximum values of the monitored heat transfer properties were taken after 150 iterations.

The surface heat transfer coefficient was solved using (1.21), where T_{ref} was set to 355.15 K which is the inlet temperature from Table 6. The heat transfer coefficient quantifies the efficiency of heat transfer within the system for the given fluid domain. The heat transfer coefficient in relationship to the Reynolds number is shown in Figure 33.

As expected, the heat transfer coefficient increases with the Reynolds number. This is due to the increases in turbulence energy, which improves heat transfer. There appears to be a linear relationship between Reynolds number and heat transfer coefficient. As the Reynolds number increases there seems plausible that the heat transfer coefficient could reach a maximum and start to plateau. Further research into higher Reynolds numbers could investigate the pattern of the heat transfer coefficient. The error bars in Figure 33 are the maximum and minimum values observed from the monitor points throughout the simulation.

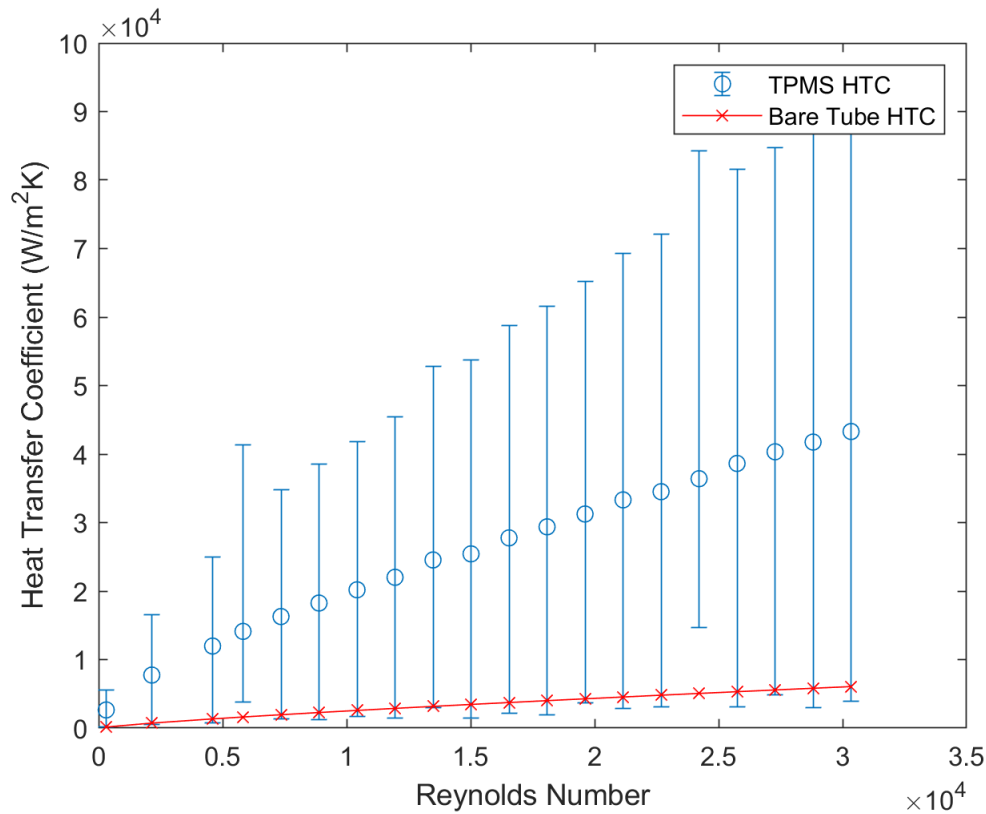


Figure 33. Heat Transfer Coefficient vs. Reynolds Number

Using the minimum heat transfer coefficient of each simulation, a conservative estimation for the overall heat transfer coefficient and heat transfer area was obtained. Using the same Reynolds number and Dittus-Boelter Correlation for a bare 19 mm tube, the heat transfer coefficient was calculated for the shell and tube heat exchanger [27].

Figure 33 shows that the mean TPMS heat transfer coefficient is four times higher than the shell and tube heat exchanger, which are similar to the trends Advanced Engineering Solutions were obtaining for their heat exchanger.

To calculate Nusselt's number, another heat transfer property derived from the heat transfer coefficient, (1.4) was used. To validate that the Nusselt number obtained from fluent was using (1.4), the first simulation's heat transfer coefficient and Nusselt number were plotted to see if there was a linear relationship.

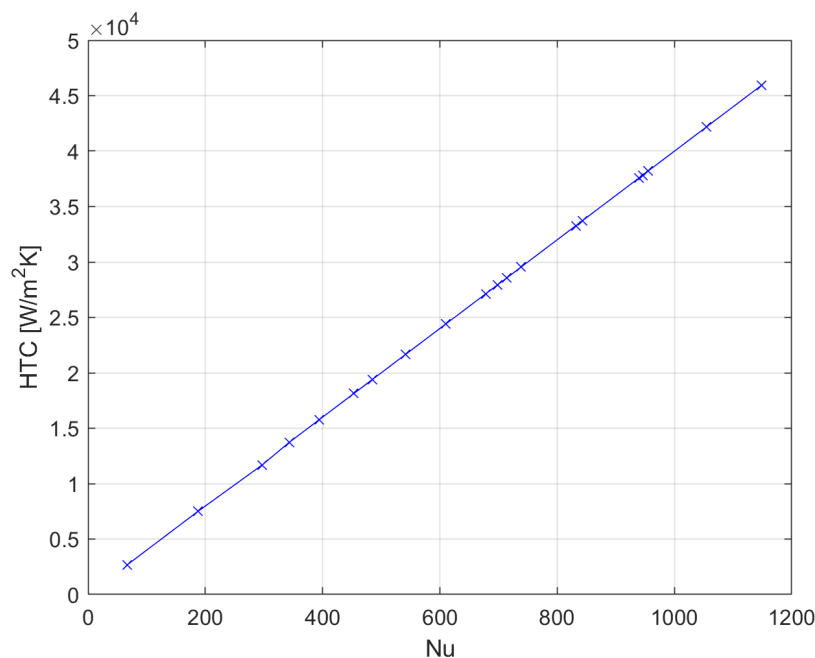


Figure 34. Nu Number Relationship to Heat Transfer Coefficient

The relationship between the Nusselt number and heat transfer coefficient is linear, it is appropriate to assume (1.4) for all simulations. The Nusselt number correlation to Reynolds number is shown in Figure 35. The Nusselt numbers obtained in this study, using water as the working fluid, are highly comparable to those reported in previous studies shown in Figure 12. W. Li's study utilized CO₂ as the working fluid [25]. It's important to note that CO₂ has a significantly lower thermal conductivity of 0.01663 W/m²·K, while water in this research has a thermal conductivity of 0.6 W/m²·K [25], [40]. Additionally,

W. Li's study used a hydraulic diameter that was half the size of the one used in this study. These disparities in hydraulic diameter and thermal conductivity explain why there is minimal variation in the Nusselt number between the two studies, despite the difference in working fluids.

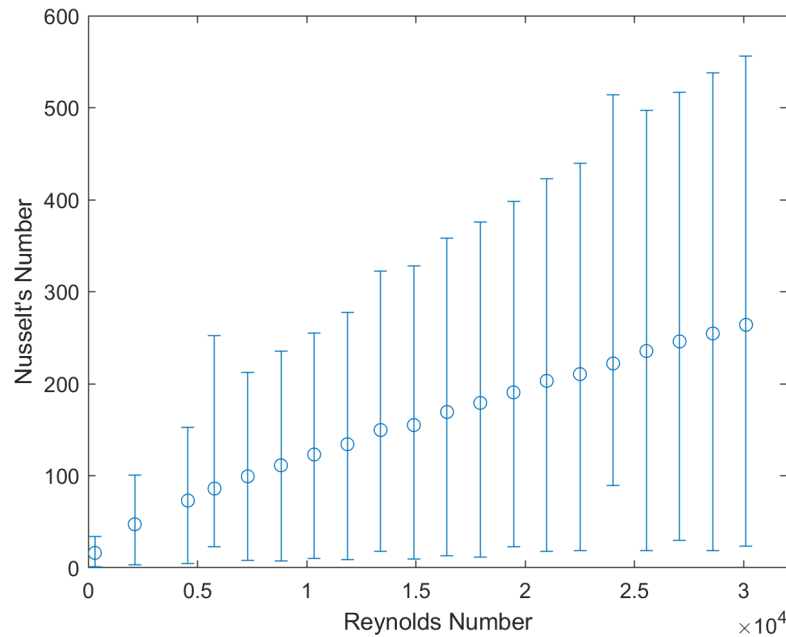


Figure 35. Nusselt Number vs. Reynolds Number

To evaluate the overall performance of a heat exchanger the overall heat transfer coefficient was analyzed. It was assumed that the heat transfer coefficients for both fluid domains are equal due to the fact that the TPMS unit cell is symmetric. Rearranging (1.2) the overall heat transfer coefficient (U) was calculated. From U , the required heat transfer area for dissipating the heat from Table 6, 5.86 MW. The results are tabulated in Table 13 where an increase by 458% for U and decreased area by 84% can be seen. Even though the large increases performance is seen with TPMS, the effects of entrance and exit length were not taken into consideration for this research.

Table 13. Heat Transfer Area and Overall Heat Transfer Coefficient

	Heat Transfer Area (m ²)	Overall Heat Transfer Coefficient (W/ m ² ·K)
Validation Calculations (1.79m/s)	277	519
TPMS (1.69m/s)	43	2894.5
	Heat Transfer Area (m ²)	Overall Heat Transfer Coefficient (W/ m ² ·K)
Validation Calculations (1.79m/s)	277	519
TPMS (1.69m/s)	43	2894.5

Table 13 focused on the design parameter flow rate found in Appendix A. Matlab code for Initial Calculations, 1.79 m/s, that was based on the design parameters in Table 6. The closest flow rate tested in CFD was 1.69 m/s, providing the more conservative simulation for heat transfer properties of a large-scale model heat exchanger. Whereas Figure 36 shows how the overall heat transfer coefficient varies with the Reynolds number. Notably, the graph reveals a trend where the overall heat transfer coefficient reaches a plateau above a Reynolds number of 10 000. This indicates that there is an optimal Reynolds number for TPMS, beyond which the increase in flow rate results in minimal improvement in heat transfer. The two spikes in Figure 36 can be attributed to the unsteadiness of the flow producing inaccurate steady-state results.

The heat transfer area was calculated using the overall heat transfer coefficient found in Figure 36, and (1.2). The heat transfer area will correlate to the overall size of the heat exchanger. Similarly to the overall heat transfer coefficient, the area begins to plateau at 10 000 Reynolds number. A line of best fit was modeled in Figure 36 and Figure 37. The equation produced from the line of best fit could be used in a design code to find the optimal operation conditions. While increasing the Reynolds number increases the performance of the heat exchanger, it increases the pressure drop. Pressure drop can be detrimental to a

heat exchanger, the greater the pressure drop the more power required to pump the fluid through the system.

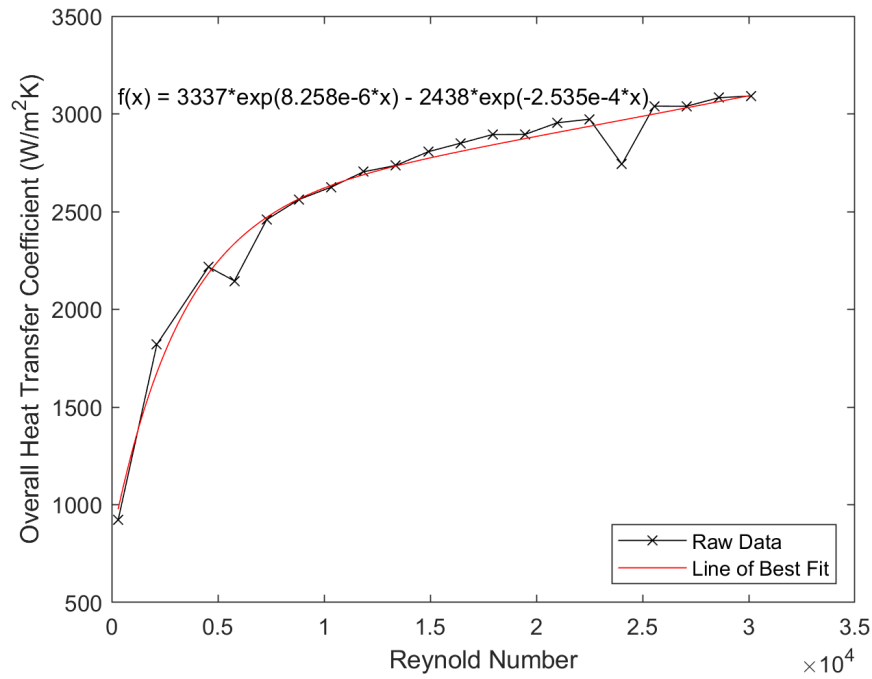


Figure 36. Overall Heat Transfer Coefficient

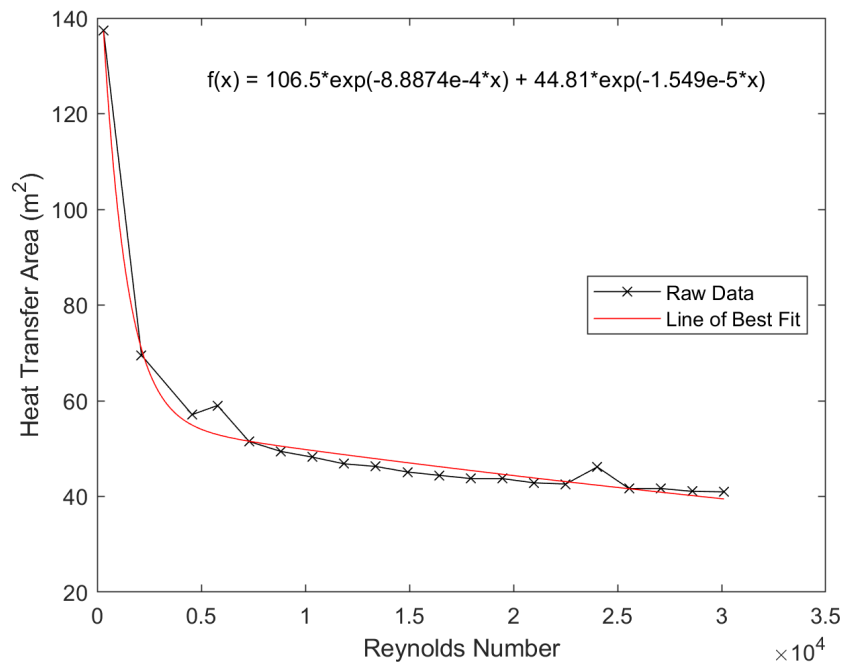


Figure 37. Heat Transfer Area vs. Reynolds Number

Figure 38 shows a parabolic relationship between pressure drops and mass flow rate, indicating that as the mass flow increases, pressure drop also increases. Consequently, higher power is required to pump the fluid through the system as pressure drop rises. The increased power demand can escalate operational costs and potentially necessitate the use of a new pump.

Establishing a correlation for pressure drop that incorporates the effects of entrance and exit length is currently challenging due to the novelty of this research field. The TPMS structure introduces a highly random flow path, making it difficult to accurately model pressure drops across a heat exchanger.

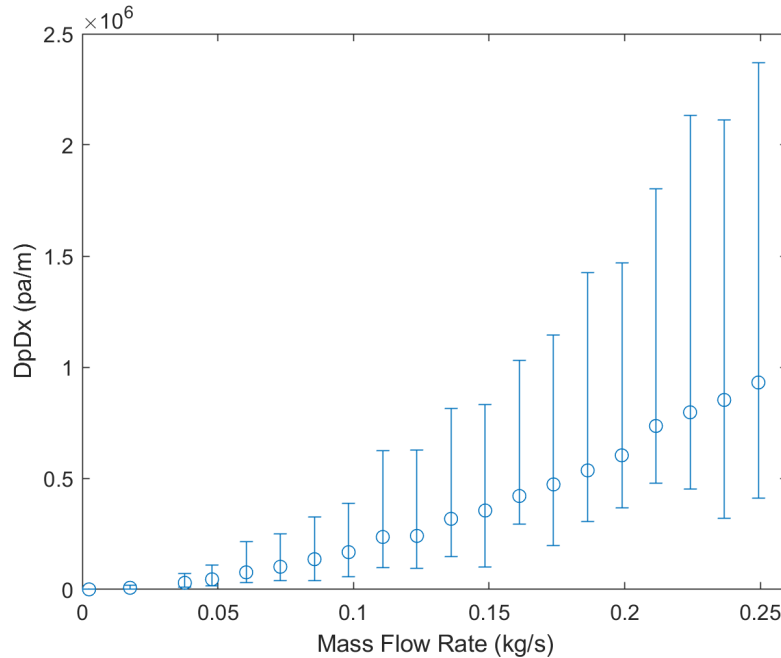


Figure 38. Pressure Drop per Unit Length

To obtain an estimate for the highest pressure drop one could achieve in a large-scale model Appendix D: Matlab Code to Post-Process the Data and (1.10) were utilized. The inlet area from Table 6 determined the number of unit cells needed to cover it in a rectangular prism. The heat transfer surface area for one fluid domain in the rectangular prism was then determined using the length of the unit cell. Dividing the total heat transfer area calculated in Figure 37 by the heat transfer surface area of one rectangular prism provided the hypothetical length of the heat exchanger. Multiplying the maximum pressure drop per unit length from Figure 38 by the hypothetical length using rectangular prisms yielded an estimation for the overall pressure drop in a large-scale model. From this correlation a relationship between heat exchanger length to Reynolds number and pressure drop to mass flow rate were found in Figure 39 and Figure 40. In Figure 40, the design

point from Table 6 is marked with a red cross, which was found using volumetric flow rate and converting it to an inlet velocity using the cross-sectional area. Therefore, the inlet velocity of the unit cell and the reference heat exchanger in Table 6 are the same.

The heat exchanger length is significantly reduced compared to the 3 m length specified in Table 6. Using design conditions from Table 6 and the correlation described above, the length is reduced by 87% which correlates to how much the heat transfer area needed decreased by in Table 13. Despite the decrease in length, the predicted pressure drop exceeds the design parameters in Table 6 by more than ten times the acceptable limit.

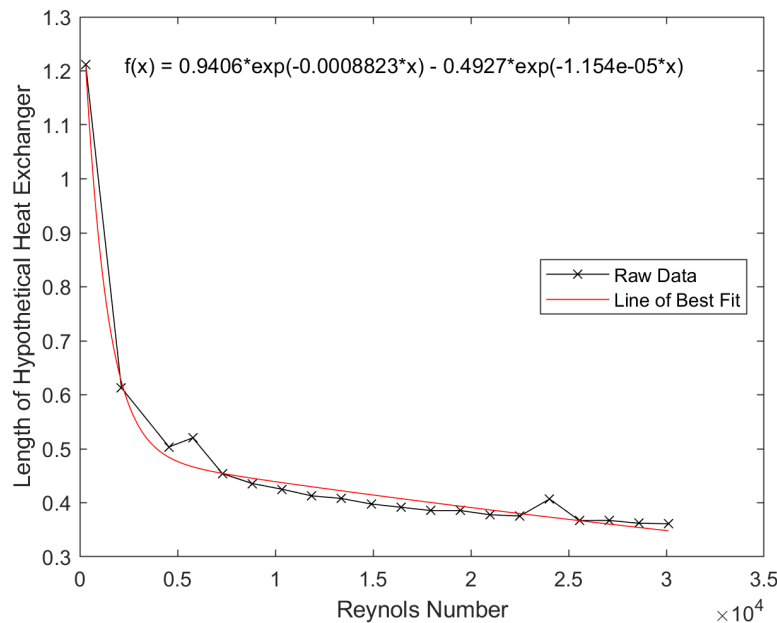


Figure 39. Heat Exchanger Length

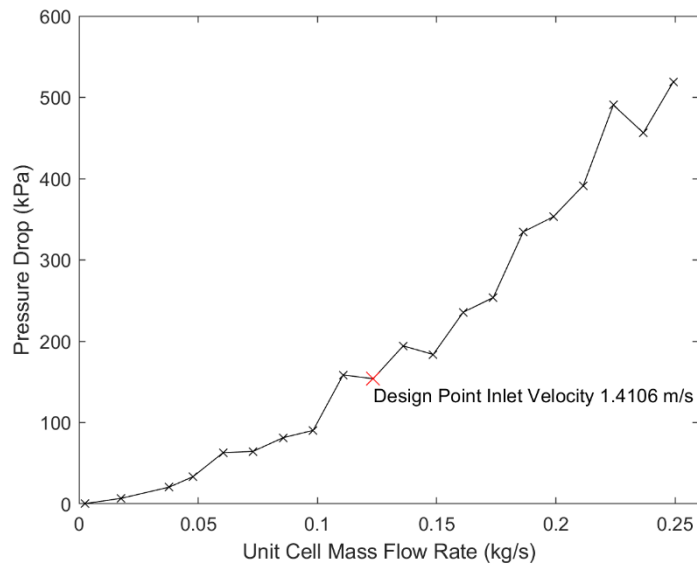


Figure 40. Large-Scale Model Conservative Pressure Drop

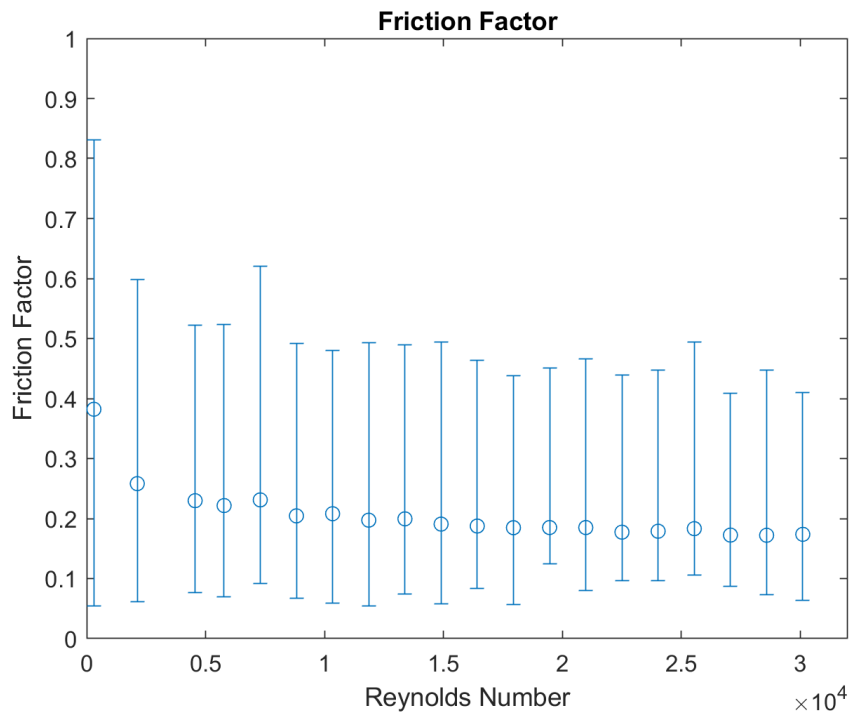


Figure 41. Friction Factor vs. Reynolds Number

The friction factor helps determine the pressure losses due to the friction between the fluid and the pipes. Similar to W. Li's study, the friction factors for TPMS geometries are constant as Reynolds number increases in Figure 41 [25]. This indicates that most of the pressure drop through the TPMS geometry is from the inherent curvature of the geometry and not the friction between the solid and the fluid.

The goal is eventually to fabricate the TPMS geometry using AM. To account for the increased surface roughness resulting from AM, three different average surface roughness, R_a , were tested. Using (1.22) the equivalent sand grain roughness, k_s , was found to model the roughness effects in fluent. For all the following results, the mass flow case held constant at 0.1234 kg/s and Re of 14 9040.

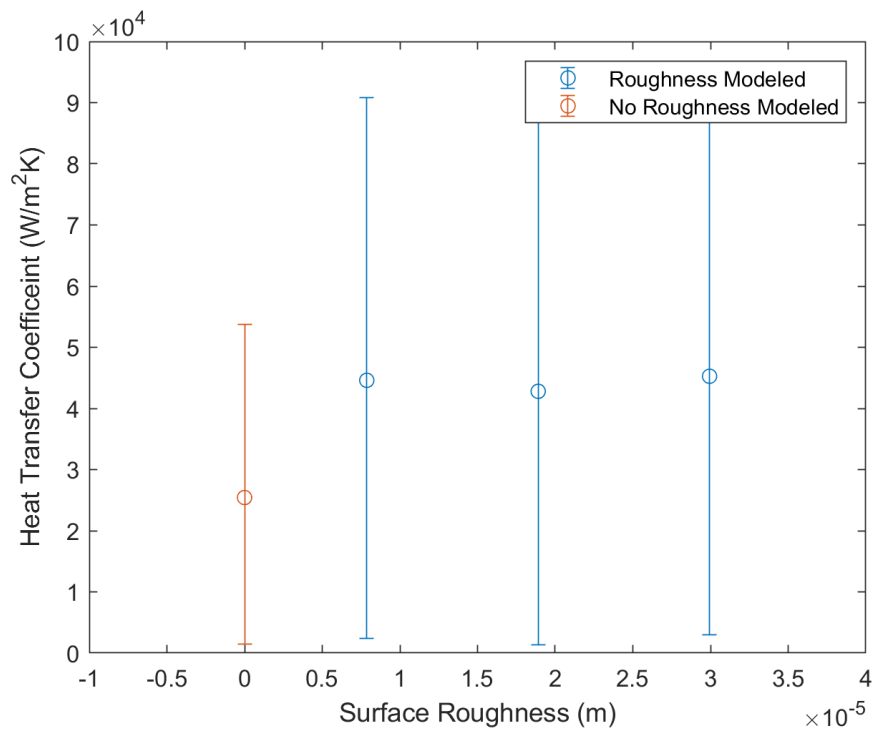


Figure 42. Surface Roughness Heat Transfer Coefficient

The heat transfer coefficient almost doubles when the roughness is modeled, highlighting the significant impact of roughness on heat transfer. This is due to the extra

heat transfer created by the rough surface. Rough surfaces will create non-uniform surfaces that could lead to high pressure drops.

From Figure 11 the friction factor is expected to increase with the increase surface roughness. Figure 43 shows that the friction factor does not change with the surface roughness, contradicting what was expected. This could be due to the curvature of TPMS, because Moody Diagrams are based on straight rough pipes.

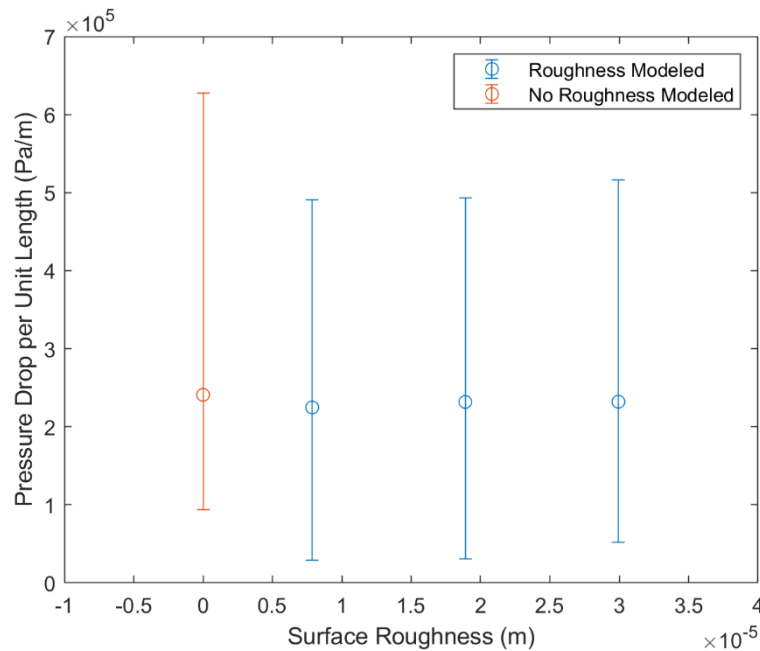


Figure 43. Surface Roughness Friction Factor

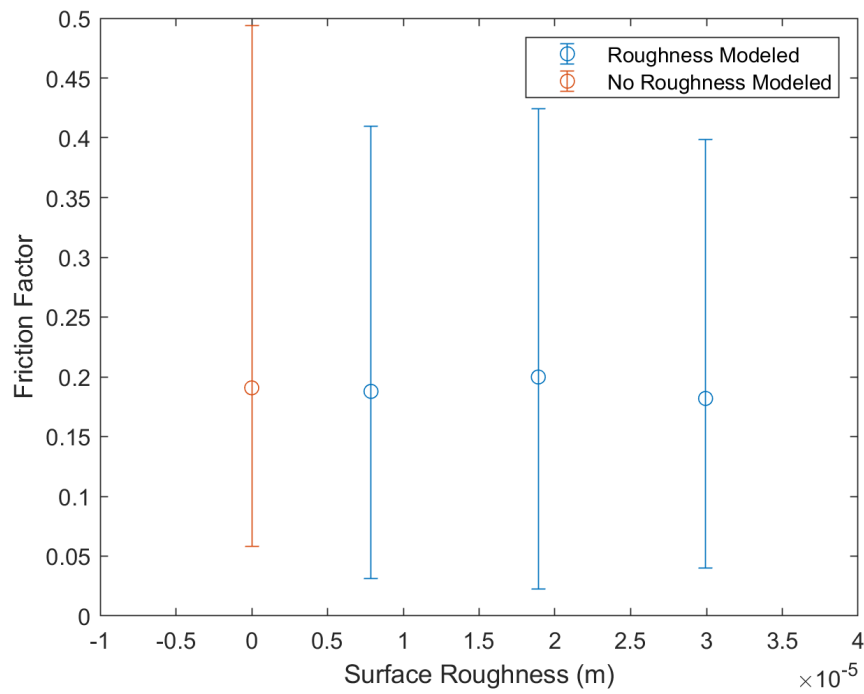


Figure 44. Surface Roughness Pressure Drop

Figure 44 shows similar trends to Figure 43 as the pressure drop does not vary with surface roughness. The results show that the surface roughness has negligible effects on overall pressure drop at high Reynolds numbers.

VI. CONCLUSION

This research demonstrates the potential of using AM to create complex geometries, such as TPMS, in heat exchangers. These novel geometries have the ability to significantly enhance performance while reducing the size of the heat exchanger. The overall heat transfer coefficient (U) increased by 458%, indicating a substantial improvement in performance. Additionally, the required surface area for heat dissipation decreased by 84% compared to initial calculations. The heat transfer coefficients for both fluid domains in the TPMS were assumed to be the same, $27\,490\text{ W/m}^2\cdot\text{K}$, and were significantly higher than those in traditional tube and shell heat exchangers, 6821.1 and $478.496\text{ W/m}^2\cdot\text{K}$ respectively. However, it is important to note that this study did not consider the effects of entrance and exit lengths on the heat exchanger.

Due to the complexity of this research it is challenging to obtain an accurate correlation for pressure drop with entrance and exit length effect taken into consideration. Pressure drop is a crucial parameter in heat exchanger design, and future research should focus on developing a correlation for pressure drop that includes the entry and exit effects into the TPMS fluid domain. This future research will include the distance it takes for the fluid to fully develop. The initial pressure drop observed in this research exceeded the required threshold for the desired heat exchanger. Thus, future efforts should focus on minimizing pressure losses through the TPMS.

Although the TPMS geometries were not physically printed in this research, cost estimations indicate that an additively manufactured heat exchanger with TPMS could reduce costs to approximately \$1 million. Obtaining actual quotes from AM vendors, including post-processing procedures to meet Navy quality standards, will be essential for future work.

To ensure the heat exchanger meets Navy quality standards, large-scale model finite element analysis (FEA) will be incorporated in future research. This analysis will evaluate how TPMS structures respond to hydrodynamic forces, providing insights into fatigue strength and the expected lifespan of the TPMS heat exchanger. It is important to

examine the thickness of TPMS structures, as thicker structures enhance structural strength but may compromise heat transfer efficiency.

A. FUTURE WORK

The field of heat exchangers using TPMS shows great promise, but further studies are necessary to address the complexity of this research. Further research on the unit cell model should include an FEA analysis to see if the gyroid lattice can withstand high pressures. Another area that warrants investigation is the bias of the TPMS unit cell, which can be adjusted to create different fluid domains with varying hydraulic diameters. Manipulating the bias allows for flexibility in design and can lead to different pressure drops across the domains. Other Future research in the short term should aim to assess the model in Appendix C: Large Scale Model in CFD. Analyzing a large-scale model will help understand the effects of the entrance and exit length of the heat exchanger to provide a more accurate pressure drop analysis. Long term future research should include manufacturing a large-scale TPMS model. Manufacturing a large-scale heat exchanger will allow for physical testing of pressure drop to validate any CFD results. In addition having a physical model will help assess the physical strength of the printed Ti-4Al-6V. Ultimately, such investigations will help determine if TPMS-based heat exchangers are a viable replacement option for the U.S. Navy in the future.

APPENDIX A. MATLAB CODE FOR INITIAL CALCULATIONS

```
%% Validation
clc
clear
close all
%% Boundary Conditions
uT = 'K';
uS = 'ppt';
uP = 'MPa';
D_i = 19.05e-3; % [m^3]
t = 2.5e-3; % [thickness]
Do = D_i+(2*t); % Diamter of outisde
T_SW_max = 32+273.15; % [C]
T_FW = 82 + 273.15; % [K]
S = 34.5; % [g/L]
P_atm = 0.101325; %[MPa] atmospheric pressure
P_max_SW = 10; % [MPa]
P_max_FW = 1; % [MPa]
[rho_SW,mu_SW,pr_SW,cp_SW,k_SW] = waterproperties(T_SW_max,S,P_atm,uT,uS,uP);
% mu_SW = 0.00087; % [Ns/m^2] seawater dynmaic viscosity @ 30
mu_water = 0.0003453; % [Ns/m^2]dynmaic viscosity @ 82
k_water = 0.6; % [W/m^2*k]
k_titanium = 6.7; % [W/m^2*k]
% k_SW = 0.66; % [approximation Thermophysical properties of seawater: a
review of existing correlations and data]
Q = 5.86e6; % [W]
V_dot_SW = 0.145; %[m^3/s]
V_dot_FW = 0.063; % [m^3/s] limiting case
rho_water = 997; % [kg/m^3]
% rho_SW = 1025; % [kg/m^3]
eps = 0.88; % Relative porosity
SSA = 3.76; %mm-1
% Q_H = Q; % Heat exchange from hot fluid
% Q_C = Q_H; % HX from cold fluid
m_dot_FW = V_dot_FW*rho_water; % [kg/s]
m_dot_SW = V_dot_SW*rho_SW; % [kg/s]
Cp_FW = 4187; % [J/kg*k]
Cp_SW = 4011; % [J/kg*k]
inlet_area = 0.75^2;
v_flow_SW = m_dot_SW/(inlet_area*rho_SW); % [m/s]
% v_flow_FW = m_dot_FW/(inlet_area*rho_water); % [m/s]
A_model = pi*(37.5*10^-3)^2; %[m^2]
m_dot_model_SW = rho_SW*A_model*v_flow_SW; % [kg/s]
% m_dot_model_FW = rho_water*A_model*v_flow_FW; % [kg/s]

%% Find outlet temperatuers
T_SW_o = -1*((Q/(m_dot_SW*Cp_SW))-T_SW_max);
T_FW_o = -1*((Q/(m_dot_FW*Cp_FW))-T_FW);
%% LMTD method
delta_1 = T_FW-T_SW_max;
delta_2 = T_FW_o-T_SW_o;
```

```

LMTD = (delta_2 - delta_1)/(log(delta_2/delta_1));
%% Reynolds Number
Unit_L = 15e-3; % [m] Unit cell length
Dh = (4*eps/SSA)*10^-3; % [m]
D_inlet = (37.5*10^-3)*2;
% D_h = (0.52*Unit_L); % [m]
Re_SW = rho_SW*v_flow_SW*D_inlet/mu_SW; % Reynolds number for seawater
%% Pressure Drop for sea water
Factor = 1.6903; % Factor from lattice to sheet
R = 37.5e-3; % Radius of model in m
Nu_SW = 220; % From literature
L = (37.5*2)*10^-3;
h_local_SW = Nu_SW*k_SW/Dh;
% ed = (7.867*10^-6)/Dh;
f_SW = 0.18; % From literature
CSA = pi*R^2;
l = 15*10^-3;
SA = (3.10*1^2)*(5^3)*Factor;
% SA_1 = 91000.1938*(10^-6)*Factor; % [m^2]
% h = 32.5e-3; % entrance length m
Delta_P_SW = (f_SW*2*rho_SW*v_flow_SW^2*L)/Dh;
% Delta_P_FW = (f_SW*2*rho_water*(v_flow_FW^2)*L)/Dh;
%% Validation for SW
Actual_P_SW = 10e3; % [Pa]
% D_h_max = (f*2*rho_water*v_flow_SW^2*L_max)/Actual_P; % [m]
L_max_SW = (Actual_P_SW*Dh)/(f_SW*2*rho_SW*v_flow_SW^2); % [m]
A_nec_SW = Q/(h_local_SW*LMTD); % [m^2]
A_nec_SW_BF = A_nec_SW*Factor;
V_nec_SW_BF = Dh*A_nec_SW_BF/4; % Volume of TPMS
V_unitcell = (0.5*1)^3;
N_unitCells_SW = round(V_nec_SW_BF/V_unitcell);
cube_SW = round(N_unitCells_SW^(1/3));
L_SW = cube_SW*(1);
%% Shell side
A_nozzle = pi*((0.75/2)^2);
v_flow_FW = m_dot_FW/(rho_water*A_nozzle); % [m/s]
m_dot_model_FW = v_flow_FW*rho_water*A_model;
Re_FW = rho_water*v_flow_FW*D_inlet/mu_water; % Reynolds number for seawater
%% Validation for FW
Actual_P_FW = 6.3e3; % [Pa]
f_FW = 0.14;
Nu_FW = 250;
h_local_FW = Nu_FW*k_water/Dh;
L_max_FW = (Actual_P_FW*Dh)/(f_FW*2*rho_water*v_flow_FW^2); % [m]
A_nec_FW = Q/(h_local_FW*LMTD); % [m^2]
A_nec_FW_BF = A_nec_FW*Factor;
V_nec_FW_BF = Dh*A_nec_FW_BF/4; % Volume of TPMS
N_unitCells_FW = round(V_nec_FW_BF/V_unitcell);
cube_FW = round(N_unitCells_FW^(1/3));
Delta_P_FW = (f_FW*2*rho_water*v_flow_FW^2*L)/Dh;
L_FW = cube_FW*(1);

```

APPENDIX B: TWO FLUID DOMAIN SETUP

Because only one periodic boundary could be defined on one fluid domain, a mass-flow inlet and pressure outlet were defined. The rest of the faces were defined as symmetry walls to closely related to the periodic boundary condition. This setup is shown in Figure 45. The results of this run are shown in Figure 47, which shows one fluid domain not developing and reaching the outlet, therefore proving inaccurate results.

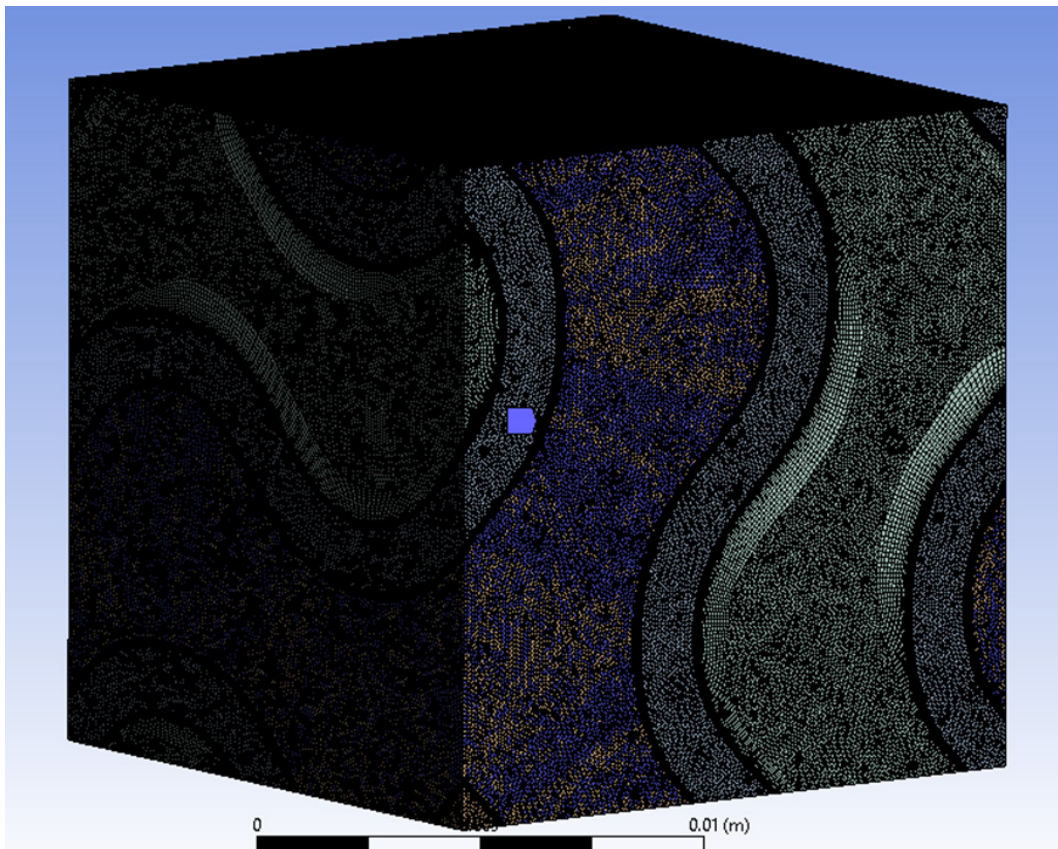


Figure 45. Mesh of Two-Fluid Domain Setup

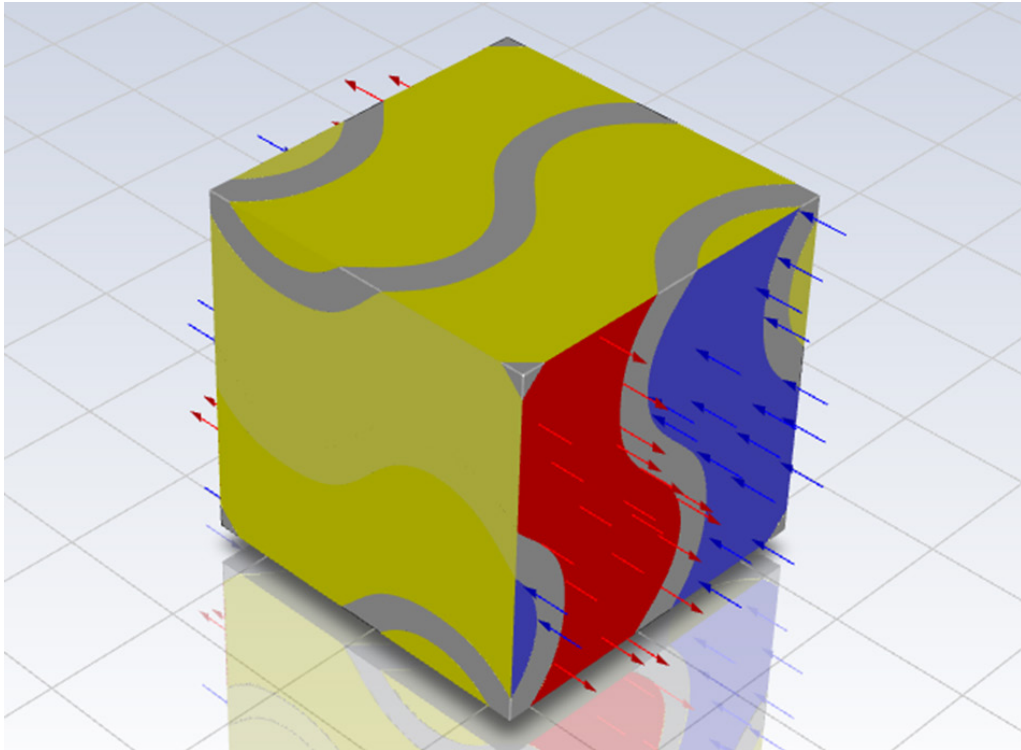


Figure 46. Setup for Two-Fluid Domain in Fluent

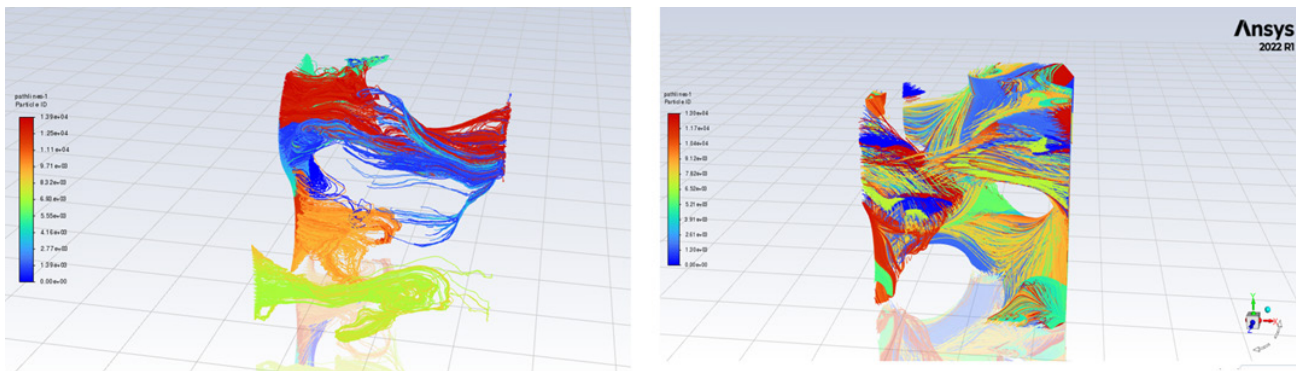


Figure 47. Pathlines of Failed Two-Fluid Domain Setup

APPENDIX C: LARGE SCALE MODEL

The larger model of the heat exchanger gives an actual heat transfer coefficient and a better understanding of how the unit cell can be interpreted. Therefore, modifications can be made to the unit cell to produce more accurate results closer to the larger heat exchanger model and make solutions converge faster. Ntopology has a built-in function that creates lattice structures of TPMS. Ntopology creates implicit bodies that are inherent to the natural language of the software. Because the implicit bodies are inherent to the software, this creates a problem of exporting the 3D model to ANSYS. The implicit body of the model is shown in Figure 48. From the implicit body a surface mesh file was created so Spaceclaim would accept the geometry. Spaceclaim would create a sdoc file that ANSYS fluent could read and analyze. The problem with creating a sdoc file from a mesh file is ANYS claims it as a “dirty geometry” [38]. Because of the nature of how the geometry was imported into ANSYS a beta function in Fluent was used to mesh and analyze the model, fault-tolerant meshing.

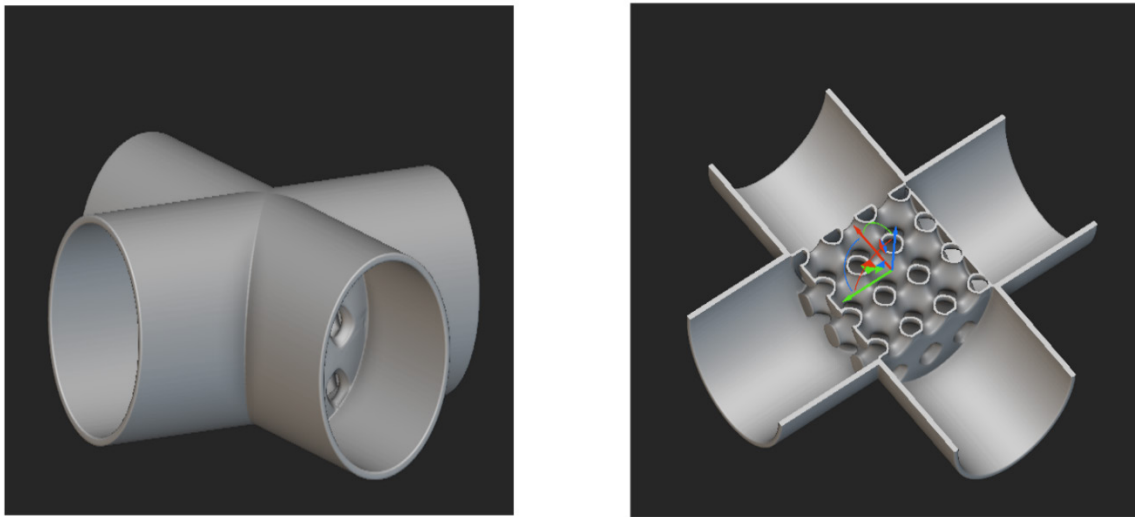


Figure 48. Implicit Model of TPMS Heat Exchanger

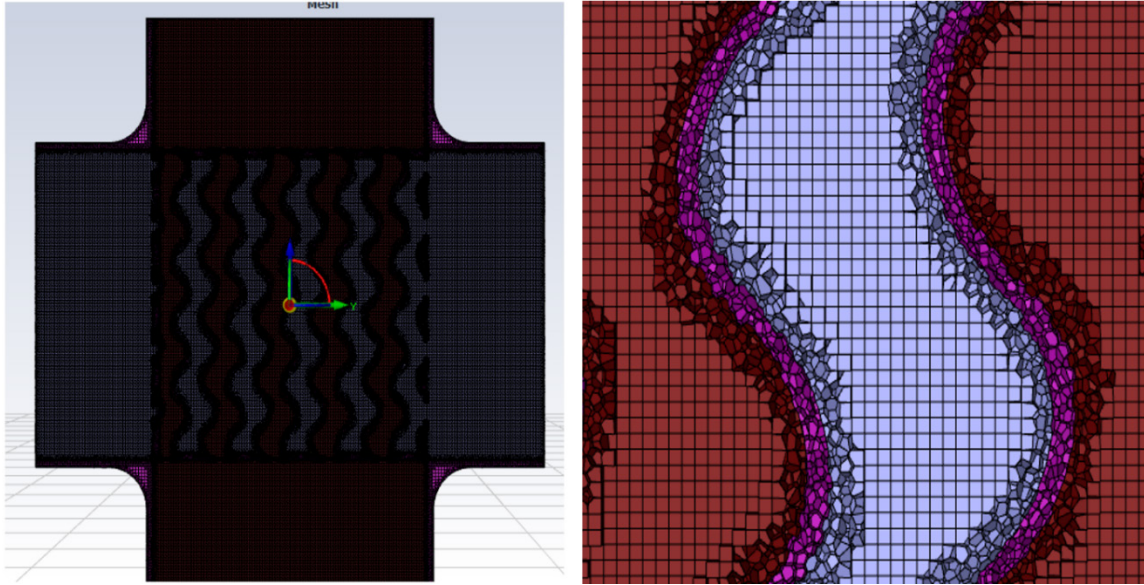


Figure 49. Fault-Tolerant Mesh without Inflation Layers

As cited from previous literature, a model with this much turbulent flow and curvature needs a wall resolved mesh to provide the most accurate data from the k- ω SST turbulence model [25]. For the model to be wall resolved y^+ values must be below 1. From the same literature the initial boundary layer height was set to $2.5e-7$ m to obtain y^+ of 1, same as the unit cell [25]. The CFD setup for the larger model heat exchanger is the same as the unit cell, with periodic boundary conditions at the inlets and outlets to obtain full developed flow. With the larger scale model, it allows for more accurate results to design a full-scale model of the desired heat exchanger. The mesh used is in Figure 49.

APPENDIX D: MATLAB CODE TO POST-PROCESS THE DATA

```
%% Results
clc
clear
close all
%% Reynolds Number
D_area = 0.75^2; % [m^2] Design CSA
CSA_unitcell = 0.015^2; % [m^2] Basic CSA
n_unitcell_CSA = D_area/CSA_unitcell; % number of unit cells CSA
D_pipe = 19e-3; % [m]
L_ = 0.015; % [unit cell length]
max_vel = 1.5*1.9; % [m/s]
mu_water = 0.0003453; %[Ns/m^2]dynamic viscosity
k_water = 0.6; % [W/m^2K]
rho = 997; %[kg/m^3]
V = 2490.1203; % [mm^3] Total volume of the fluid domain
tot_SA = 1361.0127; % [mm^2] Total wetted surface area of TPMS
Dh_tot = 10^-3*((4*V)/tot_SA);
Dh = Dh_tot/2;
A = 0.000087719; % [m^2] cross sectional area
U = linspace(0,max_vel,100);
m_dot = U.*rho.*A;
Re = rho.*Dh.*U/mu_water;
Re_laminar = [];
Re_turbulent = [];
U_res = [0.0288, 0.2015, 0.4318];
m_dot_res = [0.0025,0.0176,0.0378];
k_ti = 5.5;
for i = 20:5:100
    U_res = [U_res, U(i)];
end
for j = 20:5:100
    m_dot_res = [m_dot_res, m_dot(j)];
    j = j+5;
end
Re_res = rho.*Dh.*U_res/mu_water;

%% Data
cd('C:\Users\Derrick Thompson\OneDrive - Naval Postgraduate School\Documents\
Thesis\Results')
% data = load('dt-rfile.csv')
mydirs = dir;
meanHTC = [];
maxHTC = [];
minHTC = [];
meandpdx = [];
maxdpx = [];
mindpdx = [];
meanf = [];
maxf = [];
minf = [];
```

```

meanTI = [];
maxTI = [];
minTI = [];
% loop for rheat trasnfer coefficient
for i = 3:size(mydirs)

    curdir = mydirs(i).name;
    HTC = dlmread([curdir '\htc-rfile.out'], ' ', 3, 0);
    HTC = abs(HTC(150:600, :));
    meanHTC(i) = mean(HTC(:, 2));
    maxHTC(i) = max(HTC(:, 2));
    minHTC(i) = min(HTC(:, 2));
    figure(1);
    HTC_sub = subplot(6, 4, i-2);
    plot(HTC(:, 1), HTC(:, 2))
    xlabel('Iteration')
    ylabel('HTC (W/(m^2K))')
    title(HTC_sub, 'HTC vs. Iteration')

    dpdx = dlmread([curdir '\dpdx-rfile.out'], ' ', 3, 0);
    dpdx = abs(dpdx(150:600, :));
    meandpdx(i) = mean(dpdx(:, 2));
    maxdpdx(i) = max(dpdx(:, 2));
    mindpdx(i) = min(dpdx(:, 2));
    figure(2);
    dpdx_sub = subplot(6, 4, i-2);
    plot(dpdx(:, 1), dpdx(:, 2))
    xlabel('Iteration')
    ylabel('dpdx')
    title(dpdx_sub, 'DpDx vs. Iteration')

    f = dlmread([curdir '\frictionfactor-rfile.out'], ' ', 3, 0);
    f = abs(f(150:600, :));
    meanf(i) = mean(f(:, 2));
    maxf(i) = max(f(:, 2));
    minf(i) = min(f(:, 2));
    figure(3);
    f_sub = subplot(6, 4, i-2);
    plot(f(:, 1), f(:, 2))
    xlabel('Iteration')
    ylabel('f')
    title(f_sub, 'f vs. itaeration')

    TI = dlmread([curdir '\volaveti-rfile.out'], ' ', 3, 0);
    TI = abs(TI(150:600, :));
    meanTI(i) = mean(TI(:, 2));
    maxTI(i) = max(TI(:, 2));
    minTI(i) = min(TI(:, 2));
    figure(4);
    TI_sub = subplot(6, 4, i-2);

```

```

plot(TI(:,1),TI(:,2))
xlabel('Iteration')
ylabel('TI')
title(TI_sub,'TI vs. Iteration')

if i >= 23
figure(5)
HTCR_sub = subplot(3,1,i-22);
plot(HTC(:,1),HTC(:,2))
xlabel('Iteration')
ylabel('HTC (W/(m^2K))')
title(HTCR_sub,'HTC Roughness')

figure(6);
TIR_sub = subplot(3,1,i-22);
sgtitle('Turbulence Intensity')
plot(TI(:,1),TI(:,2))
xlabel('Iteration')
title(TIR_sub,'TI Roughness')

figure(7);
fR_sub = subplot(3,1,i-22);
sgtitle('Friction Factor')
plot(f(:,1),f(:,2))
xlabel('Iterations')
title(fR_sub,'f Roughness')

figure(8);
dpdxR_sub = subplot(3,1,i-22);
sgtitle('Pressure Drop')
plot(dpdx(:,1),dpdx(:,2))
xlabel('Iteration')
title(dpdxR_sub,'DpDx Roughness')
end

% pause(1)
end
%% Parse Data
% Roughness
meanHTC_R = meanHTC(23:25);
maxHTC_R = maxHTC(23:25);
minHTC_R = minHTC(23:25);
meanf_R = meanf(23:25);
maxf_R = maxf(23:25);
minf_R = minf(23:25);
meandpdx_R = meandpdx(23:25)./0.015;
maxdpdx_R = maxdpdx(23:25)./0.015;
mindpdx_R = mindpdx(23:25)./0.015;
meanTI_R = meanTI(23:25).*100;
maxTI_R = maxTI(23:25).*100;
minTI_R = minTI(23:25).*100;
% Re

```

```

meanHTC = meanHTC(3:22);
maxHTC = maxHTC(3:22);
minHTC = minHTC(3:22);
meanf = meanf(3:22);
maxf = maxf(3:22);
minf = minf(3:22);
meandpdx = meandpdx(3:22)./0.015;
maxdpx = maxdpx(3:22)./0.015;
mindpdx = mindpdx(3:22)./0.015;
meanTI = meanTI(3:22).*100;
maxTI = maxTI(3:22).*100;
minTI = minTI(3:22).*100;

%% Bare
Pr = 6.9;
Nu_bare = 0.023.*(Re_res.^0.8).*(Pr^0.4);
bare_HTC = Nu_bare*k_water/D_pipe;

%% PLOt
figure(9)
errorbar(Re_res,meanHTC,minHTC,maxHTC,'o')
xlabel('Reynolds Number')
ylabel('Heat Transfer Coefficient (W/m^2K)')
title('Heat Transfer Coefficient')
hold on
plot(Re_res,bare_HTC,'-rx')
legend('TPMS HTC','Bare Tube HTC')
hold off
figure(10)
errorbar(Re_res,meandpdx,mindpdx,maxdpx,'o')
xlabel('Reynolds Number')
ylabel('DpDx (pa/m)')
title('Pressure Drop')
figure(11)
errorbar(Re_res,meanf,minf,maxf,'o')
xlabel('Reynolds Number')
ylabel('Friction Factor')
title('Friction Factor')
ylim([0 1])
figure(12)
errorbar(Re_res,meanTI,minTI,maxTI,'o')
xlabel('Reynolds Number')
ylabel('Turbulence Intensity %')
title('Turbulence Intensity')
ylim([0 100])

%% Surcace area needed
SA = tot_SA*0.5*10^-6; % [surface area for one domain]
R_h = 2.7/(minHTC);
R_k = 1.5e-3/(k_ti);
U_HT = 1./(R_h + R_k);
q = 5.861e6;
LMTD = 46.3;
A_needed = q./U_HT./LMTD;

```

```

% Get coefficients of a line fit through the data.
f_SA = fit(Re_res', A_needed', 'exp2');
f_U = fit(Re_res', U_HT', 'exp2');
xFit = linspace(min(Re_res), max(Re_res), 1000);
figure(13)
plot(f_SA, Re_res, A_needed, '-kx')
text(min(Re_res)+0.5e4, max(A_needed)-10, 'f(x) = 106.5*exp(-8.8874e-4*x) + 44.81*exp(-1.549e-5*x)')
xlabel('Reynolds Number')
ylabel('Heat Transfer Area (m^2)')
title('Heat Transfer Area')
legend('Raw Data', 'Line of Best Fit')
figure(14)
plot(f_U, Re_res, U_HT, '-kx')
text(min(Re_res), max(U_HT), 'f(x) = 3337*exp(8.258e-6*x) - 2438*exp(-2.535e-4*x)')
xlabel('Reynold Number')
ylabel('Overall Heat Transfer Coefficient (W/m^2K)')
title('Overall heat Transfer Coefficient')
legend('Raw Data', 'Line of Best Fit', 'Location', 'southeast')
%% Pressure Drop
disk_area = SA*n_unitcell_CSA;
number_model_dp = A_needed/disk_area; % [number of CSA]
L_model_HX = 0.015*number_model_dp; % [length of HX]
Pressure_Drop = maxdpdx.*L_model_HX*10^-3;
% %% Bare
% Re_bare = U_res*rho*D_pipe/mu_water;
% Re_bare = linspace(0, max(Re_res), 20);
% U_bare = Re_bare*mu_water/(rho*D_pipe);
% f_bare = 64./(Re_bare(1:2)); % laminar f
% k_ss = 0.003*10^-3;
% A_p = (k_ss.^1.1098/2.8257)+(7.149./Re_bare(3:end)).^0.8981;
% i_f = -2*log10((k_ss./3.7065)-(5.0452./Re_bare(3:end).*log10(A_p)));
% f_bare_t = 1./(i_f.^2); % Turbulent f
% f_bare = [f_bare, f_bare_t];
% dp_bare = (f_bare.*(3./D_pipe).*((rho.*(U_bare.^2))./2)).*10^-3;
figure(15)
plot(Re_res, Pressure_Drop, '-kx')
% hold on
% plot(Re_bare, dp_bare, '-rx')
% legend('TPMS Dp', 'Bare Tube Dp')
xlabel('Reynolds Number')
ylabel('Pressure Drop (kPa)')
title('Full Size Model Pressure Drop')

%% Roughness
Ra = linspace(7.869e-6, 29.94e-6, 3);
Ks = Dh*(18.*(Ra./Dh) - 0.05*Ra/Dh);
dt = 1e-3/U_res(10);

%% PLOT
figure(16)
errorbar(Ra, meanHTC_R, minHTC_R, maxHTC_R, 'o')

```

```

hold on
errorbar(0,meanHTC(10),minHTC(10),maxHTC(10),'o')
hold off
legend('Roughness Modeled','No Roughness Modeled')
xlabel('Surface Roughness (m)')
ylabel('Heat Transfer Coefficeint (W/m^2K)')
title('Heat Transfer Coefficient Dependent on Roughness')
xlim([-1e-5 4e-5])
figure(17)
errorbar(Ra,meanf_R,minf_R,maxf_R,'o')
hold on
errorbar(0,meanf(10),minf(10),maxf(10),'o')
hold off
legend('Roughness Modeled','No Roughness Modeled')
xlabel('Surface Roughness (m)')
ylabel('Friction Factor')
title('Friction Factor Dependent on Roughness')
xlim([-1e-5 4e-5])
figure(18)
errorbar(Ra,meandpdx_R,mindpdx_R,maxdpdx_R,'o')
hold on
errorbar(0,meandpdx(10),mindpdx(10),maxdpdx(10),'o')
hold off
legend('Roughness Modeled','No Roughness Modeled')
xlabel('Surface Roughness (m)')
ylabel('Pressure Drop per Unit Length (Pa/m)')
title('Pressure Drop Dependent on Roughness')
xlim([-1e-5 4e-5])
figure(19)
errorbar(Ra,meanTI_R,minTI_R,maxTI_R,'o')
hold on
errorbar(0,meanTI(10),minTI(10),maxTI(10),'o')
hold off
legend('Roughness Modeled','No Roughness Modeled')
xlabel('Surface Roughness (m)')
ylabel('Turbuelnce Intensity %')
title('Turbulence Intensity Dependent on Roughness')
xlim([-1e-5 4e-5])

%% NU Number
Nu_mean = meanHTC.*Dh./k_water;
Nu_max = maxHTC.*Dh./k_water;
Nu_min = minHTC.*Dh./k_water;
% Plot
figure(20)
errorbar(Re_res,Nu_mean,Nu_min,Nu_max,'o')
xlabel('Reynolds Number')
ylabel('Nusselt's Number')
title('Nu Number')

%% NU Number Roughness
Nu_mean_R = meanHTC_R.*Dh./k_water;
Nu_max_R = maxHTC_R.*Dh./k_water;
Nu_min_R = minHTC_R.*Dh./k_water;

```

```

% Plot
figure(21)
errorbar(Ra,Nu_mean_R,Nu_min_R,Nu_max_R,'o')
hold on
errorbar(0,Nu_mean(10),Nu_min(10),Nu_max(10),'o')
hold off
xlabel('Surface Roughness')
ylabel('Nusselt's Number')
title('Nu Number Dependent on Roughness')
xlim([-1e-5 4e-5])
%% Length
figure(22)
plot(Re_res,L_model_HX,'-kx')
xlabel('Reynolds Number')
ylabel('Length of Hypothetical Heat Exchanger')
title('Heat Exchanger Length')
%% Save
cd('C:\Users\Derrick Thompson\OneDrive - Naval Postgraduate School\Documents\
Thesis\Figures')
for j = 1:22
h = get(gca(j),'title');
titles = get(h,'string');
saveas(figure(j),sprintf('%s.png',titles));
end

cd('C:\Users\Derrick Thompson\OneDrive - Naval Postgraduate School\Documents\
Thesis')

```

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF REFERENCES

- [1] J. Katzel, “Heat exchanger basics,” Plant Engineering, April 1, 2000. Available: <https://www.plantengineering.com/articles/heat-exchanger-basics/>
- [2] Manufacturing Guide, “Tube drawing.” Accessed: Dec. 13, 2022. Available: <https://www.manufacturingguide.com/en/tube-drawing>
- [3] Thomas, “Different types of casting processes used in manufacturing.” Accessed: Dec. 14, 2022. Available: <https://www.thomasnet.com/articles/custom-manufacturing-fabricating/types-of-casting-processes/>
- [4] TWI Global, “What is welding? – Definition, processes and types of welds.” Accessed: Dec. 13, 2022. Available: <https://www.twi-global.com/technical-knowledge/faqs/what-is-welding>
- [5] T. Dixit, E. Al-HAjri, M. Paul, P. Nithiarasu and S. Kumar, “High performance, microarchitected, compact heat exchanger enabled by 3D printing,” *Applied Thermal Engineering*, vol. 228, Jun. 2022. Available: <https://doi.org/10.1016/j.applthermaleng.2022.118339>
- [6] A. Foley, “How to model a shell and tube heat exchanger,” COMSOL, blog, September 11, 2013. Available: <https://www.comsol.com/blogs/how-model-shell-and-tube-heat-exchanger/>
- [7] D. Conradie, “3D printed heat exchanger uses gyroid infill for cooling,” Hackaday, October 31, 2022. Available: <https://hackaday.com/2022/10/31/3d-printed-heat-exchanger-uses-gyroid-infill-for-cooling/>
- [8] M. Mehta, “Evolution of heat exchangers,” Kinam, February 12, 2021. Available: <https://kinam.in/evolution-of-heat-exchangers-2/#:~:text=The%20recorded%20history%20of%20Heat,Dracke%20of%20Germany%20in%201878>
- [9] International Atomic Energy Agency, “Status Report 81 – Advanced Passive PWR (AP1000),” 2011. Available: <https://aris.iaea.org/PDF/AP1000.pdf>
- [10] N. Connor, “What is thermal efficiency of heat engines,” Thermal Engineering, May 22, 2019. Available: <https://www.thermal-engineering.org/what-is-thermal-efficiency-of-heat-engines-examples-definition/>
- [11] W. Martin, “Nuclear power,” Encyclopedia Britannica, May 9, 2023. Available: <https://www.britannica.com/technology/nuclear-power>
- [12] R. K. Shah and D. P. Sekulić, *Fundamentals of Heat Exchanger Design*. Hoboken, NJ, USA: John Wiley & Sons, Inc., 2003. Available: <https://doi.org/10.1002/9780470172605>

- [13] A. Fraas, *Heat Exchanger Design*. New York, NY, USA: John Wiley & Sons, Inc., 1988.
- [14] H. Peng, F. Gao and W. Hu, “Design, modeling and characterization of triply periodic minimal surface heat exchangers with additive manufacturing,” in *30th Annual International Solid Freeform Fabrication Symposium*, 2019. Available: <https://utw10945.utweb.utexas.edu/sites/default/files/2019/20Characterization%20of%20Triply%20Pe.pdf>
- [15] E. Wrokbook, “What is Rankine cycle? Process, components, diagram & efficiency,” *Electrical Workbook*, April 15, 2022. Available: <https://electricalworkbook.com/rankine-cycle/>
- [16] K. V. Wong and A. Hernandez, “A review of additive manufacturing,” *International Scholarly Research Network*, vol. 2012, Aug. 2012. Available: <https://doi.org/10.5402/2012/208760>
- [17] D. Kovalchuk, V. Melnyk, I. Melnyk, D. Savvakina, O. Dekhtyar, O. Stasiuk and P. Markovsky, “Microstructure and properties of Ti-6Al-4V articles 3D-printed with co-axial electron beam and wire technology,” *Journal of Materials Engineering and Performance*, pp. 5307–5322, Apr. 2021. Available: <https://doi.org/10.1007/s11665-021-05770-9>
- [18] United Performance Metals, “Which materials are the best fit for your heat exchanger application.” Accessed: Dec. 12, 2022 Available: <https://www.upmet.com/sites/default/files/whitepapers/heat-exchanger-materials.pdf>.
- [19] H. D. Nguyen, A. Pramanik, A. Basak, Y. Dong, C. Prakash, S. Debnath, S. Shankar, I. Jawahir, S. Dixit and D. Buddhi, “A critical review on additive manufacturing of Ti-6Al-4V alloy: microstructure and mechanical properties,” *Journal of Materials Research and Technology*, vol.18, pp. 4641–4661, Apr. 2022. Available: <https://doi.org/10.1016/j.jmrt.2022.04.055>
- [20] J. Bamberg, M. Spies, A. Dillhöfer, T. Hess and H. Rieder, “Online monitoring of additive manufacturing processes using ultrasound,” in *11th European Conference on Non-Destructive Testing*, Prague, 2014. Available: https://www.ndt.net/events/ECNDT2014/app/content/Paper/259_Spies.pdf
- [21] Horizon Technology, “How metal powder is made: Step 1 of the powder metallurgy process,” April 27, 2020. Available: <https://www.horizontechnology.biz/blog/how-are-metal-powders-made-powder-metallurgy-process>

- [22] L. Langnau, “How to select the right metal additive manufacturing machine for your needs,” Make Parts Fast, October 3, 2018. Available: <https://www.makepartsfast.com/how-to-select-the-right-metal-additive-manufacturing-machine-for-your-needs/#:~:text=For%20the%20most%20part%2C%20powder,height%20of%20250%2D300%20mm>
- [23] A. Huckstepp, “Digital alloys’ guide to metal additive manufacturing – Part 8,” DigitalAlloys, May 6, 2019. Available: <https://www.digitalalloys.com/blog/powder-bed-fusion/>
- [24] E. Utley, “An introduction to designing for metal 3D printing,” Dassault Systems, June 12, 2017. Available: <https://blogs.solidworks.com/solidworksblog/2017/06/introduction-designing-metal-3d-printing.html>
- [25] W. Li, G. Yu and Z. Yu, “Bioinspired heat exchangers based on triply periodic minimal surfaces for supercritical CO₂ cycles,” *Applied Thermal Engineering*, vol. 179, Oct. 2020. Available: <https://doi.org/10.1016/j.applthermaleng.2020.115686>
- [26] V. B. Slaughter, “Method of using minimal surfaces and minimal skeletons to make heat exchanger components,” U.S. Patent 7.866,377 B2, Jan. 11, 2011. Available: <https://patents.google.com/patent/US7866377B2/en>
- [27] F. M. White, *Viscous Fluid Flow*. Boston, MA, USA: McGraw-Hill, 1991.
- [28] M. Ferro, “Experimental study on turbulent pipe flow,” M.S. thesis, Linné Flow Cent., Royal Int. of Tech., Stockholm, SE, 2012. Available: <http://www.diva-portal.org/smash/get/diva2:558033/FULLTEXT01.pdf>
- [29] P. Zelinski, “3D printed heat exchanger uses gyroids for better cooling,” Additive Manufacturing, March 3, 2022. Available: <https://www.additivemanufacturing.media/articles/3d-printed-heat-exchanger-uses-gyroids-for-better-cooling-the-cool-parts-show-43>.
- [30] B. Spalding and J. Taborek, *Heat Exchanger Design Handbook*. London, UK: Hemisphere Publishing Corporation, 1983.
- [31] T.Kuppan, *Heat Exchanger Design Handbook*. New York, NY, USA: Marcel Dekker, Inc., 2000.
- [32] G. Electric, “Concept laser X line machine.” Accessed: Apr. 15, 2023. Available: <https://www.ge.com/additive/additive-manufacturing/machines/dmlm-machines/x-line-2000r>
- [33] ASM Aerospace Specifications Metals Inc., “Titanium Ti-6Al-4V (Grade 5), Annealed.” Accessed: Mar. 14, 2023. Available: <https://asm.matweb.com/search/SpecificMaterial.asp?bassnum=mtp641>

- [34] P. Morrow, “Postprocessing steps and costs for metal 3D printing,” Additive Manufacturing Media, May 21, 2018. Available: <https://www.additivemanufacturing.media/articles/postprocessing-steps-and-costs-for-metal-3d-printing>
- [35] P. Morrow, “Plotting a pathway to profitable AM,” Additive Manufacturing Media, May 11, 2021. Available: <https://www.additivemanufacturing.media/articles/plotting-a-pathway-to-profitable-am>
- [36] S. N. A. Yusof, Y. Asako, N. A. C. Sidik, S. B. Mohamed and W. M. A. A. Japar, “A short review on RANS turbulence models,” *CFD Letters*, vol. 12, no. 11, pp. 83–96, Nov. 2020. Available: <https://akademiabaru.com/submit/index.php/cfdl/article/view/2267>
- [37] J. Gorman, S. Bhattacharyya, L. Cheng and J. Abraham, “Turbulence models commonly used in CFD,” *Applications of Computational Fluid Dynamics Simulation and Modeling*, Apr. 2022. Available: <https://www.intechopen.com/chapters/78287>
- [38] *ANSYS Fluent Theory Guide*, vol. 15, ANSYS Inc., Canonsburg, PA, USA, 2023.
- [39] D. Deligianni, N. Katsala, S. Ladas, D. Sotiropoulou, J. Amedee and Y. Missirlis, “Effect of surface roughness of the titanium alloy Ti-6Al-4V on human bone marrow cell response and on protein adsorption,” *Biomaterials*, vol. 22, no. 11, pp. 1241–1251, Jun. 2001. Available: [https://doi.org/10.1016/S0142-9612\(00\)00274-X](https://doi.org/10.1016/S0142-9612(00)00274-X)
- [40] K. Nayar, D. Panchanathan, G. McKinley and J. L. V, “Thermophysical properties of seawater: A review and new correlations that include pressure dependence,” *Desalination*, vol. 390, pp. 1–24, Jul. 2016. Available: <https://doi.org/10.1016/j.desal.2016.02.024>
- [41] K. Bartsch, B. Bossen, W. Chaudhary, M. Landry and D. Herzog, “Thermal conductivity of Ti-6Al-4V in laser powder bed fusion,” *Frontiers in Mechanical Engineering*, vol. 8, Jul. 2022. Available: <https://doi.org/10.3389/fmech.2022.830104>
- [42] Proto300, *EOS Titanium Ti64*, 2011. Available: https://proto3000.com/wp-content/uploads/2018/03/MSDS-Titanium_Ti64.pdf
- [43] ThermExcel, “Physical characteristics of water,” 2003. Available: https://www.thermexcel.com/english/tables/eau_atm.htm
- [44] EOS, “EOS M 290 – Mid-size 3D printing.” Accessed: Dec. 14, 2022. Available: <https://www.eos.info/en/industrial-3d-printer/metal/eos-m-290>

- [45] I. Sadrehaghighi, “Mesh Sensitivity & Mesh Independence Study,” CFD Open Series, Annapolis, MD, USA, 2021. Available: <http://dx.doi.org/10.13140/RG.2.2.34847.51365/2>

THIS PAGE INTENTIONALLY LEFT BLANK

INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center
Ft. Belvoir, Virginia
2. Dudley Knox Library
Naval Postgraduate School
Monterey, California



DUDLEY KNOX LIBRARY

NAVAL POSTGRADUATE SCHOOL

WWW.NPS.EDU

WHERE SCIENCE MEETS THE ART OF WARFARE