



NAVAL POSTGRADUATE SCHOOL

MONTEREY, CALIFORNIA

THESIS

ADDITIVELY MANUFACTURED SPIRALLY CORRUGATED HEAT EXCHANGER TUBES

by

Noah H. Byrne

September 2023

Thesis Advisor:

Co-Advisor:

Second Reader:

Walter C. Smith

Sean O'Brien,

Naval Nuclear Lab

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 September 2023	3. REPORT TYPE AND DATES COVERED Master's thesis		
4. TITLE AND SUBTITLE ADDITIVELY MANUFACTURED SPIRALLY CORRUGATED HEAT EXCHANGER TUBES			5. FUNDING NUMBERS	
6. AUTHOR(S) Noah H. Byrne				
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) The Naval Nuclear Propulsion Program is interested in exploring additive manufacturing (AM) technology to improve system performance, reduce component cost, and expand the vendor base. To support that goal, this study examines additively manufactured spirally corrugated tubes for use in a seawater-to-feedwater heat exchanger. Ti-6Al-4V was selected as the material for the tubes due to its compatibility with AM technology, corrosion resistance, and strength. Heat exchangers can be improved by increasing the heat transfer surface area, which additive manufacturing facilitates through the ability to manufacture complicated geometries that are otherwise difficult to manufacture. Analysis of the proposed tubes geometry is completed using experimental correlations and computational fluid dynamics (CFD) software to analyze flow and heat transfer through a single tube and over a tube bundle. Thermal and hydraulic performance of the tubes are then applied to a full-sized heat exchanger employing the proposed tubes utilizing limiting parameters to achieve a preliminary sizing and cost analysis. It was determined that the proposed tube geometry has more than double the heat transfer coefficient compared to traditionally manufactured bare tubes. This study determined that the spirally corrugated tubes have the potential to decrease overall unit length by fifty percent.				
14. SUBJECT TERMS heat transfer, heat exchanger, additive manufacturing, AM, thermal hydraulic, mechanical stresses, design, 3D printing, enhanced heat transfer, computational fluid dynamics, CFD			15. NUMBER OF PAGES 159	
			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.

**ADDITIVELY MANUFACTURED SPIRALLY CORRUGATED HEAT
EXCHANGER TUBES**

Noah H. Byrne
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
September 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

The Naval Nuclear Propulsion Program is interested in exploring additive manufacturing (AM) technology to improve system performance, reduce component cost, and expand the vendor base. To support that goal, this study examines additively manufactured spirally corrugated tubes for use in a seawater-to-feedwater heat exchanger. Ti-6Al-4V was selected as the material for the tubes due to its compatibility with AM technology, corrosion resistance, and strength. Heat exchangers can be improved by increasing the heat transfer surface area, which additive manufacturing facilitates through the ability to manufacture complicated geometries that are otherwise difficult to manufacture. Analysis of the proposed tubes geometry is completed using experimental correlations and computational fluid dynamics (CFD) software to analyze flow and heat transfer through a single tube and over a tube bundle. Thermal and hydraulic performance of the tubes are then applied to a full-sized heat exchanger employing the proposed tubes utilizing limiting parameters to achieve a preliminary sizing and cost analysis. It was determined that the proposed tube geometry has more than double the heat transfer coefficient compared to traditionally manufactured bare tubes. This study determined that the spirally corrugated tubes have the potential to decrease overall unit length by fifty percent.

THIS PAGE INTENTIONALLY LEFT BLANK

TABLE OF CONTENTS

I.	MOTIVATION	1
II.	INTRODUCTION.....	3
A.	HEAT EXCHANGER DESIGN.....	5
1.	Shell and Tube Heat Exchangers.....	6
2.	Heat Transfer Augmentation.....	8
B.	COMPUTATIONAL FLUID DYNAMICS.....	10
1.	Fluid Flow Characterization.....	11
2.	Turbulence Models	13
C.	ADDITIVE MANUFACTURING TECHNOLOGY.....	16
1.	Current State of AM.....	17
2.	Cost Analysis	17
III.	METHODS	19
A.	GEOMETRIC PARAMETERIZATION	20
B.	PRELIMINARY PRESSURE DROP CALCULATIONS.....	22
C.	SELECTED TUBE GEOMETRY.....	23
D.	COMPUTATIONAL FLUID DYNAMICS SET UP.....	24
1.	Meshing.....	25
2.	Boundary Conditions.....	26
3.	High-Performance Computing (HPC)	29
E.	HEAT TRANSFER QUANTIFICATION.....	30
IV.	ANALYSIS	33
A.	TEST CASE – BARE TUBE.....	34
B.	EIGHT-START SPIRALLY CORRUGATED TUBE.....	37
1.	Shell-side Flow Investigation	46
C.	EIGHT-START SPIRALLY CORRUGATED TUBE WITH TWISTED TAPE INSERT	54
D.	TURBULENCE KINETIC ENERGY COMPARISON.....	58
V.	RESULTS	61
VI.	CONCLUSION	65
	APPENDIX A. MATLAB SCRIPT.....	69

APPENDIX B. CFX COMMAND LINE (CCL) FILES	73
A. MASS FLOW INLET WITH ZERO PRESSURE OUTLET CONDITION	73
B. IMPORTED VELOCITY PROFILE WITH ZERO PRESSURE OUTLET CONDITION	93
C. HEAT TRANSFER SCENARIO.....	114
 APPENDIX C. SAMPLE BATCH SHELL SUBMISSION SCRIPT	133
 LIST OF REFERENCES.....	135
 INITIAL DISTRIBUTION LIST	139

LIST OF FIGURES

Figure 1.	Basic heat exchanger illustration	5
Figure 2.	Shell-and-tube heat exchanger (STHE) illustration. Source: Adapted from [5].	6
Figure 3.	Tube heat transfer surface area illustration	7
Figure 4.	Common tube layout patterns. Source: [5].	8
Figure 5.	Front and side view of corrugated tube used in Barba’s experiment (units in centimeters). Source: [6].	9
Figure 6.	Several examples of twisted tape inserts. Source: [5].	10
Figure 7.	Flow development through a tube. Source: [12].	12
Figure 8.	MATLAB generated cross-section based on inner diameter, start diameter, number of starts, start thickness, and tube thickness	21
Figure 9.	MATLAB parameterized geometry implemented in SolidWorks CAD software	22
Figure 10.	Results of calculated pressure drops of spirally corrugated tubes with different numbers of starts	24
Figure 11.	Isometric section plane view of fluid volume marked in red.	25
Figure 12.	Results of finite element analysis on eight-start tube	33
Figure 13.	XY-plane velocity vector plot of shell side flow in bare case	34
Figure 14.	XY-plane velocity vector plot of tube side flow in bare case.	35
Figure 15.	XY-plane velocity contour for simulated flow over a triangular pitch bundle of smooth tubes	36
Figure 16.	YZ-plane temperature contour for bare case	37
Figure 17.	Centerline velocity through 500 mm section of spirally corrugated tubing, flow traveling from left to right	39
Figure 18.	Fluid flow development through 500 mm section of tube	40

Figure 19.	Fully developed flow achieved through the imported velocity profile condition	41
Figure 20.	Centerline velocity contour from completed heat transfer simulation.....	42
Figure 21.	Vector plot of XY plane shell side fluid velocity at a z-length of 250 mm	43
Figure 22.	XY-plane slice tube velocity contour	44
Figure 23.	YZ-plane centerline temperature contour for the spirally corrugated case.....	45
Figure 24.	XY-plane temperature contour for the hot inlet, cold outlet (left) and cold inlet, hot outlet (right)	45
Figure 25.	Visual representation of spirally corrugated tube bundle shell-side flow unit cell	47
Figure 26.	XY-plane velocity contour at 250 mm z-length for shell-side flow in the spirally corrugated cases	48
Figure 27.	YZ-plane centerline velocity contour for shell-side flow in the spirally corrugated cases	49
Figure 28.	XY-plane contour of turbulence kinetic energy at outlet of spirally corrugated tube.....	50
Figure 29.	XY-plane contour of turbulence kinetic energy at outlet of bare tube	50
Figure 30.	Shell-side flow average turbulence kinetic energy through tube.....	51
Figure 31.	Shell-side flow maximum turbulence kinetic energy through tube	52
Figure 32.	Shell-side flow minimum turbulence kinetic energy through tube.....	53
Figure 33.	Centerline turbulence kinetic energy for spirally corrugated tube bundle flow	54
Figure 34.	YZ-plane centerline velocity contour in twisted tape case	55
Figure 35.	XY-plane velocity vector plot at 250 mm z-length for twisted tape case.....	56
Figure 36.	YZ-plane centerline temperature contour for the twisted tape case	57
Figure 37.	XY-plane temperature contour for the hot inlet, cold outlet (left) and cold inlet, hot outlet (right) for twisted tape case	57

Figure 38.	Comparison of turbulence kinetic energy along y-axis hallway through the tubes.....	59
Figure 39.	Comparison of turbulence kinetic energy through length of tube for bare and spirally corrugated cases	60
Figure 40.	Total area as a function of length and number of tubes (bare case)	62
Figure 41.	Total area as a function of length and number of tubes (spirally corrugated cases).....	63

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF TABLES

Table 1.	Critical parameters	19
Table 2.	Material properties of titanium alloy Ti-6Al-4V	29
Table 3.	Results of mesh sensitivity study	38
Table 4.	Results from all simulations conducted	61

THIS PAGE INTENTIONALLY LEFT BLANK

ACKNOWLEDGMENTS

Thank you to Dr. Smith and Sean for being available at all hours to help me with whatever questions I could have had. Their help was crucial in getting this work done and they taught me valuable lessons about knowledge and project management. I would also like to thank my family, the core four, without them none of this would have ever been possible. They have supported me every step of the way despite me being on the other side of the country and I could not be more thankful for them.

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF ACRONYMS AND ABBREVIATIONS

AM	additive manufacturing
CAD	computer aided design
CCL	CFX Command Line
CFD	computational fluid dynamics
DMLS	direct metal laser sintering
EBAM	electron beam additive manufacturing
FEA	finite element analysis
GUI	graphical user interface
HPC	high-performance computing
LES	large-eddy simulation
PBF	powder bed fusion
STHE	shell-and-tube heat exchanger
TEMA	Tubular Exchange Manufacturers Association
TKE	turbulence kinetic energy

THIS PAGE INTENTIONALLY LEFT BLANK

I. MOTIVATION

The United States Navy operates hundreds of ships, submarines and aircraft. A tremendous amount of energy is required to fulfill the Navy's mission. Surface ships utilize hybrid diesel and gas turbines for propulsion and electric systems while aircraft carriers and submarines use pressurized water reactors to drive the power plant steam cycle. The second law of thermodynamics states that no process can be one-hundred percent efficient—every cycle has a rated percent of thermal energy that can be transferred to mechanical energy. The remaining heat must be dispensed with by rejecting it to the environment, recuperating it, or using the excess heat for other processes.

The Westinghouse AP1000, a standardized model for pressurized water reactors—a similar type of reactor to the ones used by the Navy—has a total heat output of 3415 MW and a thermal efficiency of 32.5% where 67.5% of the thermal energy is rejected from the system [1]. Heat exchanger technology is crucial in ensuring efficient use or rejection of excess thermal energy. Optimized heat exchangers can be made smaller and more efficiently reject the tremendous amount of heat generated by a power plant. In addition to rejecting the proper amount of heat, an optimized heat exchanger must also be easily serviceable to allow for routine cleaning and part replacement without significantly affecting the operation of the power plant.

Improvements in heat exchanger efficiency can significantly reduce component costs, which is of key interest to the Navy. For instance, the Columbia class submarine, the newest addition to the submarine fleet, is projected to cost in excess of fifteen billion—any reduction in component costs through the use of optimized heat exchangers would be of value to the Navy [2].

The Navy is interested in exploring the use of additive manufacturing (AM) to improve performance and reduce component and maintenance costs. Of particular concern to Navy is the design of a heat exchanger utilizing AM created components. The goal of the additively manufactured components is to create a more efficient, more easily serviceable system that can either be completely removed, replaced easily, or even allow

for on-line maintenance. Utilizing AM components would also expand the Navy's vendor base. This research seeks to examine a heat exchanger tube design that can be produced with Ti-6Al-4V metal additive manufacturing. This study will be an examination of the performance of the tubes with regard to pressure drop and heat transfer. Included will be a cost analysis for the AM parts compared to commercially available tubes. To do this, the thermal hydraulic performance of a tube design will be examined and a cost analysis comparing the cost of additive manufacturing to traditional manufacturing. The cost analysis will allow for the weighing of the benefits associated with AM technology compared to the expected increased cost of raw materials to determine the viability of the tube design. The scope of this study will be bounded by the walls of the heat exchanger—there will be no consideration for fluid head development, external piping, or the vessel construction. In order to analyze the design, 3D computer aided design (CAD) software and computational fluid dynamics (CFD) will be used. Should the first iteration of the tube design prove successful with respect to the critical parameters provided by the Navy, future work will be conducted to optimize the design and broaden the scope of the heat exchanger to potential construction and implementation.

II. INTRODUCTION

Heat exchangers play a critical role in almost every industrial field—it is necessary to be able to extract energy from various fluid flows efficiently. Improvements in heat exchanger technology serve to reduce component costs by over 50% [3]. In addition to saving costs, improving the efficiency of heat exchangers allows for their size to be reduced--furthering the applications they can be deployed in. Heat exchanger size is of critical importance in applications where space is limited such as onboard a submarine. In this application, the reduction in size can allow for ease of component replacement saving both money and time.

Basic shell-and-tube heat exchanger technology arose in the late 19th century to accommodate the growing need for energy extraction in power plants. These plants were comprised mostly of large coal-fired boilers that operated at incredibly high temperatures and pressures. In order to effectively extract the energy from these processes, a heat exchanger was required. Heat exchangers extract energy from a high temperature fluid by allowing the hot fluid to transfer its thermal energy to a lower temperature fluid. This can be extremely useful for rejecting excess thermal energy. Shell-and-tube heat exchangers continued development through the 1920s where the booming oil industry needed heat exchangers to process crude oil. At this time, the primary concern of heat exchanger technology was structural—the highly pressurized fluid caused extreme stresses in tube sheets that could lead to catastrophic failures. Once the structural aspects of heat exchanger technology became more standardized (through the use of Tubular Exchange Manufacturers Association (TEMA) standards for example), other aspects of the heat exchangers such as thermal efficiency and pumping power could be optimized. The fluid flow characteristics of shell-and-tube heat exchangers were not widely known until the 1960s, when an explosion of experimental work analyzed the thermal-hydraulic optimization of heat exchangers. The emergence of CFD enables more in-depth analysis of the heat transfer and flow in heat exchangers, allowing for the exploration of more complex geometries such as corrugated tubing.

Heat exchangers are used today in almost every application that requires the extraction of heat—power generation, electronics, and HVAC to name a few. Heat exchangers are rated according to several critical parameters including overall heat transfer capability, size, pressure drop, and pumping power. Improvements in any one of these areas can significantly reduce the size and overall cost of a given heat exchanger. Advances in manufacturing technology, such as the use of 3D printing or additive manufacturing (AM) have enabled the production of complex geometries that would have been otherwise inconceivable for manufacturing. Advances have the capability to significantly augment heat exchanger performance by allowing engineers to leverage complicated geometries that are impossible to generate accurately with traditional manufacturing. Complex geometries are crucial to the optimized design of heat exchangers as the heat capacity is directly proportional to the amount of heat transfer surface area. Thus, increasing the surface area to volume ratio will improve heat exchanger efficiency for a given volume.

Additive manufacturing technology is still relatively novel. Early work on 3D printers focused almost entirely on polymers and plastics. The first patent for a metal 3D printer was not awarded until 1995. Since this time, however, dozens of companies have begun to develop AM capabilities using a variety of different methods. Electron beam additive manufacturing (EBAM), direct metal laser sintering (DMLS), and powder bed fusion (PBF) are all metal printing technologies currently being used in industry and being further developed. These methods are capable of producing highly customizable, high fidelity parts that traditional manufacturing would be unable to produce. Large scale mass production utilizing these technologies is in its infancy; however, the benefits associated with AM technology including on demand parts and reduced waste show that these techniques are viable to pursue and examine. While AM allows for highly customizable parts, there are several obstacles to the implementation of AM on a large scale. AM produces high roughness parts which lead to an increase in pressure drop in a heat exchanger. Additionally, support structures needed for printing can impact the as-printed surface and make welding or machining more difficult.

A. HEAT EXCHANGER DESIGN

A heat exchanger is a device used to transfer energy between two fluids. The energy is transferred in the form of heat, which moves from a hot fluid to a cold fluid due to the temperature gradient. The graphic displayed in Figure 1 presents a basic overview of how a counter-flow heat exchanger works. The color gradient indicates the flow of energy from the hot fluid to the cold fluid.

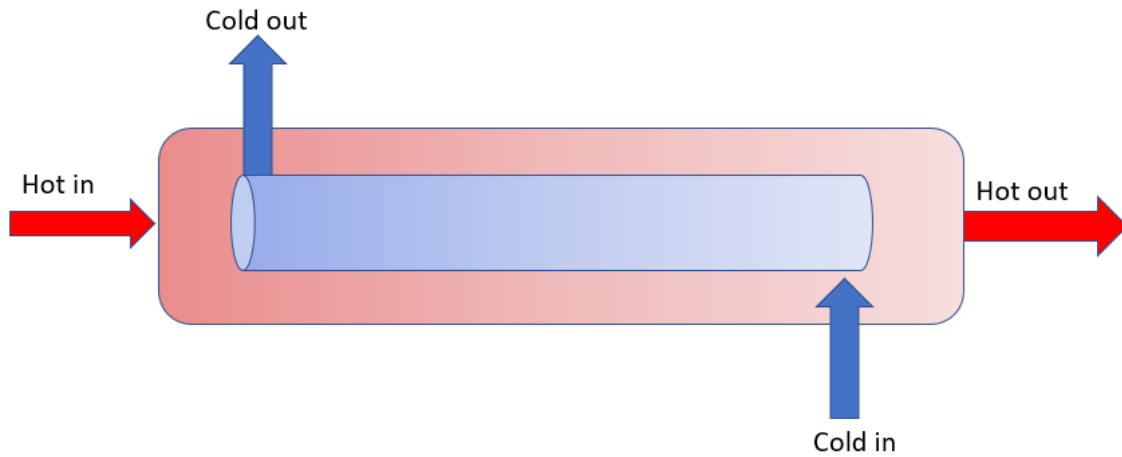


Figure 1. Basic heat exchanger illustration

A key improvement to the function of a heat exchanger is the increase in heat transfer surface area. Heat exchanger heat load can be calculated according to Equations (1) and (2)

$$Q = UA\Delta T_{LM} \quad (1)$$

with

$$\Delta T_{LM} = \frac{(T_{h,i} - T_{c,o}) - (T_{h,o} - T_{c,i})}{\ln\left(\frac{T_{h,i} - T_{c,o}}{T_{h,o} - T_{c,i}}\right)}, \quad (2)$$

where Q is the total heat load, A is the heat transfer surface area, U is the overall heat transfer coefficient and ΔT_{LM} is the log mean temperature difference, $T_{h,o}$ is the hot outlet temperature, $T_{h,i}$ is the hot inlet temperature, and $T_{c,o}$ and $T_{c,i}$ are the cold fluid outlet and inlet temperatures respectively [4]. Thus, for a given inlet and outlet temperature, the total heat load of a heat exchanger can be improved by increasing the heat transfer area or increasing the heat transfer efficiency of the heat exchanger.

1. Shell and Tube Heat Exchangers

Figure 2 depicts a common shell-and-tube heat exchanger (STHE) design [5] and Figure 3 presents a visual representation of the heat transfer surface area for a STHE. This type of heat exchanger is widely used in many large-scale industrial applications and is the type of heat exchanger that is examined in this study.

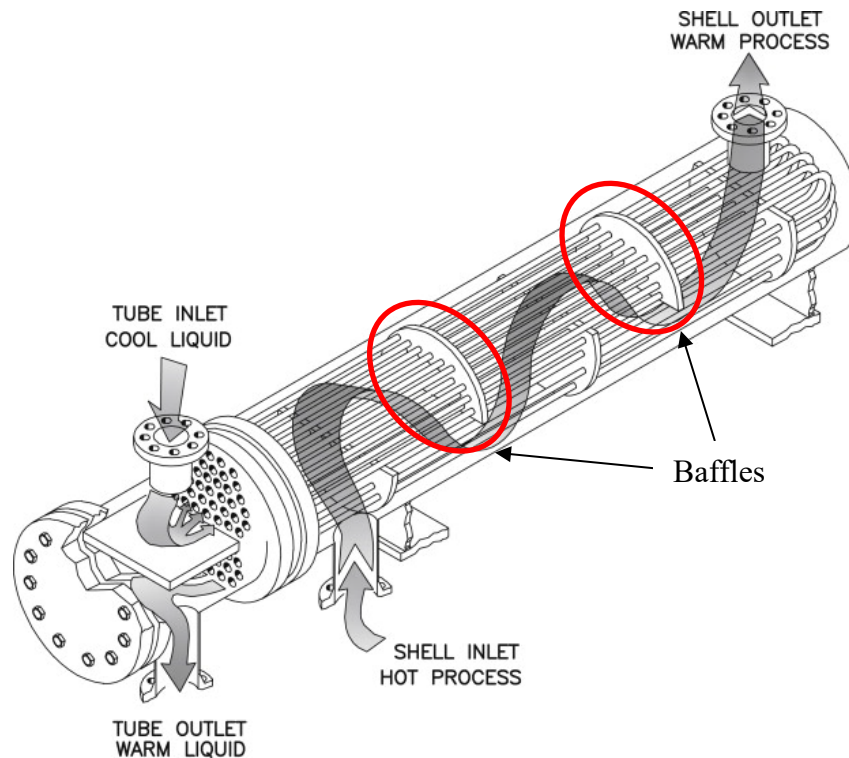


Figure 2. Shell-and-tube heat exchanger (STHE) illustration. Source: Adapted from [5].

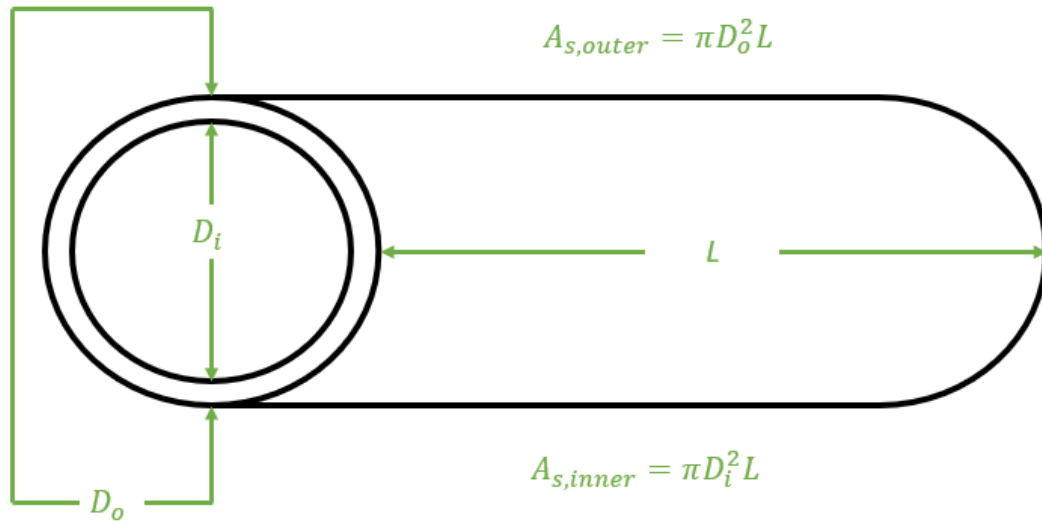


Figure 3. Tube heat transfer surface area illustration

Increases in the inner and outer tube surface area lead to overall increased heat transfer surface area (A in Equation (1)), leading to improved overall heat transfer. Shell and tube heat exchangers often utilize baffles to guide flow on the shell-side over the tube bank, as denoted by the red circles in Figure 1. The number of baffles passes that a heat exchange has generally increase the heat transfer but incur a larger pressure drop on the shell-side. In addition to customizing the number of baffles and amount of baffle passes, the heat exchanger designer also specifies the way in which the tube bank is arranged. Tube bank arrangements include the geometric configuration of the tube bank (square, triangular, diamond, etc.) as well as the spacing, or pitch, between tubes. Figure 4 illustrates several common tube bank arrangements. Once base heat transfer and pressure drop characteristics of specified shell and tube geometry are known, experimental correlations can be used to optimize the number of baffles, baffle passes, tube arrangement, and tube pitch. Options for heat exchanger design are governed by the thermal performance of the heat exchanger as well as the pressure drop. Pressure drop is considered as a higher pressure drop leads to an increase in overall pumping power required to operate the heat exchanger at given operating conditions. Thus, it is imperative to ensure improvements with regard to heat

exchanger thermal efficiency are weighed in regards to the pressure drop incurred to ensure the heat exchanger is capable of operating at the specified operating conditions.

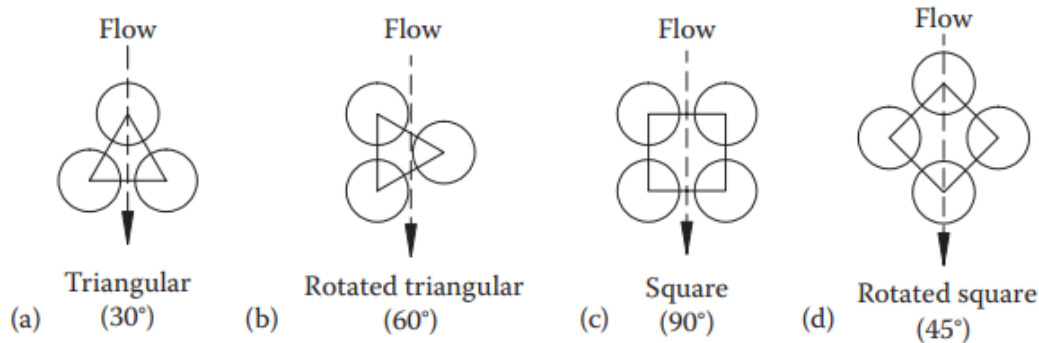


Figure 4. Common tube layout patterns. Source: [5].

2. Heat Transfer Augmentation

There are several augmentations that can be made to tubes that improve both the internal and external heat transfer coefficients thereby reducing the thermal resistance. One augmentation is the use of corrugated tubing. In Barba's 2002 paper [6], experimental methods were used to determine the effects of corrugated tubing. Figure 5 presents an image of the tube examined in the study as a visual reference for what is referred to as corrugated tubing. The results of the experiment were an order of magnitude increase in heat transfer coefficient and a pressure drop increase of between 1.83 and 2.34 times for low Reynolds number flows. Similarly, Wu et al. [7] examined four different corrugation schemes using both numerical and empirical methods. The results of the experimentation were improvements in the heat transfer coefficient for all four corrugated cases as compared to the smooth tubes—all of these corrugated tubes also incurred a higher pressure drop. The conclusions reached concur with Barba's findings. Corrugation serves to improve the heat transfer coefficient of both the internal and external walls of a tube as the corrugation affects both the external and internal areas.

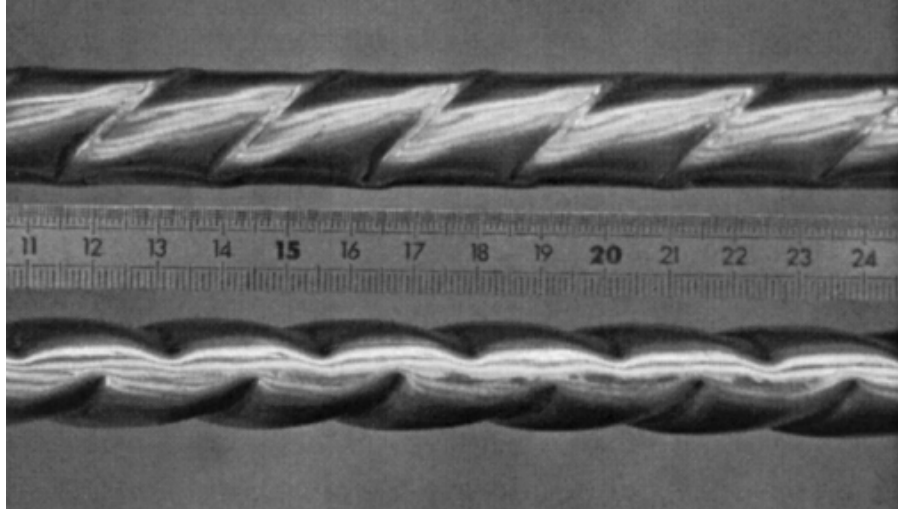


Figure 5. Front and side view of corrugated tube used in Barba's experiment (units in centimeters). Source: [6].

Another method of heat transfer augmentation in tubes is the use of inserts, which are also known as turbulators. Figure 6 depicts several types of twisted tapes used to promote turbulence in fluids [5]. Turbulators disrupt boundary layer formation on the walls of tubes and promote fluid mixing leading to improved heat transfer capability and increased pressure drop. Chiou [8] built upon a large body of work in an examination of heat transfer augmentation using turbulator coils. The experiment utilized cooling water within copper tubing that had several different types wire coil turbulators inserted to cool oil. The experiment showed significant heat transfer improvement in the augmented schemes compared to a bare tube. The effectiveness of the inserts was shown to be higher at lower Reynolds numbers, but still yielded improvements in highly turbulent flows.



Figure 6. Several examples of twisted tape inserts. Source: [5].

The union of flow turbulators and corrugated tubing yields spirally corrugated tubes. These tubes incorporate corrugations that are arranged in a helix around a tube. Vicente et al. [9] examined several iterations of spirally corrugated tubing in an experiment to examine the heat transfer and frictional performance of spirally corrugated tubes. The empirical results of the experiment showed the potential for an up to 300% increase in heat transfer coefficient in the spirally corrugated tubes as compared to bare tubes. Additionally, this experiment was performed at high Reynolds numbers in turbulent flow and presented a demonstrable increase in heat transfer capability. Qian et al. [10] determined that increasing the number of starts in a spirally corrugated tube led to higher heat transfer improvements as well as a lower pressure drop compared to other lower start spirally corrugated tubes. Zimparov [11] experimentally compared two different three-start spirally corrugated tubes with several variable pitch twisted tape inserts. The findings of this study concluded that for a constant pumping power, spirally corrugated tubes with twisted tape inserts have the potential to increase heat transfer by up to a factor of three.

B. COMPUTATIONAL FLUID DYNAMICS

The bulk of this study uses CFD to simulate fluid flow and heat transfer through complex geometries. Due to the complexity of the proposed geometries, there are not well characterized and validated experimental correlations available for the internal and external

flows that will be examined. CFD allows for the use of numerical simulations to examine the fluid flows in the proposed geometry.

For this study, all CFD analysis was completed using the implicit Navier-Stokes solver within ANSYS CFX. CFX numerically solves the Navier-Stokes equation in a finite volume across a user defined grid known as a mesh. The more well defined the mesh is, the more likely a solution is to converge. CFX breaks a finite volume into a computational domain where it is tracked through a timestep or iteration and the simulation is considered complete when the solution converges. The CFX solver determines solution converge through residuals, which are measures of how closely the simulation satisfies the laws of conservation of mass, momentum, and energy. The more complicated—or fine—a mesh becomes, the more computationally expensive the entire solution becomes. It is important to create an efficient mesh that can accurately model a physical solution while not running for an unnecessary amount of time.

1. Fluid Flow Characterization

The field of fluid dynamics seeks to characterize the complex nature of the movement of fluids. In this study, the primary means of fluid flow will be internal flows through the tube and external flows over the tube bank. In order to apply CFD to the geometries, the flows examined must be characterized as laminar or turbulent. Laminar flows are ordered flows where there is minimal fluid mixing and lacks the presence of eddies while turbulent flows have highly randomized mixing and the formation of many eddies. Additionally, laminar flows have a small, steady boundary layer while turbulent flows have large unsteady boundary layers that cause fluid mixing and even have the potential to detach from the wall.

Boundary layers in internal flows develop from the inlet of the fluid region to the outlet. Upon entering a channel, a boundary layer begins to form at the walls. These boundary layers grow until they combine at a given axial length down the channel. The region of boundary layer growth is known as the hydrodynamic entry region and the length at which the boundary layers meet is known as the entry length. Figure 7 illustrates this phenomenon [12].

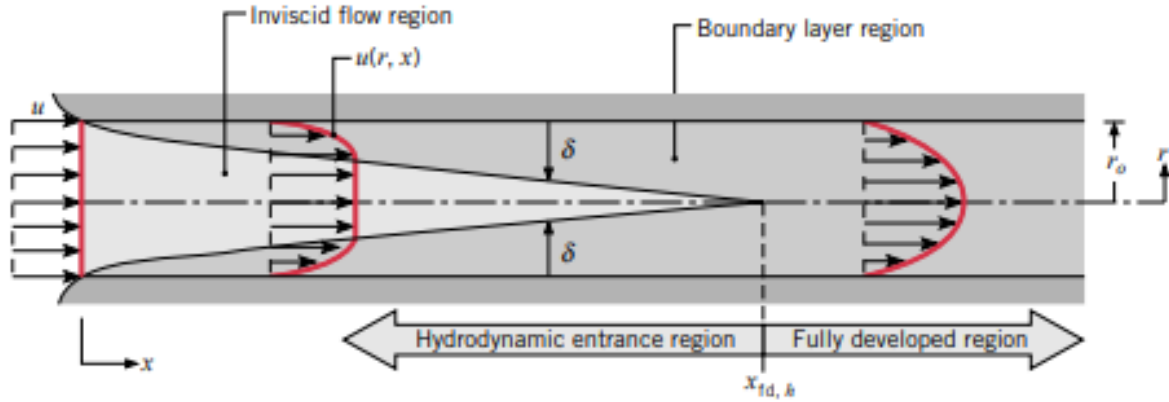


Figure 7. Flow development through a tube. Source: [12].

Fully developed flow is characterized by a uniform velocity profile, that is there are no changes in the fluid characterization in the direction of the flow. Heat transfer is improved in the entry region and pressure drop is greater as a result of increased shear stress. Many experimental correlations apply exclusively to fully developed flow, so it is important to understand the behavior of a fully developed fluid in order to apply correlations to check CFD computations.

Flows can be numerically characterized as laminar or turbulent according to their Reynolds number, which is a non-dimensional value that is defined as the ratio of inertial forces to viscous forces in a fluid flow. Reynolds number can be calculated according to Equation (3) [12]

$$\text{Re}_D = \frac{\rho u D}{\mu}, \quad (3)$$

where ρ is the density of the fluid, u is the velocity of the fluid, D is the characteristic dimension (the hydraulic diameter in this case), and μ is the viscosity. For pipe flows, Reynolds numbers below 2300 are generally found to be laminar while Reynolds numbers above 5700 are generally found to be turbulent as a rule of thumb [4]. A transition region between laminar and turbulent flow accounts for the difference between these two benchmarks. The characteristic length for the cross-sectional area of a flow can be put in terms of a diameter by a ratio of the area to the wetted perimeter. In the case of a circular

cross-section, the characteristic dimension becomes the diameter. The ratio to determine a characteristic dimension, known as a hydraulic diameter, is defined by Equation (4) [4]:

$$D_H = \frac{4A_{flow}}{P_{wetted}}. \quad (4)$$

This equation allows for virtually any geometry to be represented by a single length scale for calculation of Reynolds number from a free-flow area and wetted perimeter so long as the cross-sectional area is constant. In order to derive fluid flow parameters from specified inlet conditions, the mass flow rate relation is applied according to Equation (5)

$$\dot{m} = \rho u A, \quad (5)$$

where A is the cross-sectional area, and u is the velocity. From this equation, velocities and mass flow rates can be derived from volumetric flow rates in order to apply to solve for a Reynolds number. Determining the inlet parameters is key in setting up the correct boundary conditions to be used in CFD simulations to solve the discretized Reynolds-averaged Navier-Stokes equations.

2. Turbulence Models

All flows examined in this study were turbulent flows ($Re > 4000$), which require special treatment to properly capture the fluid behavior. Turbulent flows are numerically solved for using turbulence models. Two of the most important turbulence models to this research are the k - ϵ and k - ω models. The k - ϵ is a two-equation model which solves two transport equations for k (turbulence energy) and ϵ (energy dissipation [13]). The two equations are shown in Equation (6) and Equation (7)

$$\rho \frac{Dk}{Dt} = \frac{\delta}{\delta y} \left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon} \right) \frac{\delta k}{\delta y} \right] + \mu_T \left(\frac{\delta u}{\delta y} \right)^2 - \rho \varepsilon - 2\mu \left(\frac{\delta k}{\delta y} \right)^2 \quad (6)$$

and

$$\rho \frac{D\varepsilon}{Dt} = \frac{\delta}{\delta y} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\delta \varepsilon}{\delta y} \right] + c_1 f_1 \frac{\varepsilon}{k} \mu_T \left(\frac{\delta u}{\delta y} \right)^2 - c_2 f_2 \frac{\rho \varepsilon^2}{k} + 2\mu \mu_T \left(\frac{\partial^2 u}{\partial y^2} \right), \quad (7)$$

where c_1 and c_2 are proportionality constants, f_1 and f_2 are empirically determined functions, and σ is the Prandtl number. The k- ε model is an accurate model for high-acceleration flows in the freestream; however, the model struggles to accurately represent the behavior of fluid closer to the viscous sublayer. This is where the k- ω model proves effective by introducing a term known as specific dissipation. This model solves the same turbulence energy transport equation but introduces the specific dissipation (ω) as the second transport equation to be solved shown in Equation (8) and Equation (9):

$$\frac{D\rho k}{Dt} = \tau_{ij} \frac{\delta u_i}{\delta x_j} - \beta \rho \omega k + \frac{\delta}{\delta x_j} \left[\left(\mu + \sigma_{k1} \mu_t \right) \frac{\delta k}{\delta x_j} \right] \quad (8)$$

and

$$\frac{D\rho \omega}{Dt} = \frac{\gamma_1}{\nu_t} \tau_{ij} \frac{\delta u_i}{\delta x_j} - \beta_1 \rho \omega^2 + \frac{\delta}{\delta x_j} \left[\left(\mu + \sigma_{\omega 1} \mu_t \right) \frac{\partial \omega}{\partial x_j} \right]. \quad (9)$$

The k- ω model more accurately represents slightly detached flows and has improved accuracy closer to the viscous sublayer [14,15]. The union of these two turbulence models is the k- ω SST model which uses a blending function, F_1 , to apply the k- ε model in the free stream and the k- ω model at the wall. The k- ω SST is a useful turbulence model in situations where the flow separation is of interest, and due to the complex boundary layer in the geometries examined, this model will prove useful [16].

In this study, it is necessary to achieve a boundary layer resolved model. The degree to which a model is wall resolved is characterized by a parameter known as y^+ , which is defined in Equation (10) [17]

$$y^+ = \frac{yu_T}{\nu}, \quad (10)$$

where y is distance from the wall, u_T is normalized friction velocity, and ν is the kinematic viscosity. This value indicates how well resolved the flow behavior is at the wall (the lower, the y^+ , the more well defined the fluid flow is at the wall). Properly resolving this viscous sublayer is of particular interest in a heat exchanger heat transfer scenario where the majority of heat transfer will occur at the wall as this is where the two fluids are closest. The remaining heat transferred will move from the outer wall of the tube to the free stream inside the tube as energy in the form of heat flows from a hot body to a cold body. The target y^+ for a wall resolved model is between one and five [18], with an ideal value being approximately one. The y^+ influences the size of the mesh required to properly resolve the viscous sublayer and can therefore cause the simulation to become quite computationally expensive. The $k-\omega$ SST requires a y^+ of between one and five to be considered valid in capturing the viscous sublayer.

The use of numerical solver ANSYS Fluent was validated in an experiment conducted by Tang et al. [19] who compared several turbulence models to experimental results to validate the use of the $k-\omega$ SST model in the analysis of multi-start spirally corrugated tubes—yielding a tolerance between the experimental model and numerical simulations for Nusselt number and friction factor of 2.7% and 4.6% respectively. Bernadin et al. [20], [21] similarly designed and tested an additively manufactured twisted tube heat exchanger and compared experimental results to a numerical simulation using ANSYS CFX. The simulations were initially conducted using the $k-\epsilon$ turbulence model, but this simulation yielded poor results. Upon switching the analysis to the $k-\omega$ SST model, the solution conservatively reached within 10% of the empirical results. These experiments validated the assumption that the $k-\omega$ SST model will be the most applicable for the spirally corrugated tube scenario and this model will be used throughout the study.

C. ADDITIVE MANUFACTURING TECHNOLOGY

A relatively new novel way of increasing the heat transfer surface area of components is through additive manufacturing. AM provides several unique advantages over traditional means of manufacturing. AM allows for the design of far more complex geometries than traditional manufacturing is capable of—thereby allowing for the creation of parts with extremely high surface area to volume ratios. AM also has the capability to produce high fidelity parts. For example, General Electric’s redesigned fuel nozzle produced using AM resulted in a 25% reduction in weight and a 500% increase in strength [22]. The use of metal AM produces a more sustainable supply chain with up to ninety percent of metal powder being able to be reused in future builds. Metal AM frequently produces parts with significantly increased strength at the cost of ductility. The principle advantage of metal AM is the complexity of geometries that can be achieved while maintaining structural integrity of parts. This feat is not easily accomplished in traditional manufacturing, and is especially relevant to the design of heat exchangers where optimization of heat transfer surface area is paramount.

In addition to improving overall heat transfer surface area, AM also has the capability to improve thermal-hydraulic performance through novel geometries that promote turbulence and promote fluid mixing leading to slightly lower pressure drops and higher heat transfer coefficients. Arie et al. [23] developed an additively manufactured multi-furcating heat exchanger using Ti-64 that yielded a 15% to 50% increase in heat transfer coefficient compared to a legacy design and showed that with improvements in resolution a 75% to 90% increase can be expected. Zhang et al. [24] built a similar AM heat exchanger for high temperature applications that significantly improved the area to volume ratio compared to typical air-cooled heat exchangers and validated numerical results with experimental data. Gerstler and Erno [22] designed and tested an additively manufactured manifold heat exchanger with a UA improvement of 16% and reduction in mass and volume by 66% and 50% respectively compared to a legacy design. Utilizing AM allowed for an improvement in both heat transfer surface area and heat transfer coefficient with a reduction in both mass and volume. This body of work demonstrated the

ability of AM to construct geometries with body surface area improvements and overall heat transfer improvements.

1. Current State of AM

As of May 2023, AM technology is severely limited by size and production cost. One vendor contacted, a company called Velo3D, produces powder bed fusion systems and maintains a network of contractors with manufacturing capabilities. Currently, the largest powder bed fusion printer they have available is a 600 mm diameter by 550 mm z-height build plate. Due to powder management concerns, the most feasible print height currently is 400 mm [25]. Additionally, the z-height is what drives the cost of printing more than powder used as the increased height leads to increased stresses while printing. EOS is another manufacturer of powder bed fusion metal 3-D printing systems capable of printing with Ti-64. Their largest system is the EOS M400-4, which utilizes four high powered lasers to create parts on their 400 mm x 400 mm x 400 mm build plate [26]. Hundreds of tubes will likely be required; however, less material can be used should the heat transfer properties of the material warrant a smaller heat exchanger than the legacy design. Ultimately, the technology is still in its infancy and will be costly. Future expansion and competition in the AM field will increase the viability of this manufacturing method for larger scale industrial products.

2. Cost Analysis

A vendor, Duncan Machine Products, that is equipped with a PBF printing system was contacted for a quote to gauge printing costs. The part submitted was a 400 mm section of tube. The maximum amount of tubes that would fit on their build plate is 54 tubes. Printing 54 tubes at 400 mm in length using PBF was quoted for a total amount of \$70,865.28, or \$1,312.32 per tube with vacuum annealing included to relieve residual stresses. This amounts to \$3.28/mm per tube. This value will be used to calculate an estimated total material cost upon completion of sizing analysis. The vendor (Duncan Machine Products) stated that costs generally increase with part z-height, so it may be more feasible to print smaller heights and use circumferential welding to bind them together [27].

THIS PAGE INTENTIONALLY LEFT BLANK

III. METHODS

Several critical parameters were defined at the beginning of this study. These parameters are necessary to meet in order to successfully design a heat exchanger to function at operational requirements. The performance of the overall selected geometry and heat exchanger as a whole must fall within the benchmarks specified and all subsequent analysis was completed with the criteria prescribed in Table 1.

Table 1. Critical parameters

Parameter	Value in Metric Units
Power	5.861 MW
Size	3 m x 0.75 m x 0.75 m
Temperature	Seawater: -2°C to 32°C * Feedwater: 82°C
Maximum pressures	Seawater: 10 MPa Feedwater: 1 MPa
Flow rates	Seawater: 0.38 m^3/s * Feedwater: 0.23 m^3/s *, 0.19 m^3/s , 0.13 m^3/s
Pressure drop	Seawater: 10 kPa at 0.145 m^3/s Feedwater: 6.9 kPa at 0.063 m^3/s
Cost	Less than \$2.25 million

*Bolded values indicate limiting parameters

In order to analyze the heat exchanger's performance, limiting parameters needed to be derived from the critical parameters defined in Table 1. These values would be the extreme ends of heat exchanger operation to ensure the heat exchanger functions at the limit. Should the heat exchanger function properly at the limiting condition, it can be assumed that under more ideal operating conditions it will perform within design limits. The limiting temperature selected was 32°C for the seawater and 82°C for the feedwater.

The 32°C temperature was chosen based on the hottest month of sea surface temperature data collected by NASA, 32°C is the mean of all temperatures above the temperature of 31°C [28]. This seawater temperature was chosen as it leads to the lowest temperature difference between the two fluids. In a heat exchanger, the larger the temperature difference is between fluids, the better the performance of the heat exchanger. Assuming the minimal specified temperature difference ensures the heat exchanger will function at its limit and perform better at increased temperature differences. The flow rates selected were 0.38 m³/s for seawater and 0.23 m³/s for the feedwater. The highest given volumetric flowrate was selected as the limiting parameter because pressure drop incurred will be highest. The pressure drop incurred at the faster moving flow rates will be greater, meaning if the tubes meet the criteria at the elevated flowrates, they will be well below the maximum expected pressure drop at design conditions.

The task to design a heat exchanger utilizing additively manufactured components continued with an examination of geometry. Examining the literature surrounding corrugations showed promise—several advanced heat exchangers utilize corrugation schemes to improve heat transfer efficiency and reduce fouling of tubes. Further study revealed that imparting swirl flow to the fluid and enhancing both fluid mixing and turbulence could serve to improve heat transfer while also offsetting the pressure drops typically incurred with corrugation schemes. Considering pressure drop is crucial in this study as printed metals have order of magnitude higher roughness values as compared to wrought materials, which greatly impacts the pressure drop due to increased wall friction.

A. GEOMETRIC PARAMETERIZATION

To successfully perform baseline calculations, it was necessary to parameterize a spirally corrugated tube based on several input parameters. These simple parameters could then be used to calculate a perimeter and flow area in order to determine a hydraulic diameter according to Equation (4). A MATLAB code was implemented to construct any spirally corrugated tube by using a tube thickness, start thickness, tube inner diameter, start thickness, and number of starts. Equations used to generate the circle-to-circle intersection plots were obtained from Wolfram MathWorld [29]. Owing to the nature of spiral tubes,

the cross section and perimeter are uniform throughout despite the twist in the longitudinal direction—meaning a hydraulic diameter can be applied to represent the geometry by a characteristic dimension. The MATLAB code generated intersecting circles and used geometric relationships between intersecting circles to calculate cross-sectional area, flow perimeter, and outside tube diameter based on the specified input parameters. The MATLAB code developed can be found in Appendix A. Figure 8 and Figure 9 show a cross section generated by the MATLAB code and subsequently implemented into CAD software.

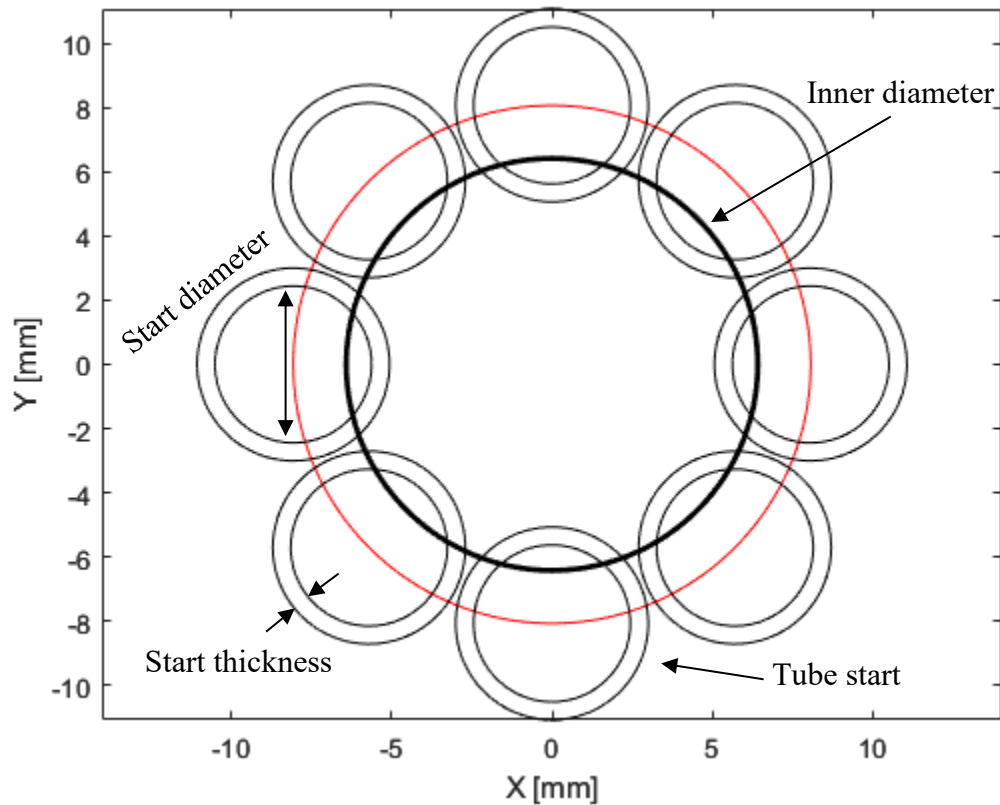


Figure 8. MATLAB generated cross-section based on inner diameter, start diameter, number of starts, start thickness, and tube thickness

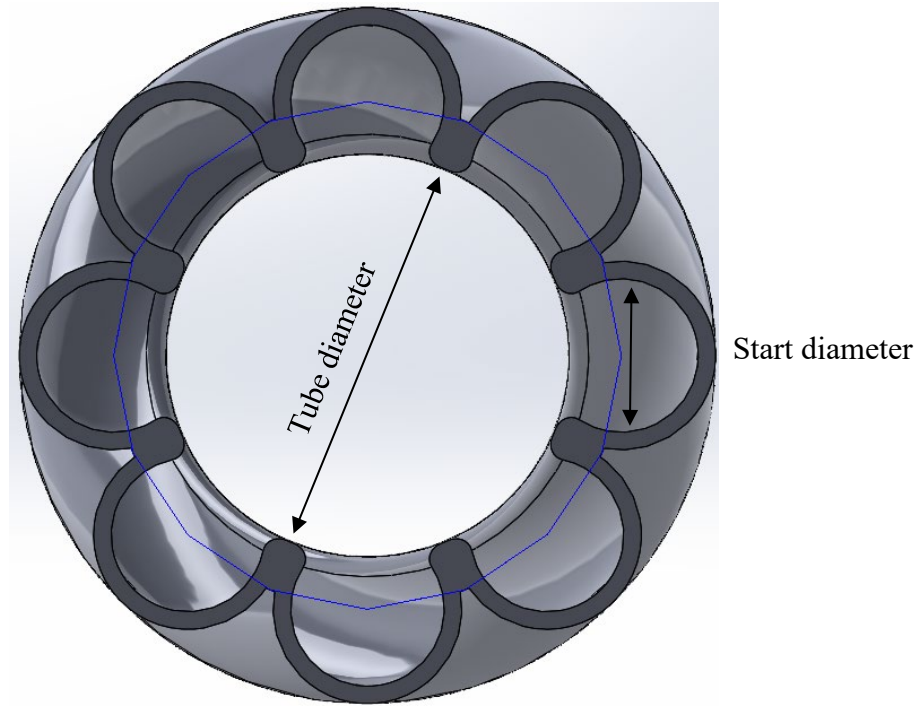


Figure 9. MATLAB parameterized geometry implemented in SolidWorks CAD software

B. PRELIMINARY PRESSURE DROP CALCULATIONS

Initial calculations of pressure drop were completed utilizing several experimental correlations as well as the geometric information. A MATLAB function (seen in Appendix A) was created in the same geometric parameterization file. This function took flow condition and geometric inputs, ran the values through experimental correlations, and produced a final pressure drop value based on the specified conditions. The flow conditions input were the fluid density (ρ), volumetric flowrate (V), maximum heat exchanger area (A_{HX}), kinematic viscosity (ν), heat exchanger length (l), and relative roughness. First, a mass flow rate and inlet velocity were derived from the specified volumetric flow rate and area according to Equation (5). The geometric data was then used to determine a hydraulic diameter which allowed for the computation of a Reynolds number. These values were then used to calculate a friction factor, f , using the Swamee-Jain correlation shown in Equation (11) [12]:

$$f_{sj} = \frac{0.25}{\left[\log \left(\frac{\varepsilon}{3.5d} + \frac{5.74}{\text{Re}_D^{0.9}} \right) \right]^2}. \quad (11)$$

Swamee-Jain is used to calculate friction factors in turbulent flows. Prior to using this correlation, the Reynolds number was examined to ensure it was greater than 4000 so turbulent correlations apply. Should the flow not be turbulent, the MATLAB code would issue a warning. After the friction factor was calculated, it was possible to calculate pressure drop. Equation (12) presents the Swamee-Jain correlation used to predict pressure drop:

$$\Delta P = f_{sj} \frac{l}{D_H} \frac{\rho u_m^2}{2}. \quad (12)$$

C. SELECTED TUBE GEOMETRY

The tube geometry selected for the study was an eight-start tube. This design was chosen as it was based upon the minimal inner diameter for mechanical cleaning (3/4 inch) [5]. The eight-start design was able to fit the maximum number of starts without starts intersecting. The literature review indicated that increased in depth of corrugations improved the heat transfer coefficient [27]. Additionally, literature pointed to mitigated pressure drop through increasing the number of starts used [10]. These two criteria, as well as hand calculations indicating that pressure drops decreases as the number of starts increase influenced the selection of the geometry examined. A graphical representation of the hand calculations is shown in Figure 10.

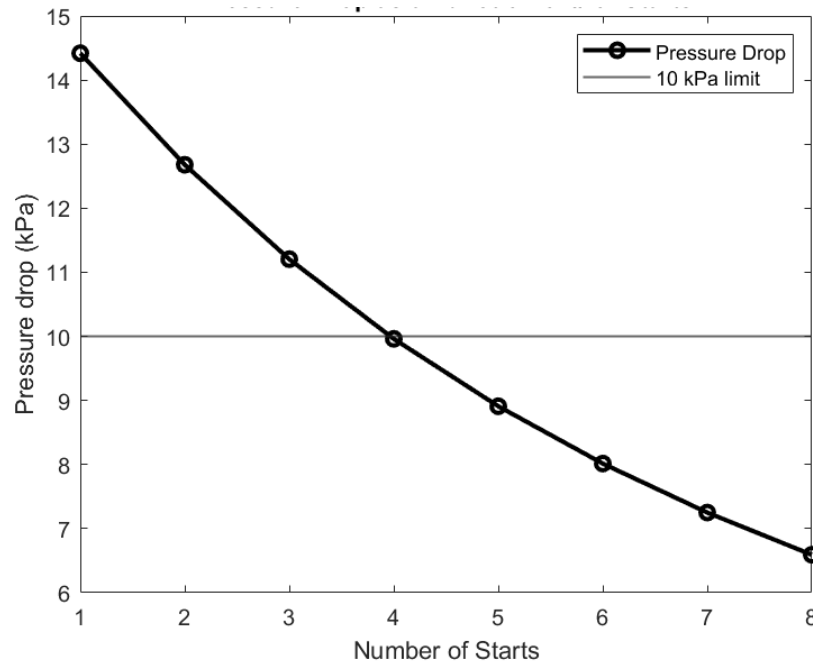


Figure 10. Results of calculated pressure drops of spirally corrugated tubes with different numbers of starts

The analytical results concur with experimental results analyzing pressure drop in multi-start spirally corrugated tubes [10]. The eight-start tube was examined for all subsequent analysis in this study. The tube has a 16.13 mm inner diameter with a tube thickness of 1.651 mm while the starts are six mm in diameter and 0.559 mm thick. The total outside diameter of a single tube was found to be 21.13 mm. From this value, an estimate could be determined for the maximum number of tubes that could fit into the heat exchanger by dividing the total permissible area by the square of the outer diameter of a single tube. With an estimated number of tubes, a mass flow rate for each tube was specified which allowed for the analysis of the performance of a single tube that could be used to derive parameters to model the performance of an entire heat exchanger.

D. COMPUTATIONAL FLUID DYNAMICS SET UP

All CFD conducted in this study utilized ANSYS CFX. To use this software, a CAD model was developed using SolidWorks and then imported into ANSYS workbench as a parasolid file. To conduct a fluid flow analysis in CFX, the solver requires that the fluid volume be input. The created CAD file contained the geometry of a tube and not the

fluid volume, so an additional step was required within SolidWorks. This consisted of capping the ends of the tube and using the intersect feature to generate two bodies in the CAD file: the tube wall and the internal fluid volume. Figure 11 illustrates the extracted fluid volume that needed to be input to CFX.

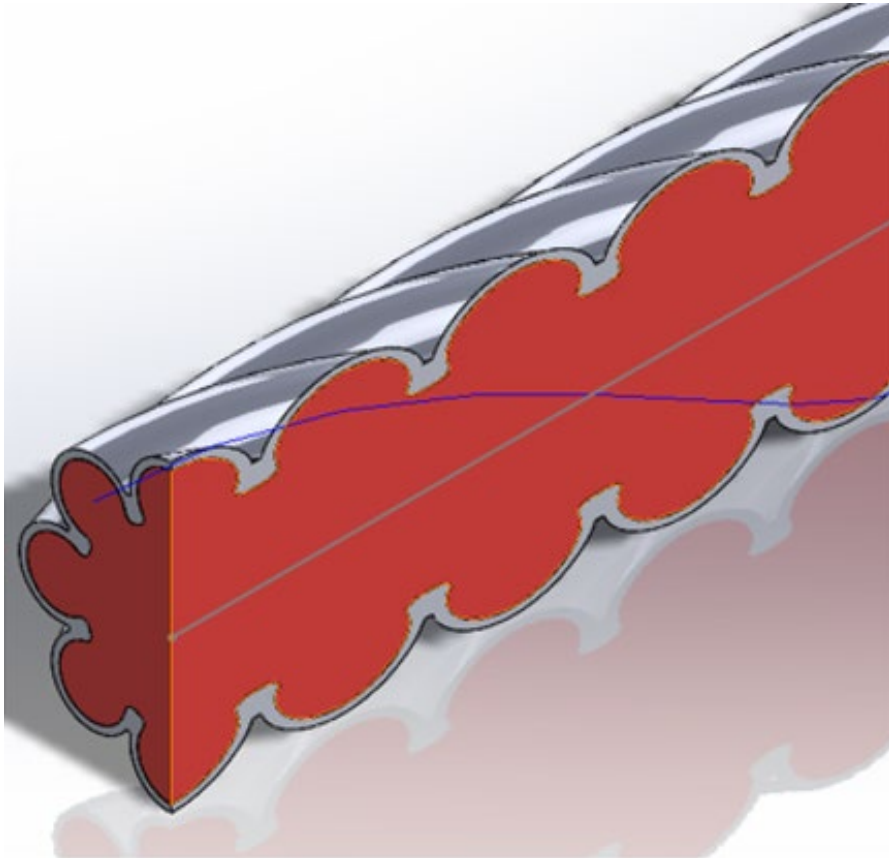


Figure 11. Isometric section plane view of fluid volume marked in red

1. Meshing

In this CFD analysis a mesh was generated on the part to be analyzed. ANSYS CFX has a built in mesher where the user can specify parameters to tailor the mesh to a specific geometry. Once the CAD file was imported to the mesher, faces were labeled inlet, outlet, and wall. The default element size was selected at 2.5×10^{-2} m, which was the element size ANSYS CFX defaulted to for the particular geometry. Tetrahedral elements were selected with curve mapping feature enables to ensure the mesh would capture the curvature of the

starts [29]. Inflation layers were specified utilizing a first-layer thickness. Growth rate was set to the recommended value of 1.2 [29]. The number of layers and first layer thickness were determined through an iterative process of choosing a target y^+ and back solving for a y -height value according to Equation (8). The function of the inflation layers is to capture the boundary layer behavior of the fluid by adding additional mesh refinement around the walls.

To speed up the meshing process, parts to be meshed were sliced into multiple bodies utilizing the SolidWorks CAD software. This was done because ANSYS CFX meshing allows the user to mesh multiple imported bodies in parallel while importing an entire geometry as a single body relegates the user to serial meshing. The effect of meshing on multiple CPUs is that the time to generate the mesh decreases by a factor of the number of CPUs used. Interfaces were used at the slices that mapped the end of one face to the beginning of the next one with a fluid-fluid interface.

a. Mesh Sensitivity

In this CFD study, a mesh sensitivity study is needed. The size of a mesh has serious implications for the validity of a solution—a simulation ran at one size mesh may yield completely different results from another simulation with the same conditions but a different sized mesh. A mesh sensitivity study is used to ensure that any conclusions reached from a CFD simulation are mesh independent, meaning that the selected mesh is of appropriate size to ensure consistent results. Another benefit of a mesh sensitivity study is that it can save computational power if running multiple times. A mesh sensitivity study is necessary in this case to ensure conclusions reached through a CFD simulation are not a function of the refinement of the mesh. The quality assurance plan for this study additionally required the use of a mesh sensitivity study.

2. Boundary Conditions

To solve the transport equations, Equations (6)–(9), that a CFD simulation uses to resolve fluid flows, properly specified boundary conditions are necessary. This study utilized several different boundary conditions in various simulation. The most basic boundary conditions used were a mass flow rate inlet, zero pressure outlet, and no slip wall

specification as recommended by the literature and the ANSYS CFX set-up guide [30]. Mass flow rates were calculated utilizing the parameterized MATLAB script (Appendix A) for the given geometry. Zero pressure outlet was selected as it drives the flow in the direction of the outlet and allows for calculation of a pressure drop by solely measuring the solution converged inlet pressure. No slip walls were selected as initial calculations made the smooth tube assumption. Wall roughness functions were input in later simulations to compare the effect of the smooth tube assumption compared to the actual as-printed rough surface that is expected when using direct metal laser sintering with Ti-64.

Inlet and outlet boundary were also specified as a recirculating boundary condition. This inlet was given a translational periodicity designation and a mass flow rate. All other surfaces were designated no slip walls as before. The effect of this boundary condition was to map the mass flow rate of the inlet onto the outlet, simulating a recirculation and as a result modeling fully-developed flow. Recirculating boundary simulations were more computationally expensive as the solver needed to iterate pressure and velocity values required to maintain a uniform velocity profile through the tube. This method did not allow for the use of heat transfer models, so the velocity profiles achieved with this method needed to be exported and stored for use in future simulations. Exported velocity profiles could be imported into future simulations as inlet boundary conditions to capture the effect of fully developed flow while also using heat transfer models.

Once flow conditions were specified, fluid models needed to be applied to the simulations. Simulations were designated as steady-state as no transient behavior was observed. Pressure drop simulations were run using an isothermal $k-\omega$ SST model with water properties at 32°C. The $k-\omega$ SST model was chosen due to its performance resolving the boundary layer and free stream by combining the $k-\epsilon$ and $k-\omega$ models. The literature reviewed pointed to the validity of this model in spiral tube geometries.

For heat transfer scenarios, a thermal energy model was selected as opposed to an isothermal model to allow for the transfer of energy between domain. The thermal energy model allowed for the specification of inlet temperatures and facilitated the conjugate heat transfer simulation. The turbulence model chosen was still $k-\omega$ SST. Additionally, beta features were enabled and constant domain physics was disabled to allow for the use of

separate fluid domains with different material properties (freshwater and seawater). With the thermal energy model, temperatures were specified at the domain inlets in addition to either mass flow rates or imported velocity profiles. All surfaces that were not used as an inlet or outlet were modeled as no slip walls. The fluid-fluid interface between the two fluids was modeled as a heat conducting thin material, Ti-6Al-4V in this study, with a wall thickness equal to the specified tube thickness.

After defining boundary conditions, the user must enter the solver manager where resources are allocated for computation of the simulation. The user can specify large problem or double precision models that are more computationally expensive, but allow for higher fidelity results. The solver manager allows for serial and parallel processing. For this study, parallel processing was exclusively used as it is the most efficient means of running large simulations by distributing computing resources across multiple CPUs. The CFX Command Line (CCL) files used in CFD simulations can be found in Appendix B, this file lists all the input parameters that the CFD code uses when running a given simulation in the form of commands that the computer uses.

a. Material Properties

Material properties for water and seawater used for this study were derived from Massachusetts Institute of Technology's water properties library and implemented using MATLAB [33], [34]. Titanium grades indicate an increase in strength as the grade number increases. Grade five titanium was examined in this study for its high strength and corrosion resistance. Material properties from wrought, grade five titanium [35] and PBF printed Ti-6Al-4V [36]–[38], are presented in Table 2.

Table 2. Material properties of titanium alloy Ti-6Al-4V

	Wrought grade 5 titanium	PBF printed Ti-6Al-4V
Density [g/cm ³]	4.43	4.41
Modulus of Elasticity [GPa]	113.8	110 ± 10
Ultimate tensile strength [MPa]	950	1230 ± 50
Yield Strength [MPa]	880	1060 ± 50
Thermal Conductivity [W/m-K]	6.7	5.5

Adapted from [35]–[38].

3. High-Performance Computing (HPC)

Naval Postgraduate School has access to a high-performance computing (HPC) cluster known as Hamming. Hamming is a Linux machine that has CFX installed and can run CFD simulations utilizing a great deal of computing power, allowing for larger simulations to be run at faster speeds than running on a standard computer. Interfacing with the HPC cluster requires additional action than simply running on a normal machine. To interface with Hamming, the user must login remotely to the submit node using an SSH client (MobaXterm for example). Then, a bash shell script file, or .sh file, must be used to send executable commands to the Linux cluster. A list of Linux commands uses as well as an example of a .sh file can be found in Appendix C. In these files, the user requests computing resources and allocates space. The final line in the bash shell script file requests a run of CFX calling a definition file that is created by the user in ANSYS workbench. The simulation can be monitored using the -tail command, which can read the most recent written lines in the result file, letting the user see live progress of the simulation. Upon

completion of the simulation, an output file will be created in the user's directory on Hamming, which can then be transferred to another computer with a more user-friendly graphical user interface (GUI) for postprocessing using file sharing software such as WinSCP.

E. HEAT TRANSFER QUANTIFICATION

Estimates for the geometry's heat transfer coefficients and outlet temperatures were extracted. The heat transfer coefficients were generated by determining a Reynolds number for the flow in accordance with Equation (3). Then, the Gnielinski correlation was used to generate a Nusselt number and friction factor for the given situations, as defined by Equation (13) and Equation (14) respectively [4]:

$$Nu_D = \frac{(f/2)(Re_D - 1000)Pr}{1 + 12.7(f/2)^{0.5}(Pr^{2/3} - 1)} \quad (13)$$

with

$$f = (1.58 \ln(Re_D) - 3.28)^{-2}. \quad (14)$$

Once a Nusselt number was generated for the flow, the hydraulic diameter was applied along with the fluid thermal conductivity following Equation (15):

$$h = \frac{Nu_D k}{D_H}. \quad (15)$$

This h value was used to represent the heat transfer coefficient for the geometry. To evaluate the actual heat transfer performance of the design, the log-mean temperature difference method was used. Outlet temperatures were extracted from the CFD simulations and used to calculate a log-mean temperature difference. Additionally, values for heat load, q , were determined from the CFD simulations using the outlet temperatures and flow parameters. With these values, it was possible to generate an overall heat transfer efficiency independent of area, UA , utilizing Equation (16) [4]:

$$UA = \frac{q}{\Delta T_{LM}}. \quad (16)$$

UA values were calculated based on the boundary temperatures and total heat transferred, q , in the CFD studies conducted. The heat transfer coefficient independent of area allows for the quantification and comparison of heat transfer performance of each design. This figure does not account for the increased area of the spirally corrugated tubes. The additional 30% increase in area will be factored in separately from the comparison of the overall heat transfer efficiency independent of area.

THIS PAGE INTENTIONALLY LEFT BLANK

IV. ANALYSIS

An initial stress analysis was conducted on the chosen design to ensure it can withstand the maximum loads it could be subjected to as prescribed in Table 1. Finite element analysis (FEA) was conducted in ANSYS Mechanical ADPL. Von Mises stress criteria was chosen as this model more accurately represents yield in brittle materials, which the Ti-6Al-4V will function as post heat treatment [35]. Figure 12 depicts the results of the FEA stress analysis conducted on the part.

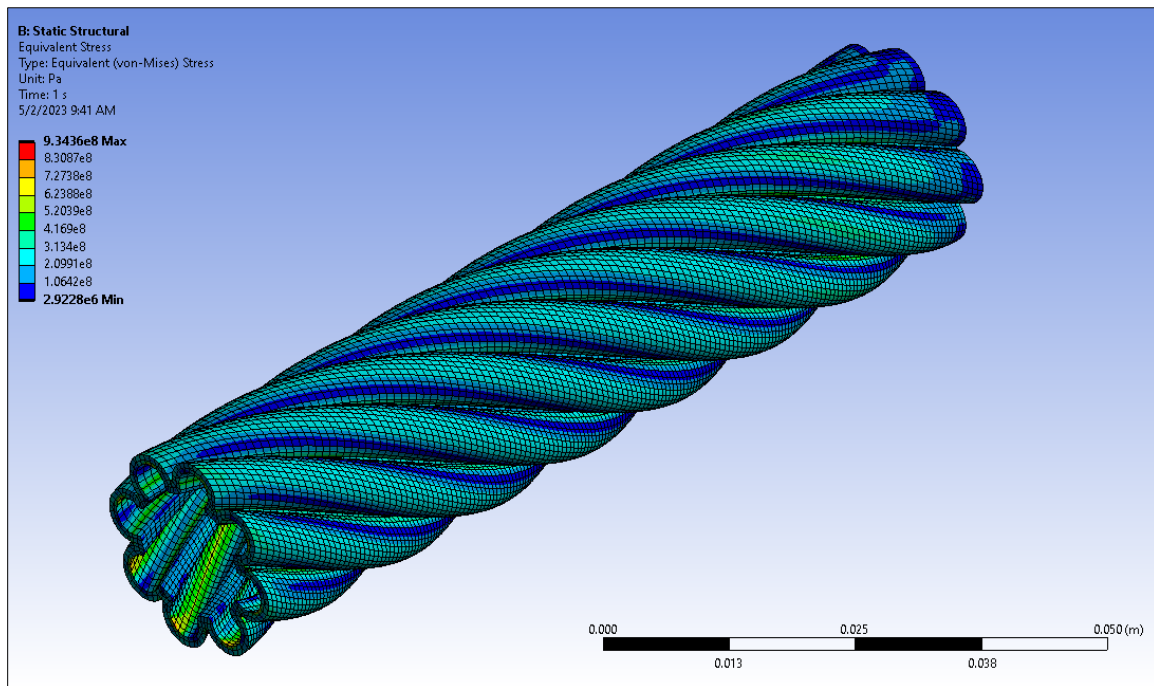


Figure 12. Results of finite element analysis on eight-start tube

The tube was subjected to a ten MPa pressure stress with a frictionless support and displacement condition. Stresses seen are an order of magnitude below the ultimate strength at one mm tube thickness. The main stress concentrators are at the edges of the starts, but this is where the tubes would be supported by a tube sheet, strengthening the parts anticipated to yield. To increase the factor of safety, thicker tubes would be advisable.

For the purposes of this study, it is assumed the tube sheet will prevent yield and that the tubes will rarely see pressures of ten MPa.

A. TEST CASE – BARE TUBE

A control simulation was run with a bare tube using the same boundary conditions as the analysis for the spirally-corrugated tubes. The bare tube had an outer diameter of 22.13 mm, which is the same outer diameter as the spirally corrugated tube. The bare test case is easily validated with correlations that apply to fully developed pipe flow in a triangular flow arrangement. The results of the simulation are represented visually in Figures 13–15.

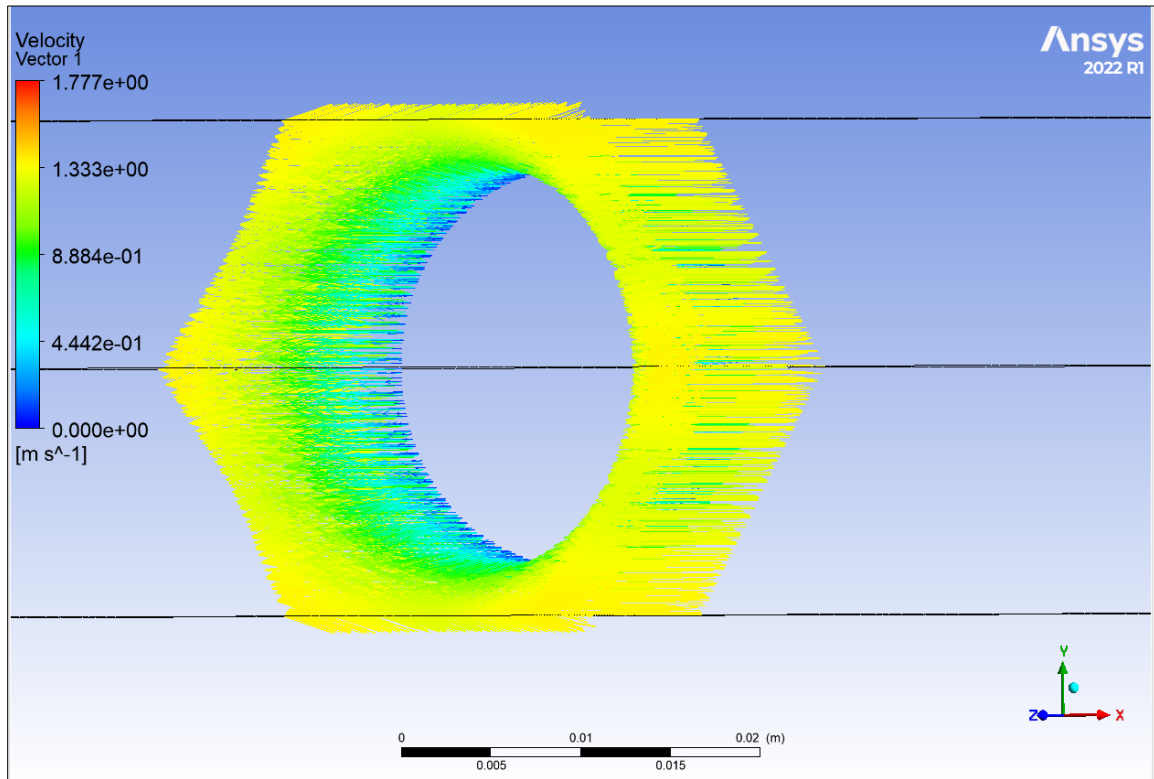


Figure 13. XY-plane velocity vector plot of shell side flow in bare case

The shell side flow in the bare tube behaves as expected. The shell-side flow moves along the tube with decreased velocity in the region attached to the tube due to increased shear stresses.

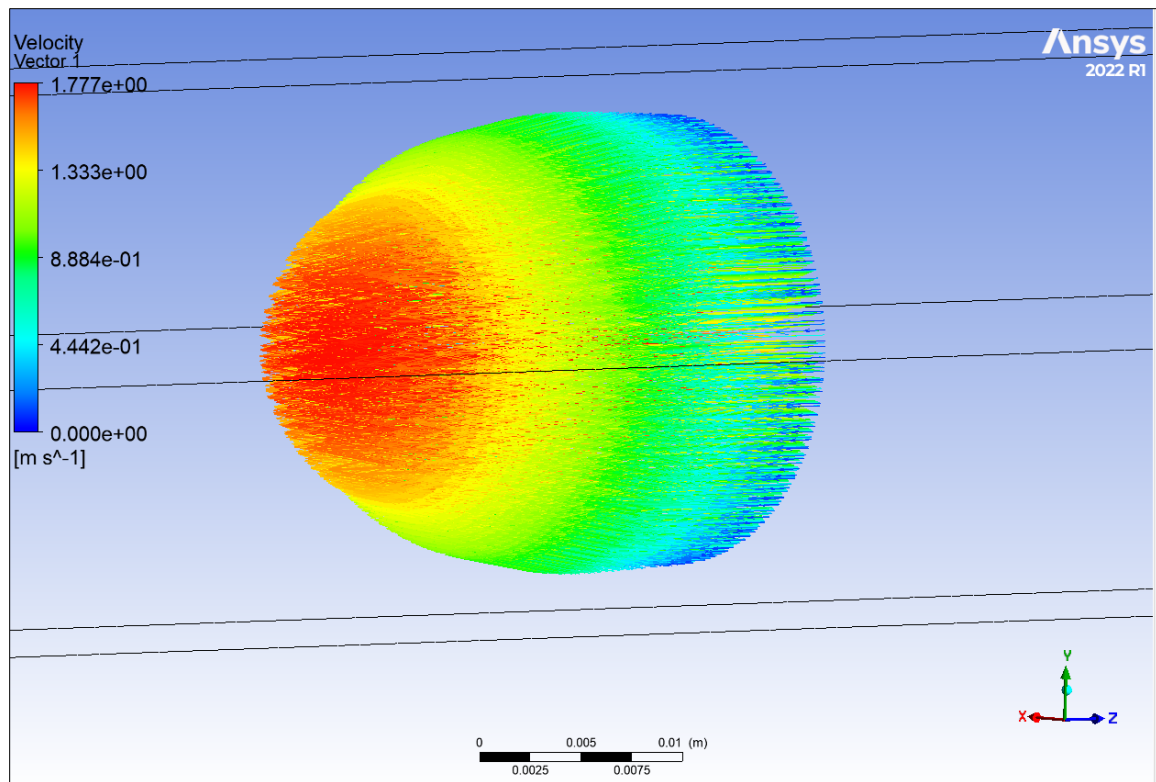


Figure 14. XY-plane velocity vector plot of tube side flow in bare case

As seen in Figure 14, the fluid behaves as predicted in the tube following a fully developed velocity profile in agreement with Figure 6. The shell side flow through a tube bundle was also examined to characterize the pressure drop. Figure 15 displays the results of the bundle flow analysis.

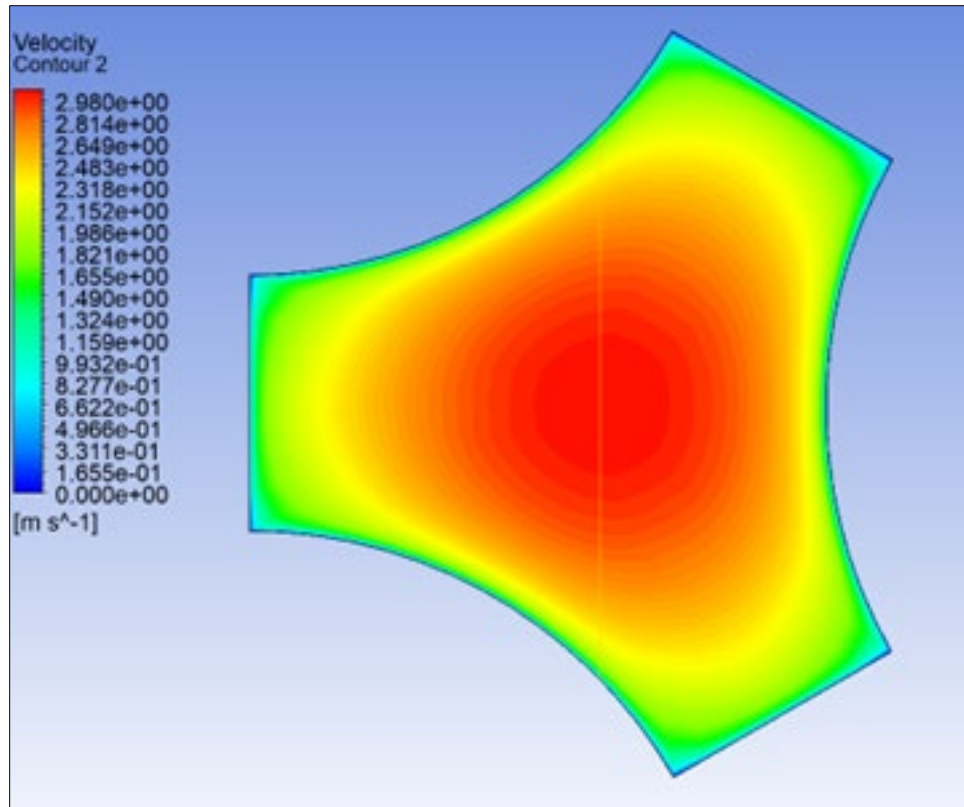


Figure 15. XY-plane velocity contour for simulated flow over a triangular pitch bundle of smooth tubes

The geometry seen in Figure 15 presents a unit cell of what flow would see passing over a tube bundle in coaxial flow. A triangular pitch was selected to characterize the flow over a bundle of smooth tubes as the same metric will be applied to the spirally corrugated tube case. The total pressure drop across this tube bundle was calculated as 3290.38 Pa. This flow has minimal swirl and a central jet forms at the annex of the three tubes. The temperature gradient of the bare tube scenario is displayed through the temperature contour in Figure 16.

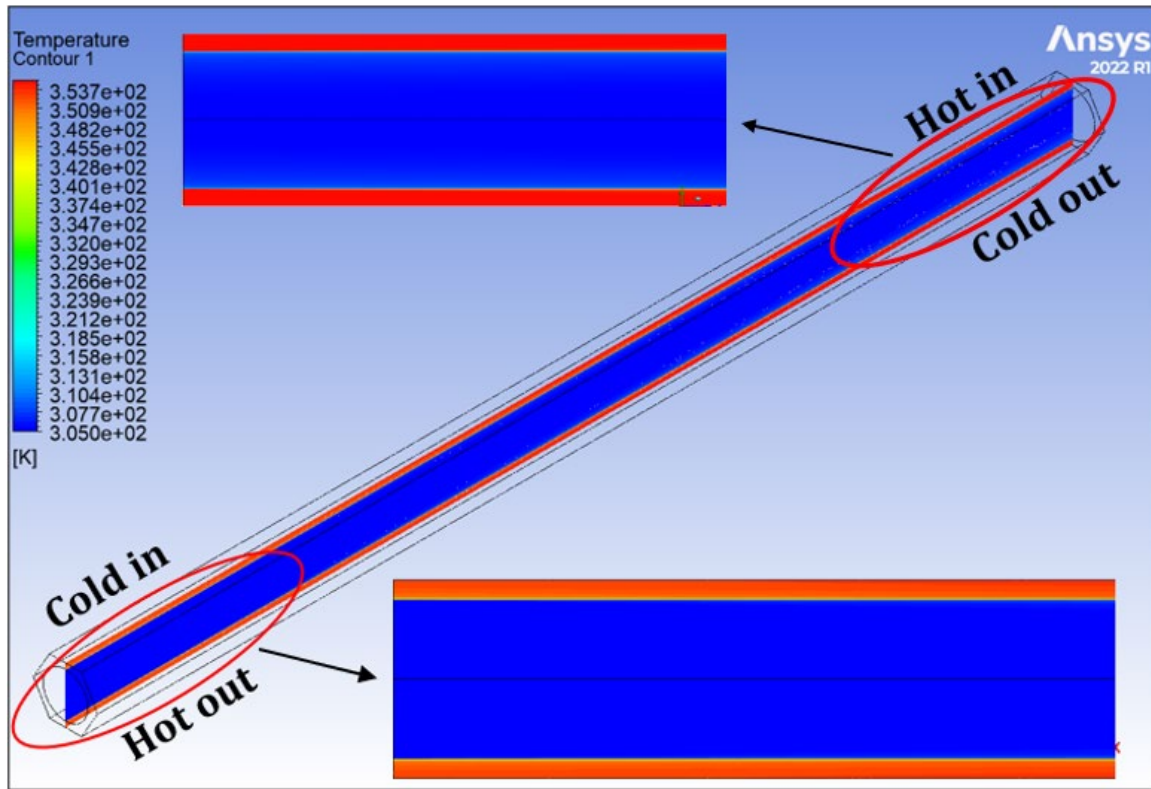


Figure 16. YZ-plane temperature contour for bare case

In the bare heat transfer scenario, it is observed that the majority of heat transfer occurs as the wall, which is to be expected. The bulk temperature of the tube side fluid did not change as much as the shell-side fluid because it is the higher velocity fluid. The final reported outlet temperatures for this case were 352.573 K and 306.508 K for the hot and cold outlets respectively. This is based on a hot inlet temperature of 355 K and a cold inlet temperature of 305 K. The inside and outside heat transfer coefficients were calculated as $6320 \frac{W}{m^2-K}$ and $3436 \frac{W}{m^2-K}$ respectively according to Equations (13)–(15).

B. EIGHT-START SPIRALLY CORRUGATED TUBE

Preliminary analysis was completed on a 500 mm section of tube cut into 100 mm slices. This analysis examined the pressure drop and fluid flow characteristics in the spirally corrugated geometry based on the limiting parameters. Results of the mesh sensitivity study are presented in Table 3.

Table 3. Results of mesh sensitivity study

Boundary Condition	Parameters	Mesh 1 (19 million elements)	Mesh 2 (37 million elements)	Mesh 3 (53 million elements)	Mesh 4 (70 million elements)
Mass flow inlet, zero pressure outlet	Pressure drop	2046.56 Pa	2041.86 Pa	2062.99 Pa	2065.99 Pa
	y^+	4.953	2.821	2.280	2.451
Imported Velocity Profile	Pressure drop	1318.44 Pa	1283.74 Pa	1283.52 Pa	1283.41 Pa
	y^+	3.683	1.783	1.393	0.831

Table 3 presents the results of the mesh sensitivity study for the initial simulations with no heat transfer. Calculations performed using the Swame-Jain correlation yielded a pressure drop of 1141 Pa, meaning the numerical simulation was 12.5% conservative compared to the correlation, which concurs with literature that reported a 10% conservative difference [21]. The 37 million element mesh size was selected due to the agreement with the literature and the hand calculations completed. Figure 17 shows velocity profile development through the tube length, with flow travelling from bottom left to upper right.

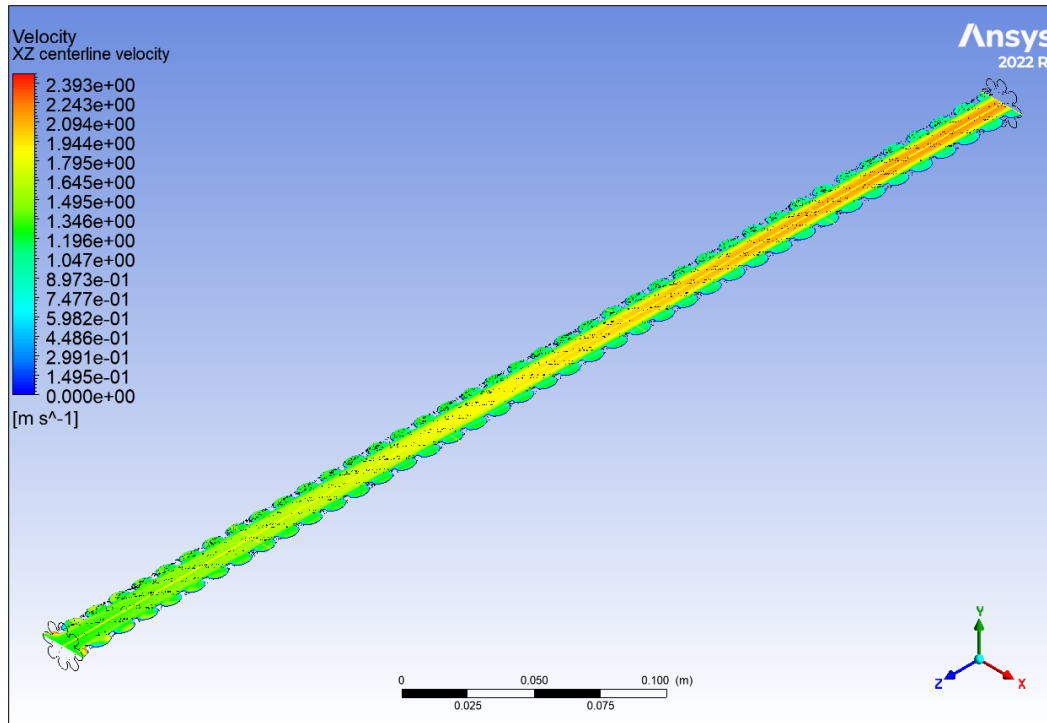


Figure 17. Centerline velocity through 500 mm section of spirally corrugated tubing, flow traveling from left to right

The increase in velocity towards the outlet is indicative of developing fluid flow. Pressure drop in this case was higher due to entry effects. The flow behavior was further examined by taking samplings every 100 mm along the x-direction to generate a plot of the developing velocity profile. This was done to determine whether or not the flow was fully developed at the outlet. A fully developed velocity profile could be exported to another simulation where a fully developed pressure drop could be extracted and compared to calculations done using experimental correlations to validate the model. Figure 18 shows a plot of the centerline velocity through the tube length.

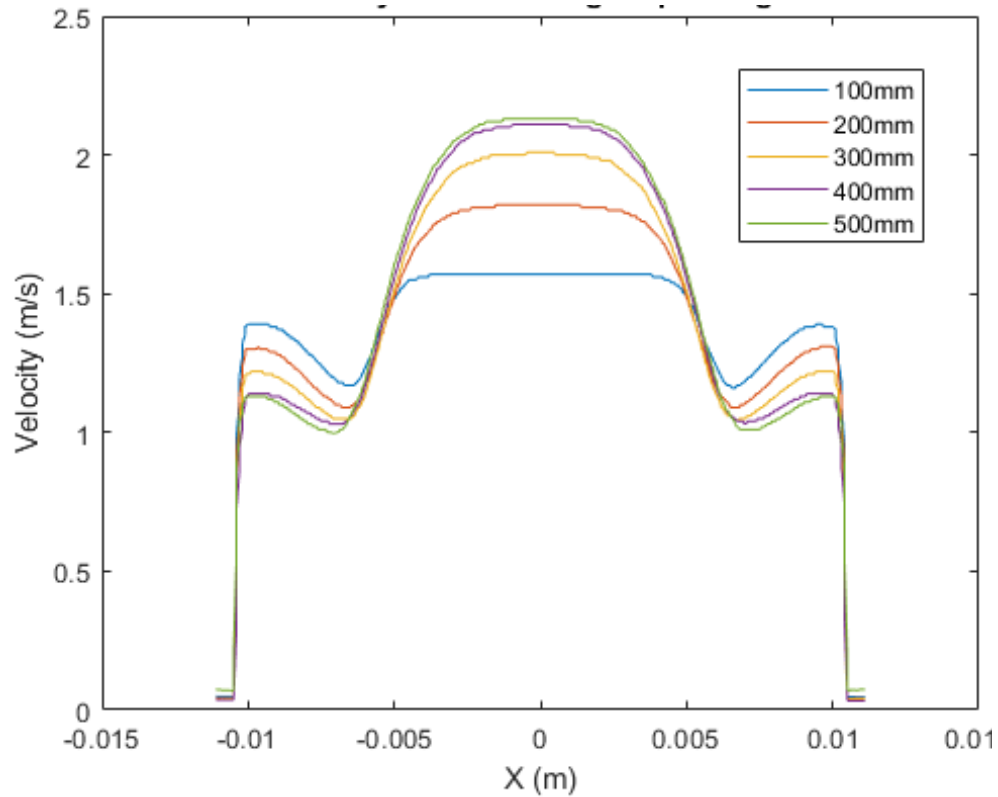


Figure 18. Fluid flow development through 500 mm section of tube

The pitch of the helical spiral is 100 mm, so each interval is in the same position as the last allowing for flow development to be characterized. The flow is considered fully developed when the profile is constant. The difference between the 400 mm and 500 mm profiles indicates that the flow is fully developed at the outlet of this simulation. Figure 19 demonstrates the centerline velocity profiles through the tube when the initial velocity profile was imported to a separate simulation.

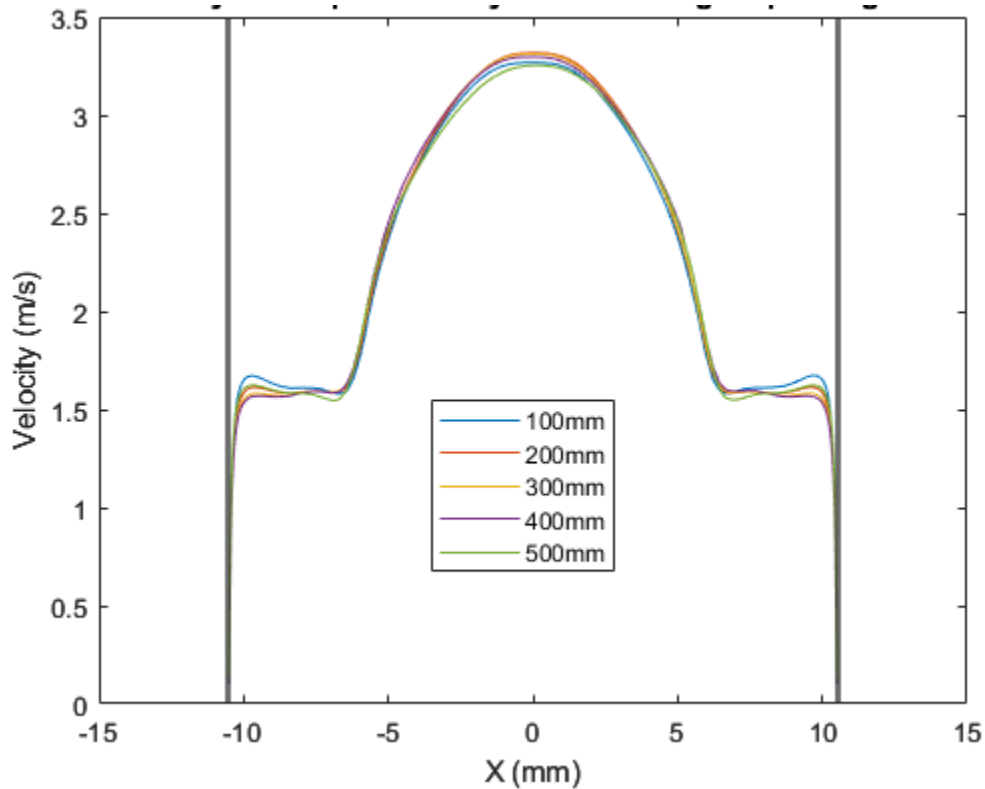


Figure 19. Fully developed flow achieved through the imported velocity profile condition

This profile is said to be fully developed as there is a constant velocity profile, which allows for the neglect of entrance effects, which are difficult to predict. This fully developed profile was then used as the inlet for a heat transfer simulation.

Figure 18 presents the development of the velocity profile through the length of tube. The initial lengths examined demonstrate the inviscid region while the outlet at 500 mm displays the beginning of a fully developed profile. Figure 19 displays the fully developed profile that was generated based on importing the outlet velocity profile as the inlet for a separate simulation. This yielded the fully developed profile with a constant velocity through the length shown in Figure 19. A velocity contour of the completed heat transfer simulation is shown in Figure 20.

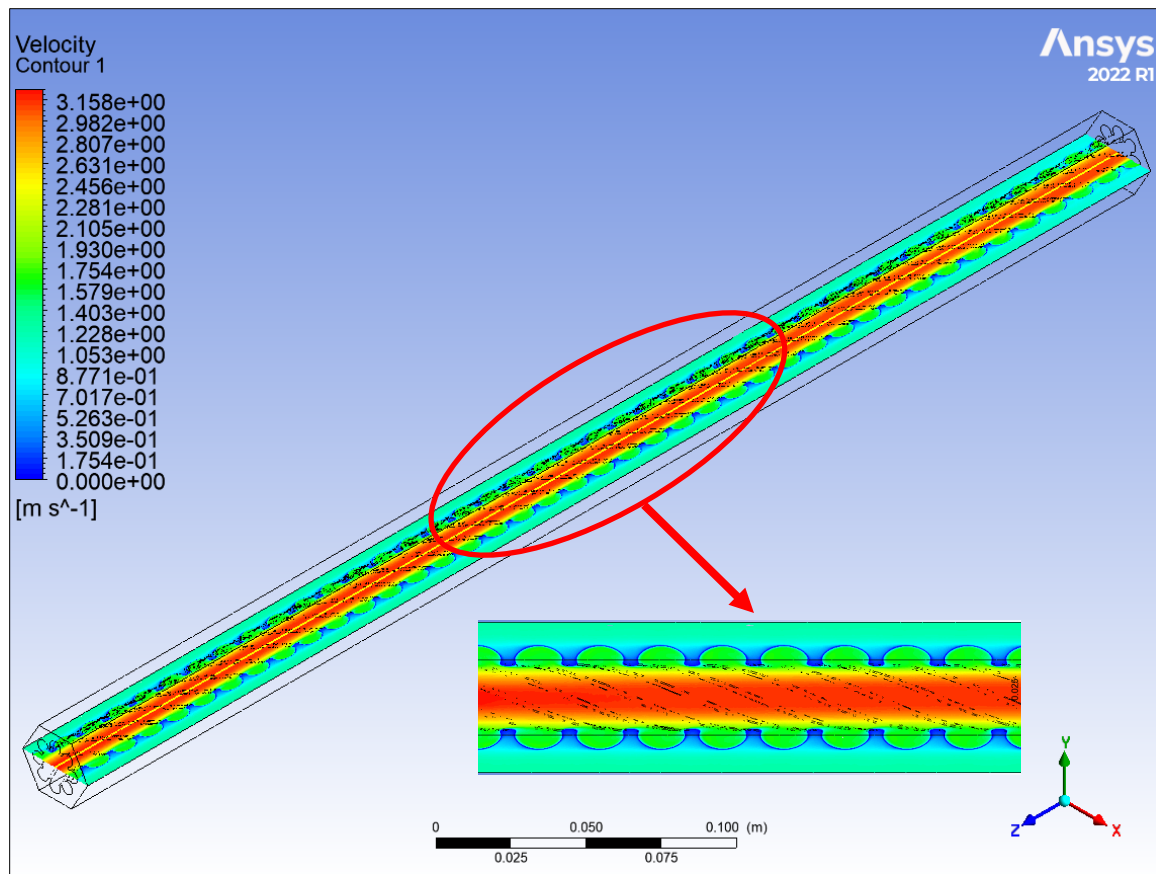


Figure 20. Centerline velocity contour from completed heat transfer simulation

The contour displays the fully developed tube velocity profile, which allows for further analysis and extrapolation of the results due to lack of thermal and hydraulic entry effects. Flow can be seen attaching to the walls of the tube, demonstrated by the decreases in the shell fluid velocity on the tube walls. This allows for improved heat transfer in these regions. Figure 21 displays a vector plot of the velocity on the shell side halfway through the system.

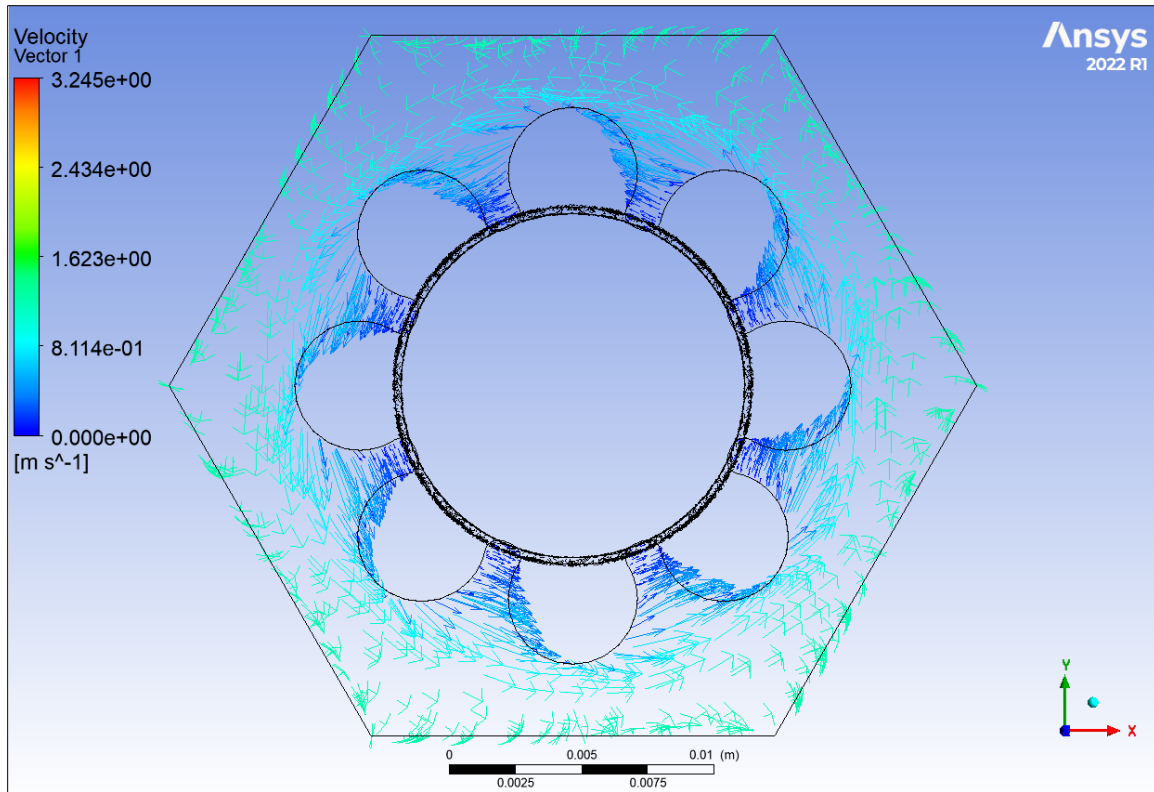


Figure 21. Vector plot of XY plane shell side fluid velocity at a z-length of 250 mm

The arrows in Figure 21 represent the magnitude and direction of sampled velocity vectors in the plane. The spiral in the tube induces a swirl in the shell side flow, which enhances heat transfer by promoting turbulence despite the two fluids being entirely in coaxial flow. This behavior is not achieved in bare tubes. Figure 22 presents the same vector plot for the tube-side flow.

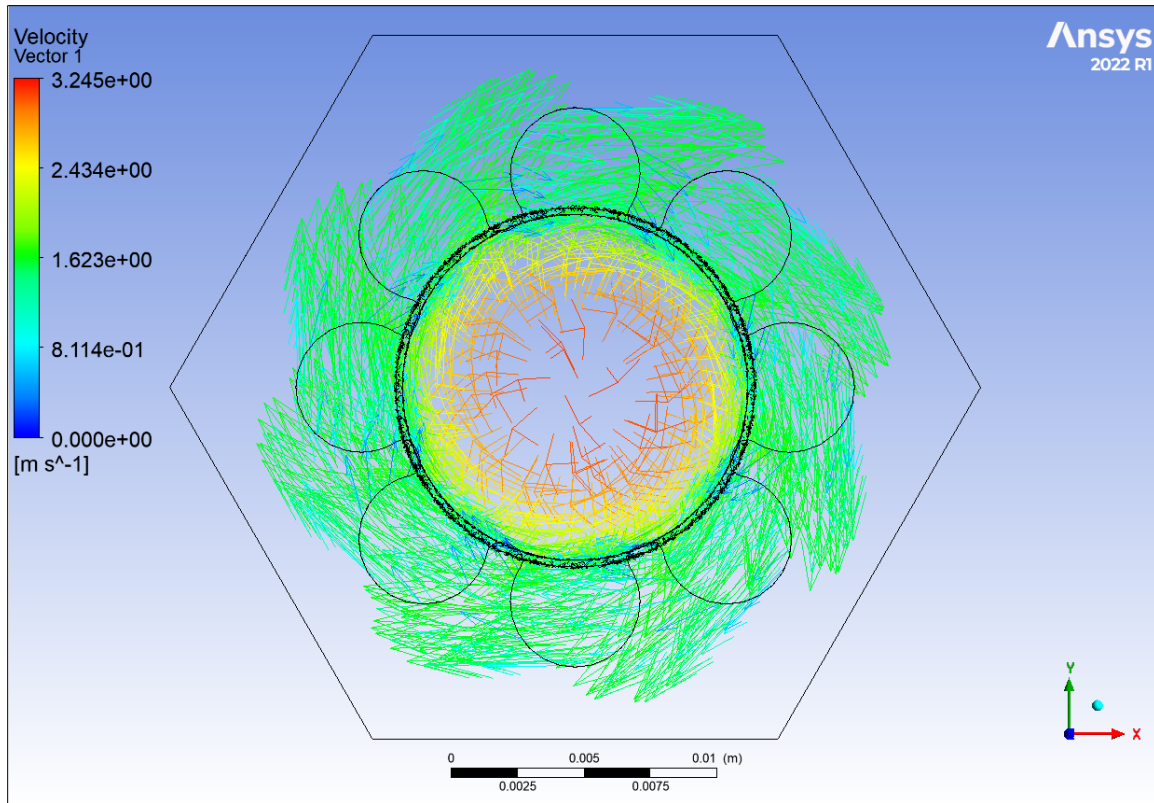


Figure 22. XY-plane slice tube velocity contour

Figure 22 presents a velocity vector plot of the eight-start spirally corrugated tube at a z-length of 250 mm. There is a central high-velocity stream though the center of the flow channel. It is also observed that the spiral corrugation has imparted a swirl into the flow, contrary to the bare tube test case where all vectors are parallel to the flow direction. This imparts turbulence in the flow, which improves heat transfer and increases pressure drop. The temperature contour for this case is shown in Figure 23, while Figure 24 presents a contour plot of the XY-plane temperature profile for each outlet.

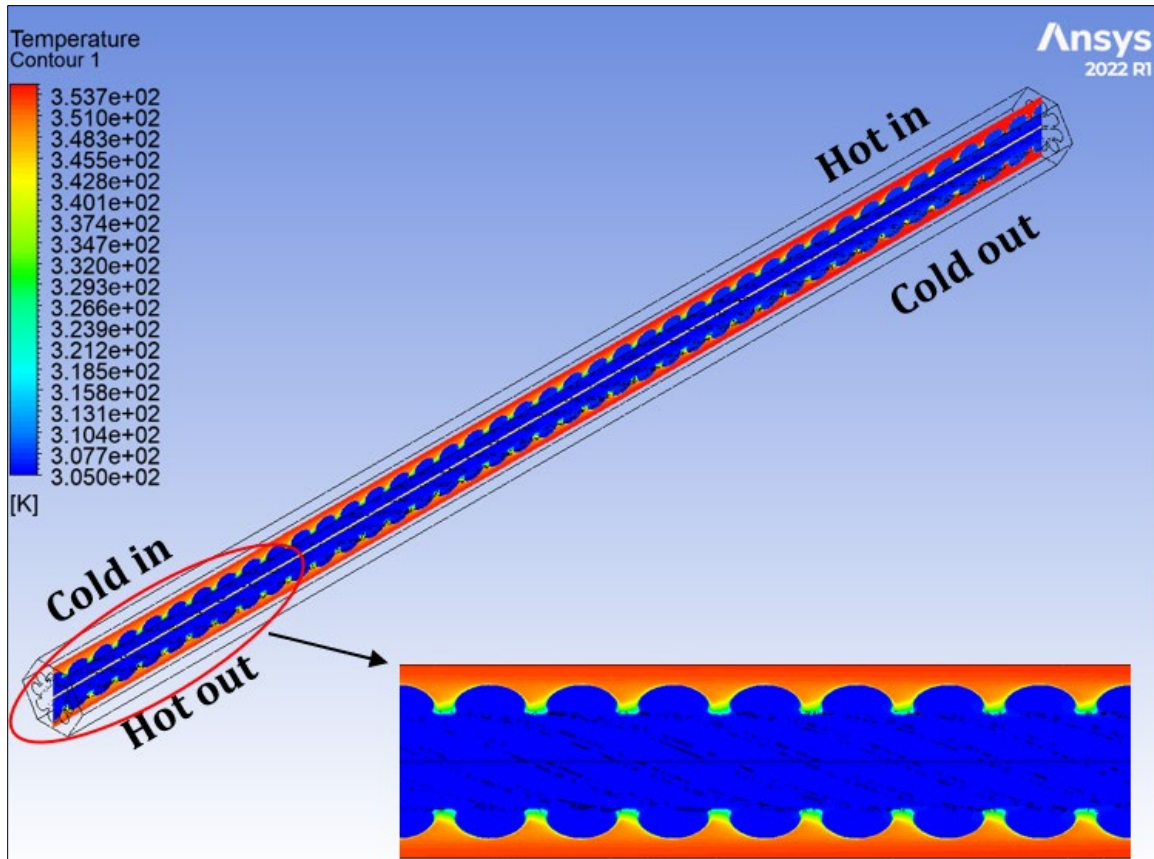


Figure 23. YZ-plane centerline temperature contour for the spirally corrugated case

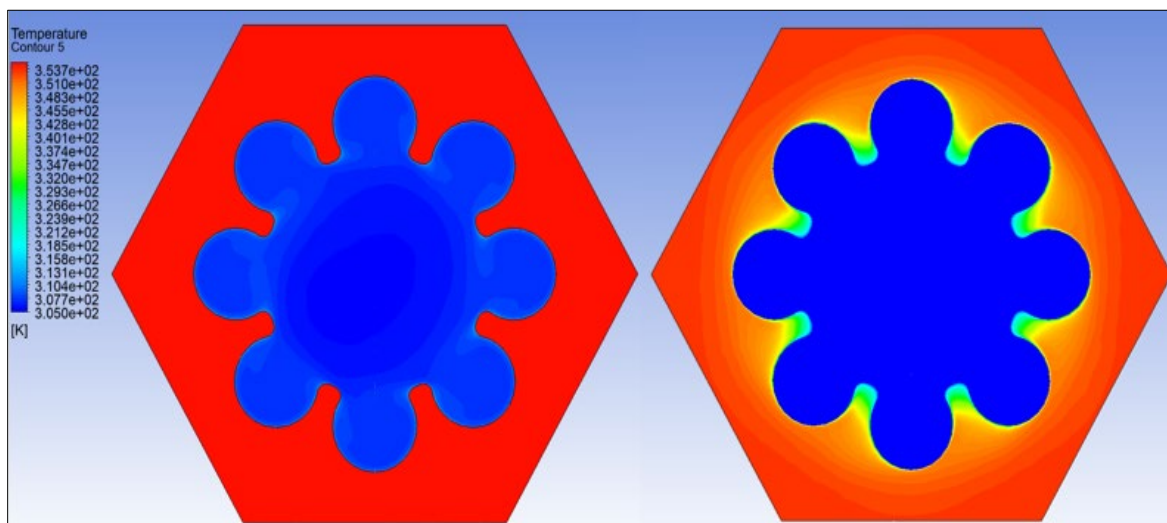


Figure 24. XY-plane temperature contour for the hot inlet, cold outlet (left) and cold inlet, hot outlet (right)

As predicted from the velocity contour plot, heat transfer is significantly enhanced in the grooves between the tube starts. This is indicated by the high temperature gradient seen in the region between starts. The hot outlet temperature was 352.208 K and the cold outlet temperature was 306.769 K. The inside and outside heat transfer coefficients were calculated as $9832 \frac{W}{m^2-K}$ and $6800 \frac{W}{m^2-K}$ respectively according to Equations (13)–(15).

1. Shell-side Flow Investigation

A separate simulation was run on the shell-side flow. This characterized a flow channel in a triangular pitch to model flow in a tube bundle assembly and ascertain a more realistic estimate of shell-side pressure drop. The results of this study are presented in Figure 25 and Figure 26.

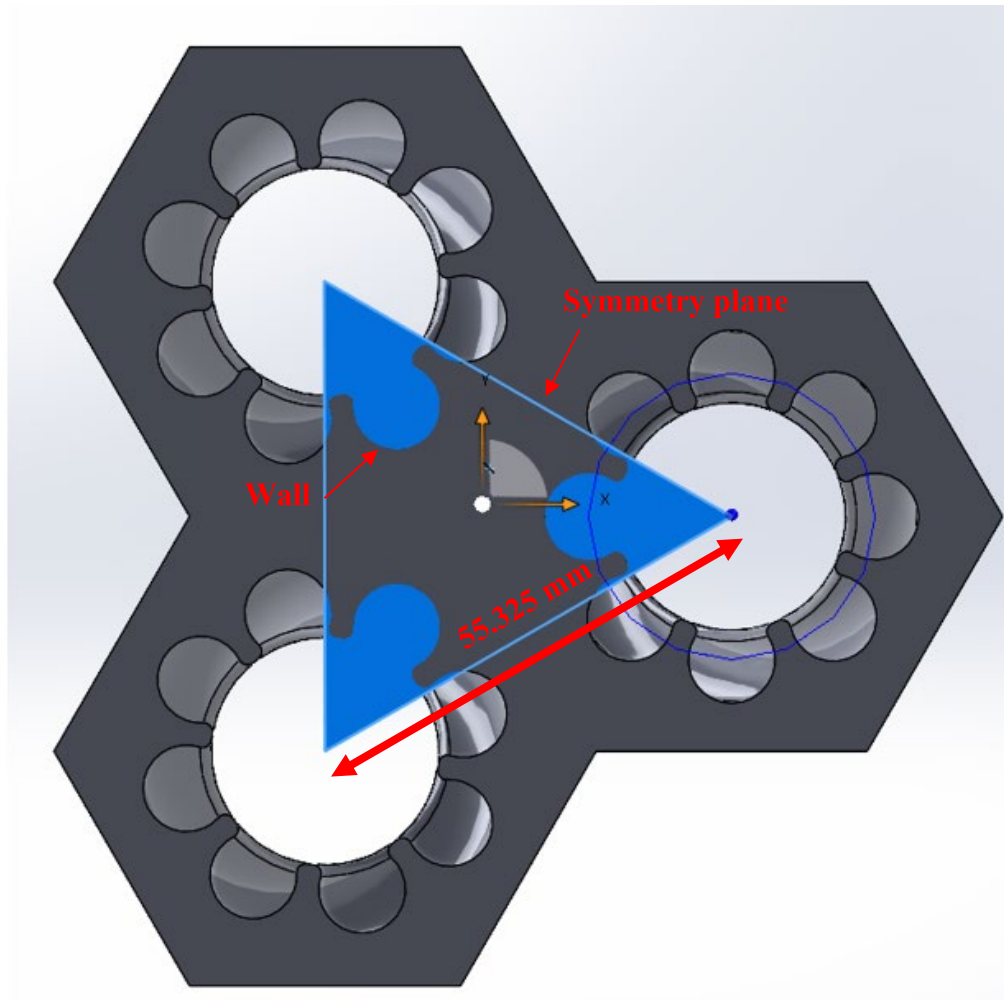


Figure 25. Visual representation of spirally corrugated tube bundle shell-side flow unit cell

Figure 25 presents a small assembly of the spirally corrugated tubes. The reason the outer geometry was chosen as a hexagon for the heat transfer simulations was to facilitate analysis of a full tube assembly. This allows for a more accurate representation of the coaxial pressure drop through the tube bundle, neglecting the effects of cross flow regions. Figure 26 demonstrates the velocity contour of flow through this arrangement.

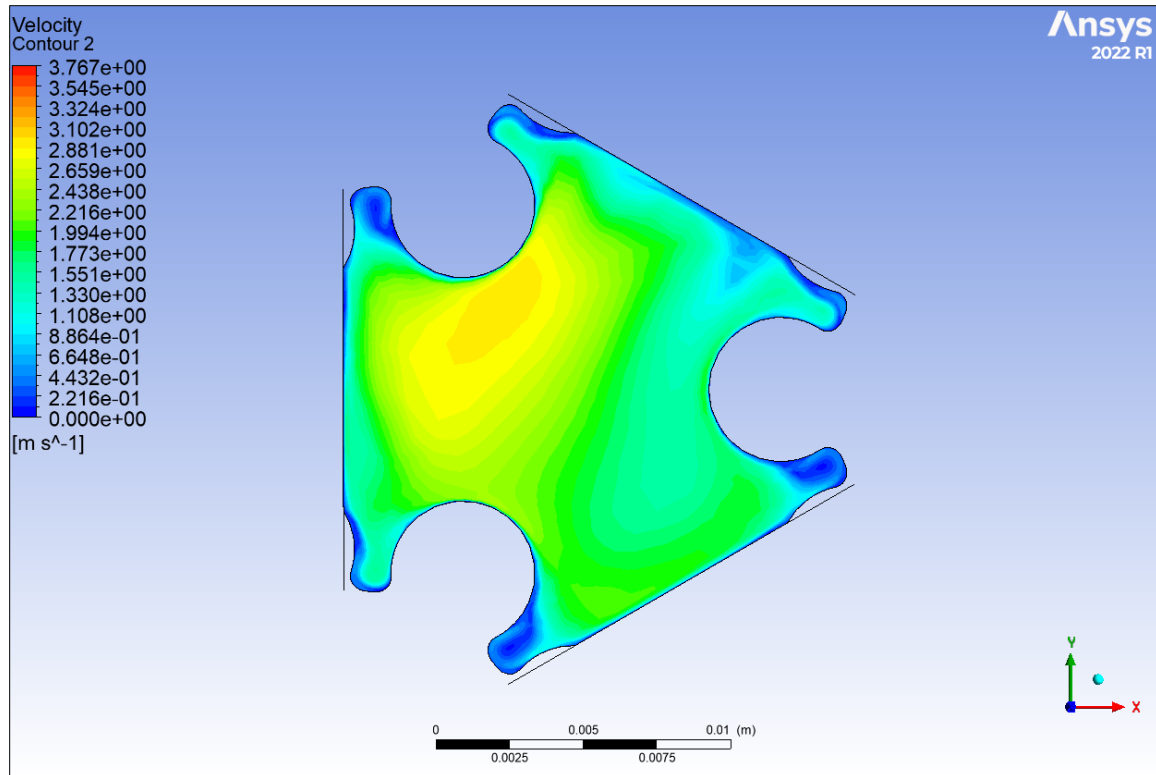


Figure 26. XY-plane velocity contour at 250 mm z-length for shell-side flow in the spirally corrugated cases

A triangular pitch was chosen to characterize the shell-side flow, similar to the bundle flow examination of the smooth tubes. The unit cell seen in Figure 26 presents an area from which a pressure drop can be determined through the calculation of a hydraulic diameter. There is also a noticeable flow stagnation in the area between the starts, where the majority of heat transfer occurred. This profile is the same between the two spirally corrugated cases examined as the twisted tape does not affect the internal area. Figure 27 presents the entire examined geometry in an isometric view.

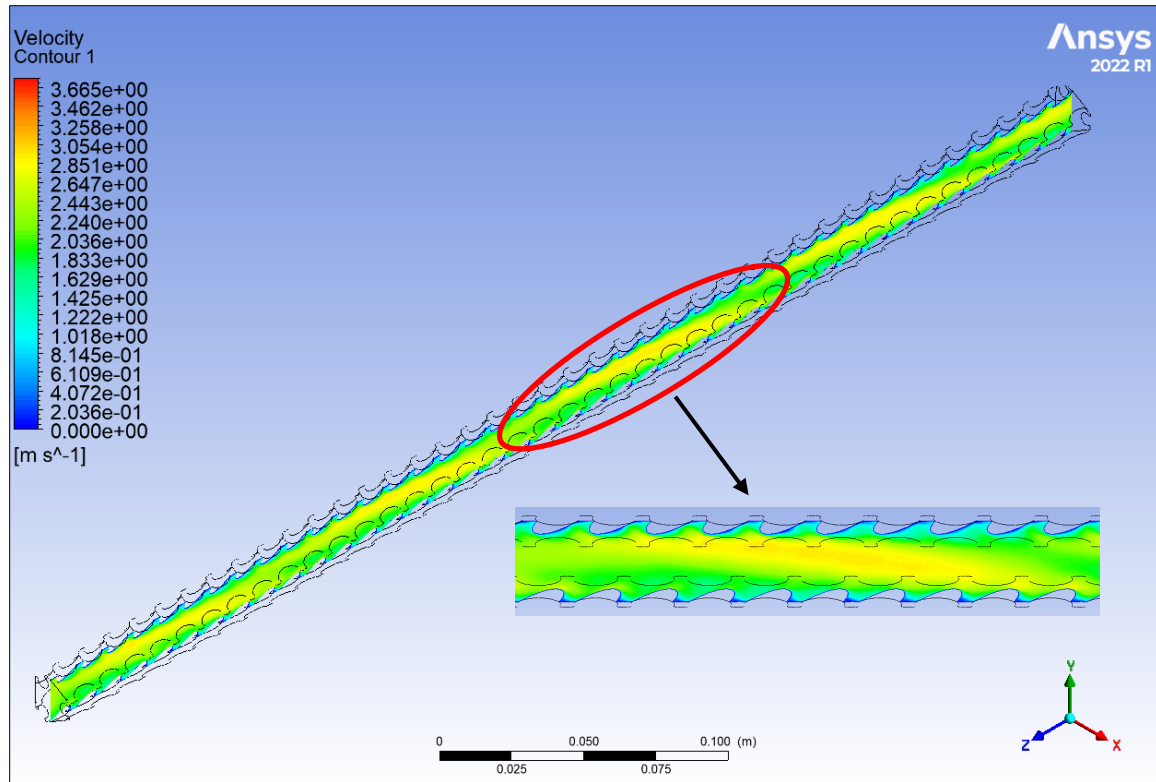


Figure 27. YZ-plane centerline velocity contour for shell-side flow in the spirally corrugated cases

Figure 27 shows the flow velocity periodicity that the geometry imparts in the shell side flow. This is evident from the cyclic rotation of the high velocity stream throughout the length. This behavior follows the 100 mm pitch of the starts, with the pattern repeating with each revolution of the tube. The overall pressure drop through this flow channel was determined to be 7.231 kPa.

The turbulence kinetic energy (TKE) was evaluated to quantize the turbulence in the shell side geometries. Initially, the $k-\omega$ SST model was used to run the simulation. This resulted in a lower turbulence in the spirally corrugated case as opposed to the bare case, which was unexpected. This abnormality prompted an investigation where it was determined that the blending function was not changing for the spirally corrugated case, indicating the model was transitioning to $k-\epsilon$ too early. In response to this, following TKE simulations were run using solely the $k-\omega$ model, which yielded the results shown in Figure 28 and Figure 29.

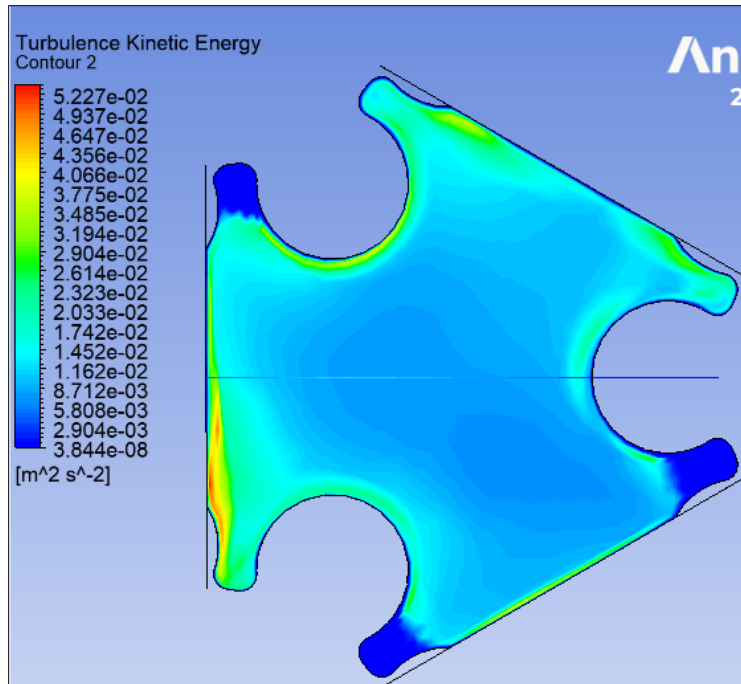


Figure 28. XY-plane contour of turbulence kinetic energy at outlet of spirally corrugated tube

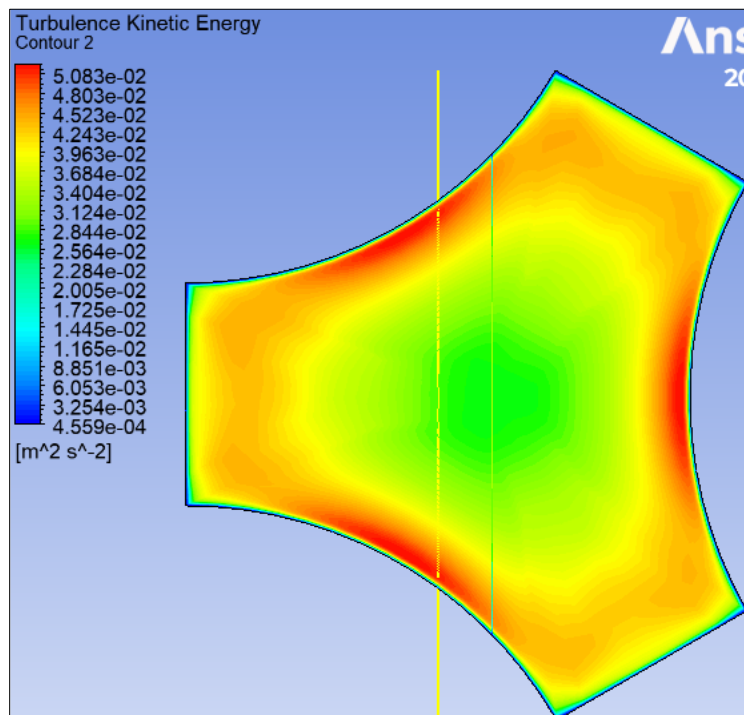


Figure 29. XY-plane contour of turbulence kinetic energy at outlet of bare tube

Figure 28 presents the turbulence kinetic energy of the spirally corrugated case. Notice the large amount of local turbulence as well as the scale. The scale in this case is four orders of magnitude lower than the bare case and the bulk turbulence is lower in the spirally corrugated case than the bare case as evident by the contours. Figure 29 presents TKE of the bare case, which behaves as expected with the majority of the turbulence occurring at the wall closest to the central annex. The expectation that the spirally corrugated case should have more turbulence prompted further investigation. Three planes were sampled at lengths along the z-axis of 0.075 m, 0.100 m, 0.200 m, 0.250 m (halfway), 0.300 m, 0.400 m, and 0.500 m (outlet) where the minimum, maximum, and average TKE were reported. The results of this examination are shown in Figure 30 through Figure 32.

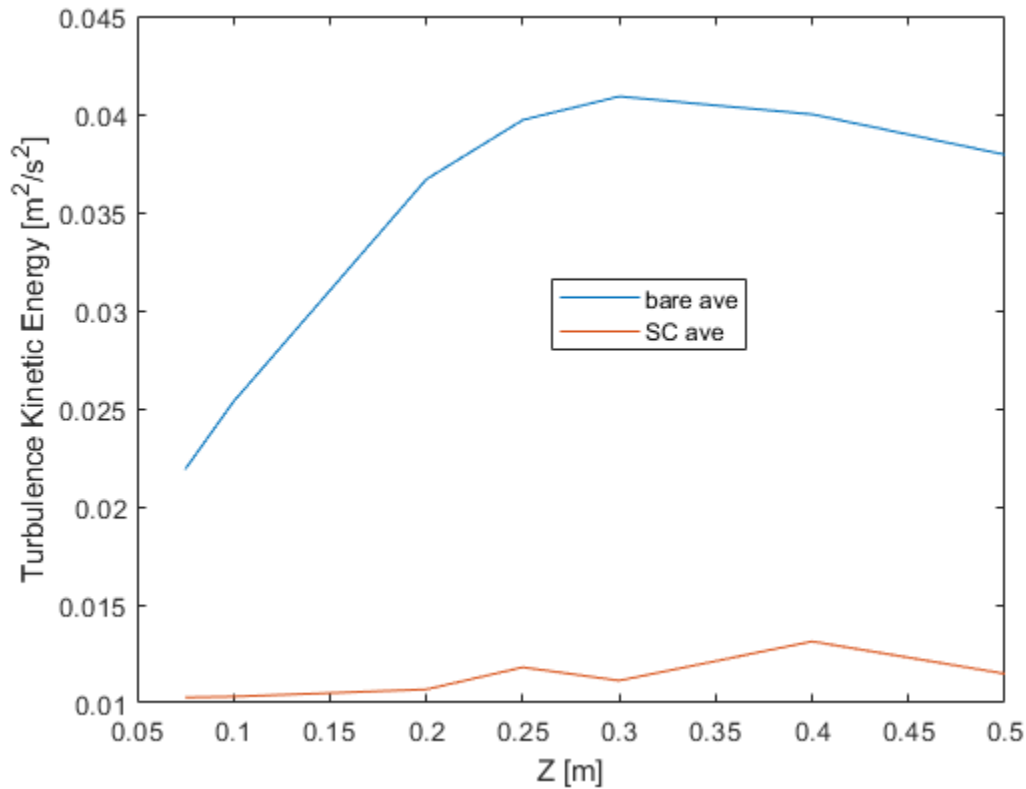


Figure 30. Shell-side flow average turbulence kinetic energy through tube

The area-averaged turbulence kinetic energies at three points along the tube length are shown in Figure 30. This demonstrates, as observed with the contour plots, that the

turbulence is higher in the bare tube case than in the spirally corrugated cases. Further details can be extracted by looking at the minimum and maximum TKE.

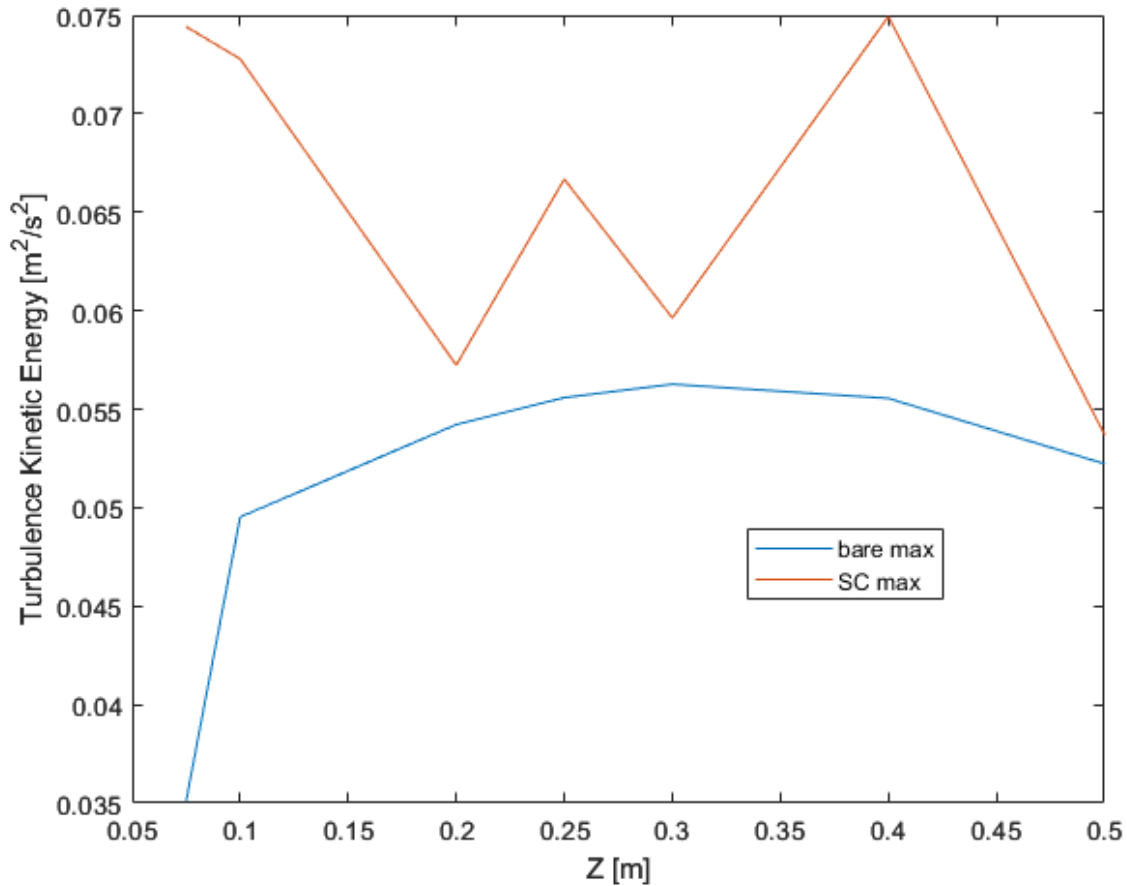


Figure 31. Shell-side flow maximum turbulence kinetic energy through tube

Figure 31 shows that the maximum TKE in the bare case is lower than that of the spirally corrugated cases, which is as expected due to the enhanced swirl in the spirally corrugated flow. In fact, all of the maximum values are higher than those observed in the bare case. The minimum cases shown in Figure 32 display why the average is skewed in favor of higher TKE in the bare case.

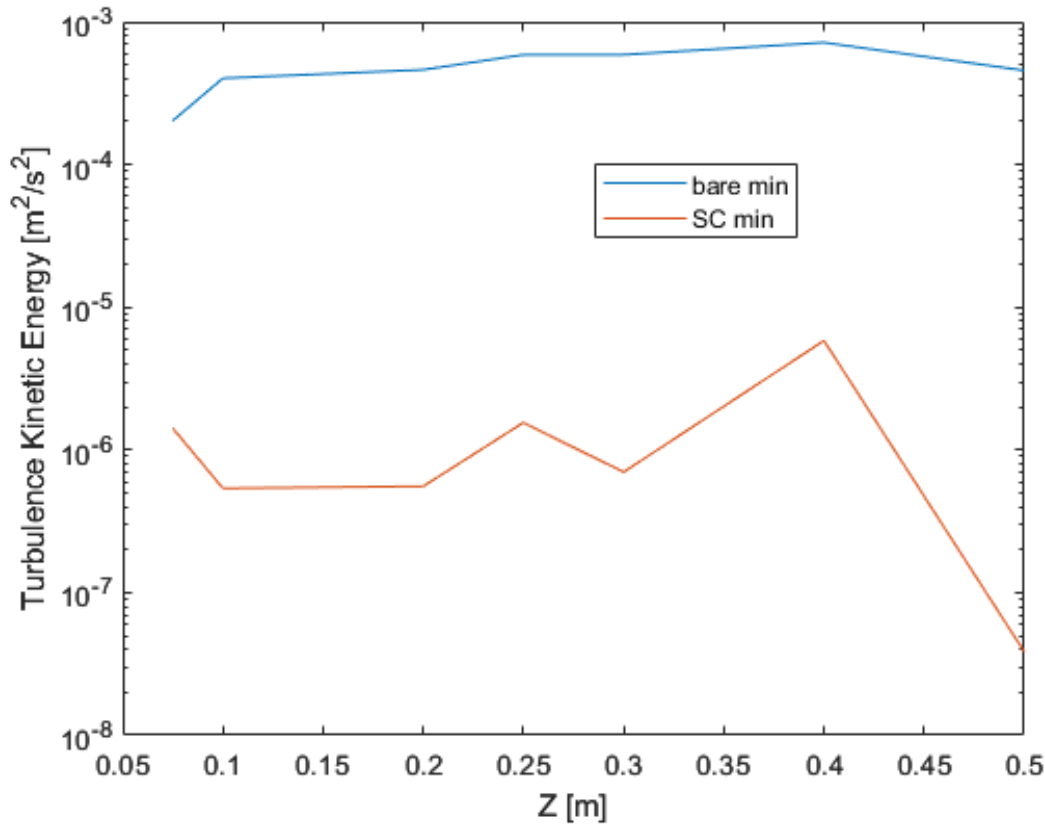


Figure 32. Shell-side flow minimum turbulence kinetic energy through tube

Figure 32 presents the local minimum values of TKE sampled on the same plane as the previous cases. Note that the y-axis is a log-scale, indicating that the TKE for the spirally corrugated case is significantly lower than that of the bare tube, which is driving down the average. The abnormality in the flow seems to indicate that a large-eddy simulation (LES) is required to fully capture the turbulence behavior in the flow. LES simulations are closer to approximating a direct numerical solution, which increases their computational cost and accuracy. This type of simulation is most likely required to adequately capture the eddies that are created in the flow by the spirally corrugated geometry. Figure 33 presents a plot of the centerline TKE through the spirally corrugated geometry.

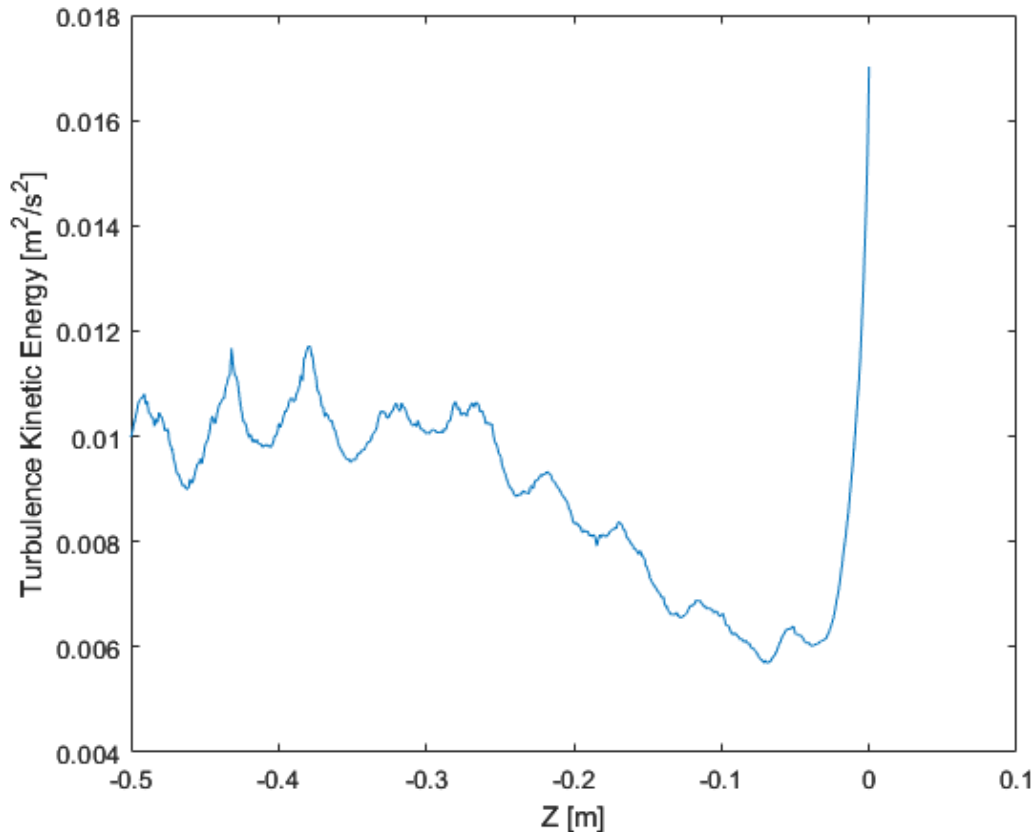


Figure 33. Centerline turbulence kinetic energy for spirally corrugated tube bundle flow

The turbulence kinetic energy through the spirally corrugated tube bundle indicated that there is an unsteady flow as the fluid moves from the inlet to the outlet. This is evident from the sawtooth pattern towards the outlet in Figure 33. This analysis points to an unsteady simulation needing to be run to fully capture the turbulence behavior on the shell side of the spirally corrugated tube bundle. A large eddy simulation would be more effective at capturing this behavior than the $k-\omega$ model.

C. EIGHT-START SPIRALLY CORRUGATED TUBE WITH TWISTED TAPE INSERT

The shell side velocity profile is identical between the twisted tape spirally corrugated tube and the standard spirally corrugated tube because there is no external geometry change. The fully-developed centerline velocity profile for this case is shown Figure 34.

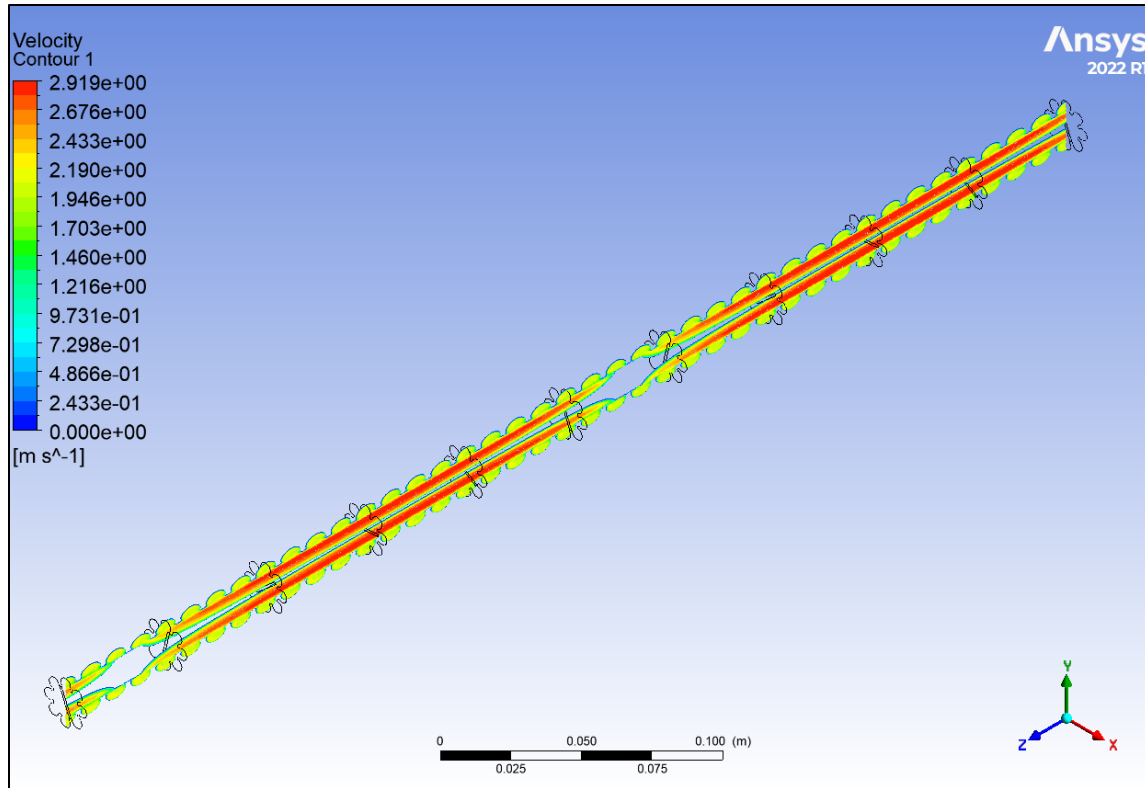


Figure 34. YZ-plane centerline velocity contour in twisted tape case

The results presented in Figure 34 were achieved by importing the fully developed outlet profile of a previous simulation to a second simulation where the flow was fully developed throughout. This had the effect of reducing the central jetting and promoting additional turbulence and more uniform velocity in the tube. Figure 35 demonstrates the velocity vector plot for the tube side flow.

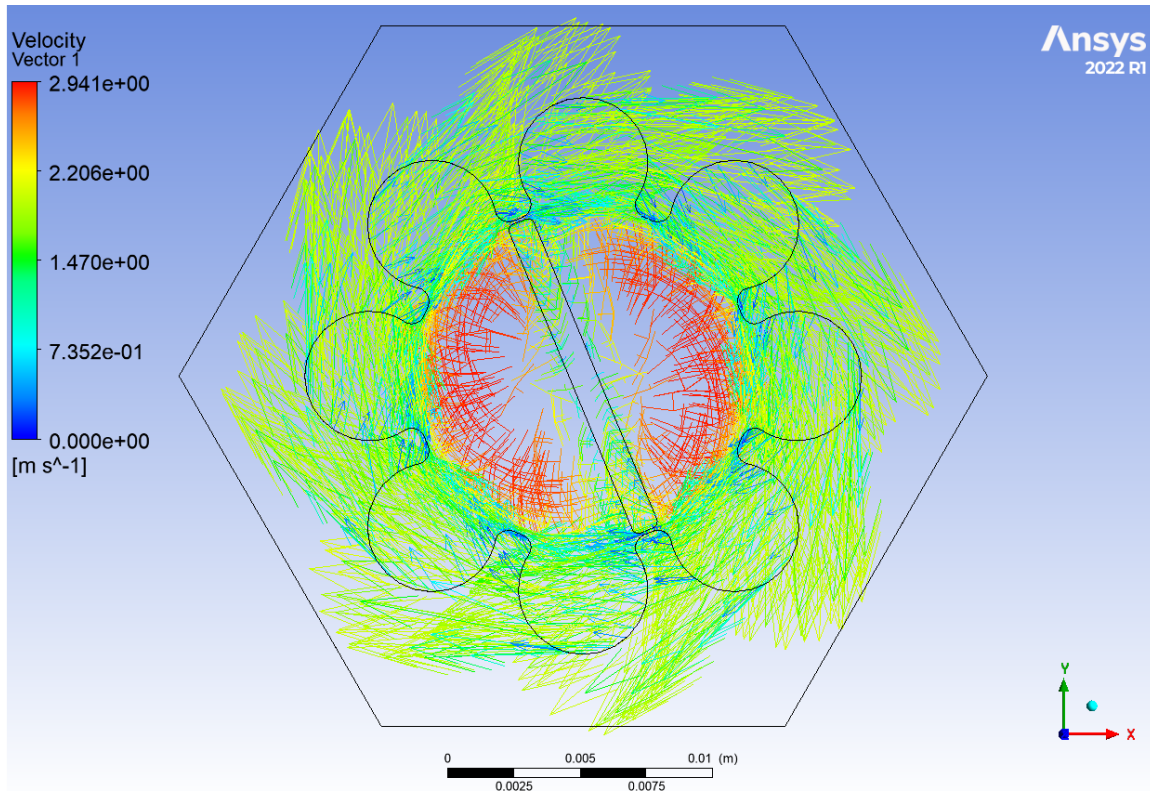


Figure 35. XY-plane velocity vector plot at 250 mm z-length for twisted tape case

The twisted tape provides significantly more swirl to the flow, as evident from the vector plot shown in Figure 35. There is higher velocity in the tube starts and the central high-velocity jet has become smaller due to the addition of the twisted tape. This design has a significantly higher pressure drop of 4.877 kPa compared to the standard spirally corrugated tube pressure drop of 2.508 kPa. The temperature gradient is shown through the temperature contours in Figure 36 and Figure 37.

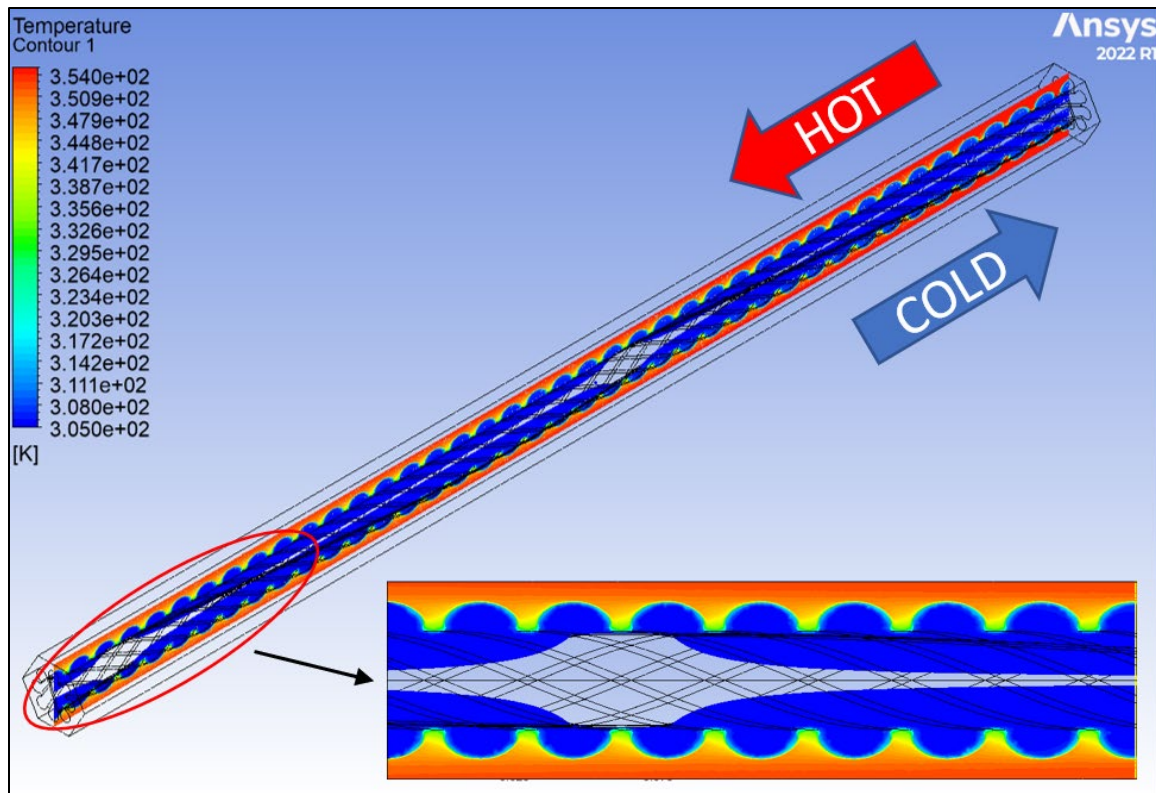


Figure 36. YZ-plane centerline temperature contour for the twisted tape case

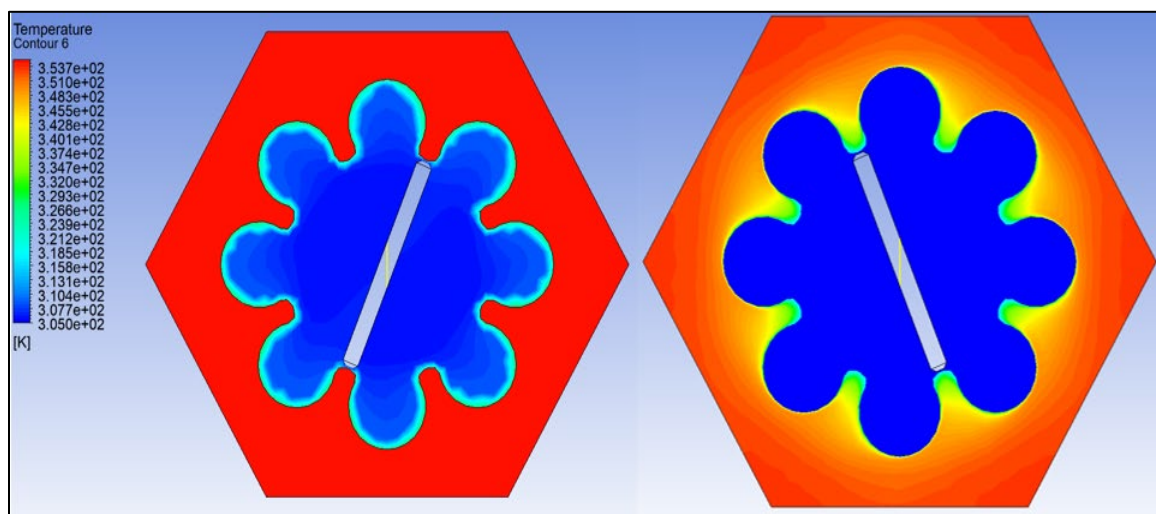


Figure 37. XY-plane temperature contour for the hot inlet, cold outlet (left) and cold inlet, hot outlet (right) for twisted tape case

The twisted tape scenario significantly increased heat transfer in the starts as seen in Figure 36. It should also be noted that the cold fluid carries significantly more energy than the previous case, as shown in Figure 37. The final reported outlet temperatures for this case were 351.463 K and 307.188 K for the hot and cold outlets respectively. This is based on a hot inlet temperature of 355 K and a cold inlet temperature of 305 K. The inside heat transfer coefficient was calculated as $9832 \frac{W}{m^2-K}$ according to Equations (13)–(15). The outside heat transfer coefficient is equal to the spirally corrugated case as the external geometry did not change with the addition of the twisted tape.

D. TURBULENCE KINETIC ENERGY COMPARISON

To better characterize the behavior of the tube-side flow, the turbulence kinetic energy was analyzed. Figure 38 shows a comparison of the TKE along the y-axis at a z-length of 250 mm (halfway through the tubes) for all three cases.

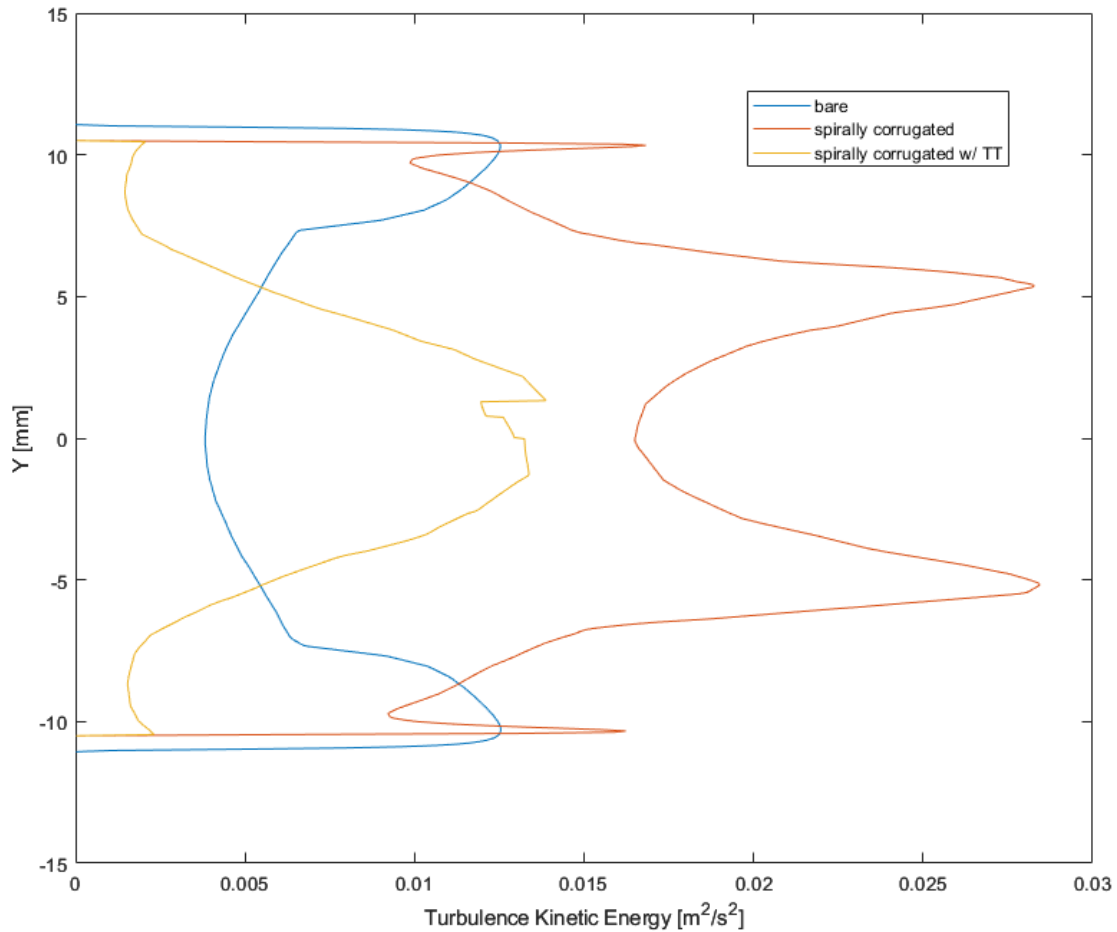


Figure 38. Comparison of turbulence kinetic energy along y-axis hallway through the tubes

The spirally corrugated and spirally corrugated with twisted tape insert cases had increased turbulence in the flow. Along the walls, there is a marked increase in the TKE indicating increased turbulence in the flow. The twisted tape case has a higher magnitude of TKE at the center due to the tape being present in the middle of the tube. There is higher localized turbulence closest to the tape due to the effects of the boundary layer on the free-stream flow. Figure 39 presents a plot of the TKE through the z-length of tube for the bare and spirally corrugated cases.

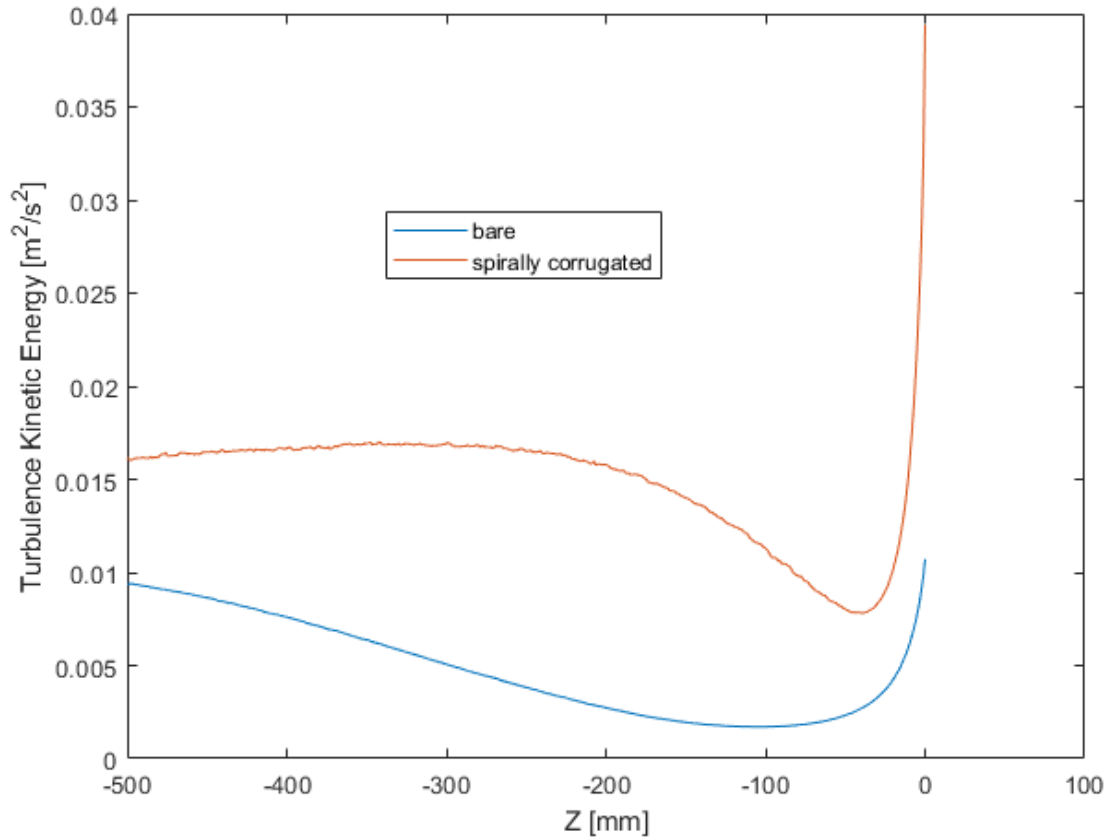


Figure 39. Comparison of turbulence kinetic energy through length of tube for bare and spirally corrugated cases

In Figure 39, the inlet is located at zero (0) mm and the outlet is located at negative 500 mm. A direct comparison was unable to be drawn along the length for the twisted tape case as the tape runs through the centerline and there would be no sampling in this region as it is a solid. The TKE for the spirally corrugated tube reaches a larger value than the bare tube, indicating the spiral geometry imparts increased turbulence in the flow. Additionally, the TKE increases at a substantially larger rate through the centerline in the spirally corrugated case.

V. RESULTS

The tabulated results from all simulations run are shown in Table 4.

Table 4. Results from all simulations conducted

	Bare	Spirally corrugated	Spirally corrugated with twisted tape
Pressure drop [kPa] (tube)	0.398	2.508	4.877
Pressure drop [kPa] (shell)	3.290*	7.231*	7.231*
Overall heat transfer coefficient (UA) [$\frac{W}{K}$]	66.12	76.25	97.80
Total heat transfer area [m^2]	278	240 (13.3% reduction)	187 (32.3% reduction)
Cost (\$)	\$2,250,000	\$3,135,455	\$3,155,267

* Shell-side values reported only as coaxial flow pressure drops.

The results of the study indicate that the spirally corrugated tube led to a reduction in overall heat transfer surface area (13% reduction for the spirally corrugated case and 32.3% reduction in the twisted tape case). This is due to the larger thermal efficiency of the design. The area is further reduced when considering the spirally corrugated tubes have 30% increased surface area for the same diameter of a bare tube. Pressure drop was greatly increased by the amount of turbulence promoted in the flow due to the swirled design. The shell side pressure drop is greater than anticipated; however, the tube side has reasonable agreement to experimental correlations. The results of the sizing analysis are presented graphically in Figure 40 and Figure 41.

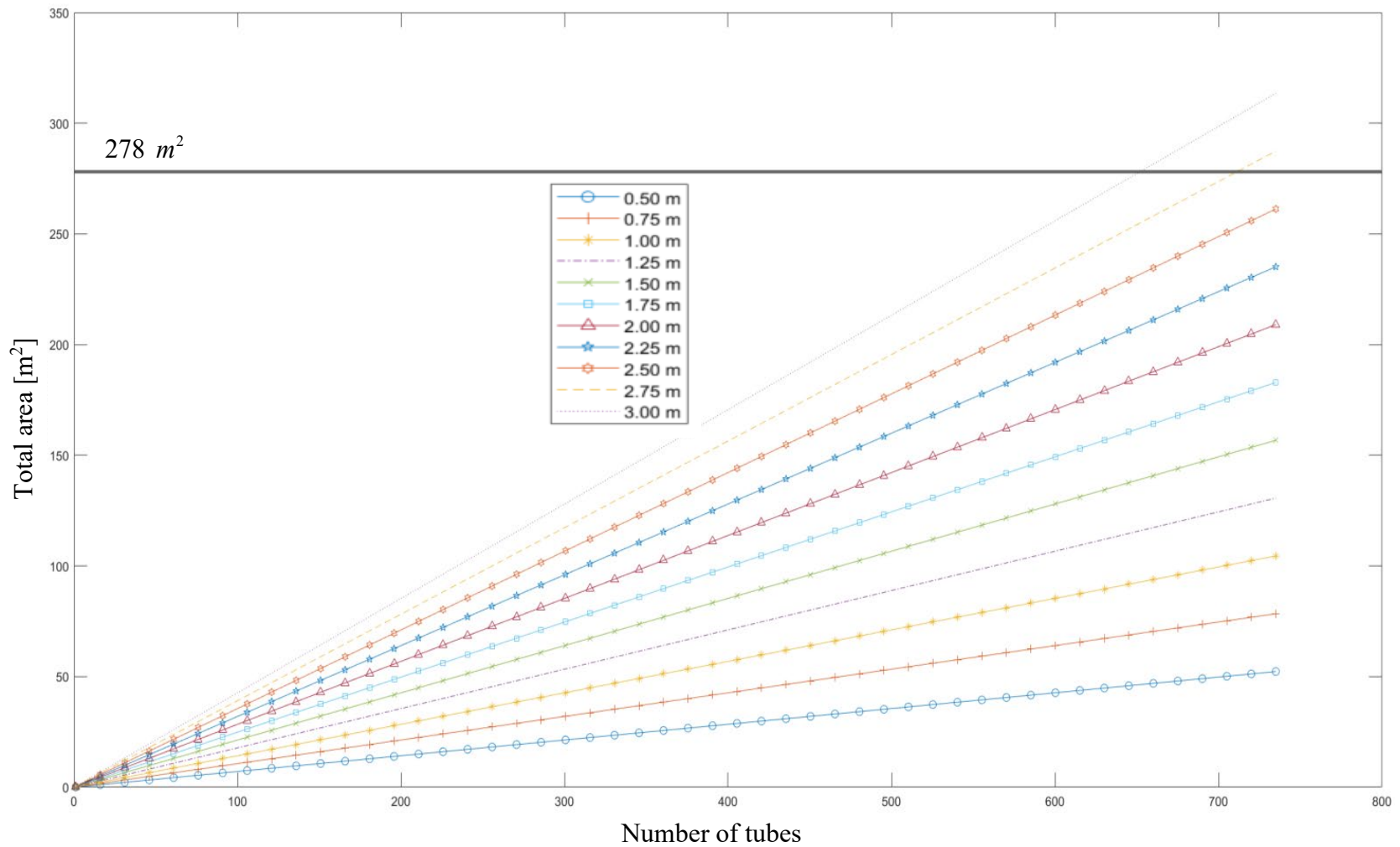


Figure 40. Total area as a function of length and number of tubes (bare case)

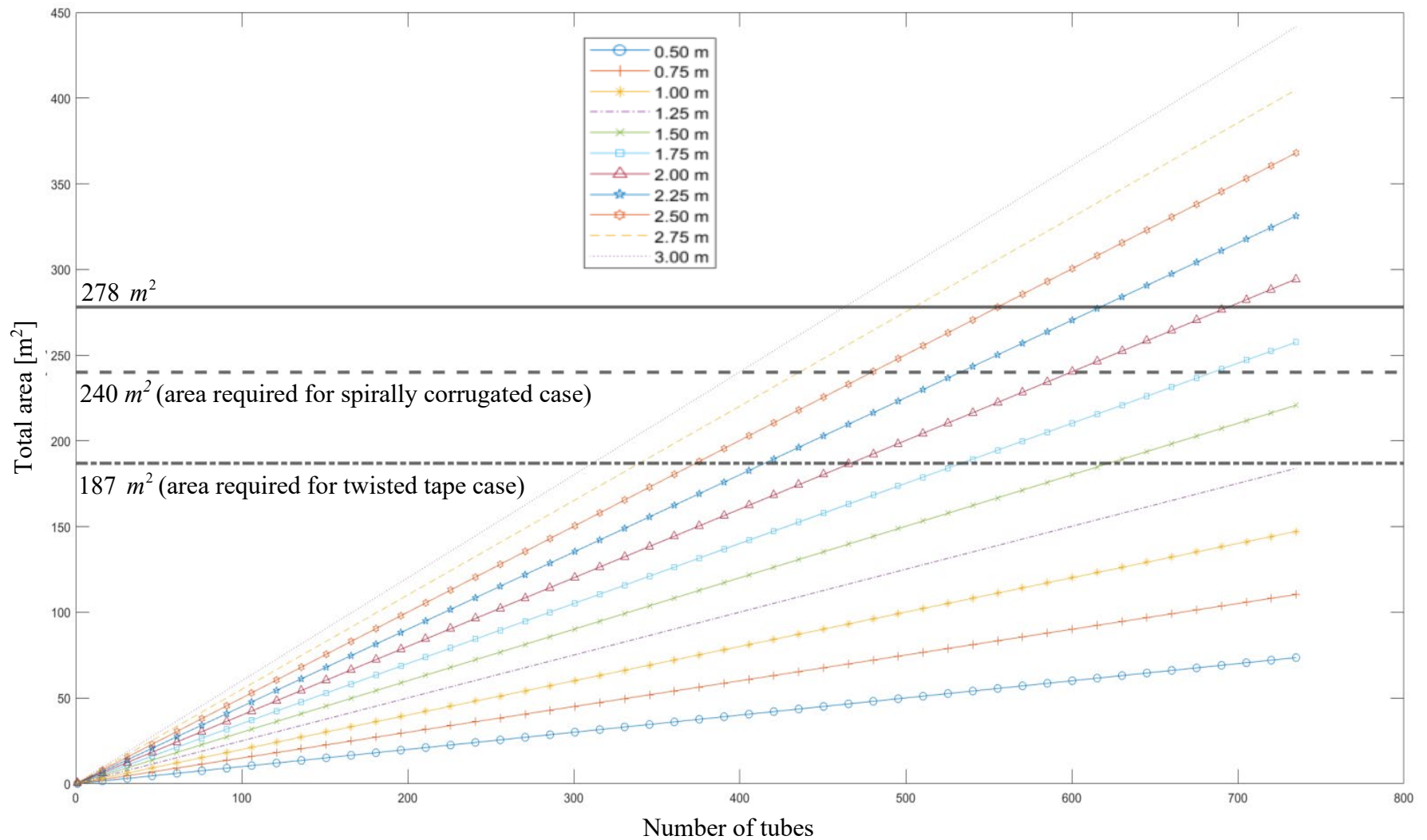


Figure 41. Total area as a function of length and number of tubes (spirally corrugated cases).

Figure 40 uses the bare tube dimensions to determine an overall heat transfer area required based on an overall length (shown in the legend) and the number of tubes, displayed on the x-axis. The black line indicates the minimum required area to dissipate the 5.861 MW the heat exchanger was tasked with managing. Each condition above the black line satisfies this condition for the bare tube. Figure 40 uses the dimensions of the spirally corrugated tubes, which have a 30% increased surface area compared to the bare tubes, to determine an overall heat transfer area for the spirally corrugated and the spirally corrugated with twisted tape insert cases. The dashed line represents the calculated area required for the spirally corrugated tube and the dot-dashed line represents the required area for the spirally corrugated tube with twisted tape insert.

Figures 40 and 41 indicate that the spirally corrugated tube geometries can significantly decrease the size of the overall heat exchanger unit. The spirally corrugated cases also allow for a wider range of configurations that satisfy the design requirements provided by the critical parameters. The maximum number of tubes was set as 735, as this is the maximum amount of tubes that would be able to fit into a 0.75 m x 0.75 m area with a pitch to diameter ratio of 1.25. Using these conditions, an overall reduction in length by up to 1.5 m (50% of the total length) is possible.

VI. CONCLUSION

The goal of this research was to analyze the thermal-hydraulic performance of additively manufactured spirally corrugated tubes for use in a heat exchanger. The geometry was parameterized utilizing a MATLAB code that allowed for basic preliminary analysis with respect to parameters laid out by the sponsor of the project. This code allowed a final geometry to be modeled in CAD software SolidWorks. A computational fluid dynamics study was then completed on the CAD model using ANSYS CFX. A mesh sensitivity study was completed to ensure a solution independent of mesh size and boundary conditions were applied. The solutions were examined for convergence based on how well they satisfied the laws of conservation of mass, momentum, and energy equations. Results of the CFD study were presented utilizing plots and all results were tabulated to compare the thermal and hydraulic performance of the spirally corrugated tubes to a bare test case. Overall heat transfer coefficients were determined from the CFD simulations. These results informed the calculation of an overall heat transfer surface area, which then allowed for the estimation of unit size and cost based on correspondence with vendors.

This study determined that the use of additively manufactured eight-start spirally corrugated tubes has the potential to significantly improve the thermal-hydraulic performance of heat exchanger tubes. Leveraging additive manufacturing allows for the implementation of complex geometries that have a higher surface area compared to traditional tubes. Improving heat transfer surface area improves the overall efficiency of a heat exchanger. This, combined with the improved turbulence promoted by the spiral corrugations, leads to increased heat transfer and higher pressure drops. The tubes examined in this study met up to the performance metrics provided by the Navy. The tubes are more expensive than the legacy design. In addition to the more efficient heat transfer, the spirally corrugated tubes have 30% more area, which further decreases the size of the overall unit.

The proposed spirally corrugated tube had twice the inside and outside heat transfer coefficient compared to the bare tube. This is largely due to the promotion of turbulence in

the spirally corrugated tube, as demonstrated by the comparisons of turbulence kinetic energy. Additionally, holding the number of tubes constant, the overall heat exchanger length can be reduced by a maximum value of one half utilizing the spirally corrugated tubes with twisted tape inserts. Despite the increase in heat transfer performance, the spirally corrugated tubes yielded a pressure drop an order of magnitude higher than the bare tubes and the twisted tape scenario doubled the pressure drop of the standard spirally corrugated tube. This value of pressure drop is likely an underestimate as smooth tubes were assumed. The increased pressure drop and pumping power must be weighed with respect to the overall increase in heat transfer capability and the overall 50% possible reduction in unit length.

The scope of this study examined solely fully developed flow without consideration for head development or external piping. The inclusion of head development in the flow will significantly increase the pressure drop of the overall system. Calculations were greatly simplified by assuming fully developed flow, and this provides a reasonable estimate of thermal-hydraulic performance. In future studies, the examination of head development will further increase heat transfer and will also incur a higher pressure drop. Future work in examining these tubes should include an examination of fluid flow through an entire tube bundle (considering both coaxial and crossflows) to get a more realistic insight into what the overall performance of a heat exchanger implementing these geometries would be. There is also room for further improvement in the design of the tubes themselves. The eight-start geometry was selected and examined based off of a 19.05 mm inner diameter tube. Further improvements to the design can examine how corrugation pitch, depth, and spacing affect the thermal-hydraulic performance of the tubes. These characteristics could then be optimized to produce a tube with the highest relative heat transfer coefficients and lowest relative pressure drops. The twisted tape insert significantly increased the pressure drop, and this could be mitigated by optimizing the pitch of the tape and starts. Future work should consider examining the effect of the tape pitch and the spiral corrugation pitch to determine optimal arrangements to increase heat transfer while minimizing pressure drop losses. Another recommendation is to run a large-eddy simulation on the spirally corrugated bundle assembly. Abnormal fluid behavior was noted

when analyzing this case, and the investigation of the turbulence behavior in the spirally corrugated assembly may allow for the optimization of outer geometry (perhaps a combination of left and right-handed starts) that promote the most turbulence on the flow.

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX A. MATLAB SCRIPT

```
%% Multistart tube size and pressure drop
% ENS Noah Byrne, Naval Postgraduate School
% This code uses the inner diameter, number of starts, and diameter of the
% starts to predict area and outer diameter for a spirally corrugated HX
% tube design allowing for the visualization and size determination before
% finalizing a design.

clear all
close all
clc

%% Enter Parameters

t = 1.651;           % tube thickness, mm
t_start = 0.559; % start thickness, mm
ID = (16.13);        % nominal inner diameter, mm
D_start = (6);       % nominal start diameter, mm
numstarts = 6;       % number of starts

%% Calculations

A_inner = (ID^2*pi) / 4;
A_starts = numstarts*(D_start^2*pi) / 4;

d = (ID)/2;
r_starts = (D_start)/2;
r_inner = (ID)/2;

% Plot design with a single start

syms x y
fimplicit((x).^2 + (y).^2 - (d).^2)
hold on
fimplicit((x-(d)).^2 + (y).^2 - (D_start/2).^2,[-ID ID])
axis equal

% let R be radius of the inner channel
% let r be radius of the starts

r = r_starts;
R = r_inner;
x = (d^2 - r_starts^2 + r_inner^2) / (2*d);
d1 = (d^2 - r^2 + R^2) / (2*d);
d2 = d-x;
y = ((4*d^2*r_inner^2 - (d^2 - r_starts^2 + r_inner^2)^2)) / (4*d^2);

% solve for chord length, a
a = (1/d)*sqrt((-d+r-R)*(-d-r+R)*(-d+r+R)*(d+r+R));

% solve for intersection area between starts and inner diameter

A_intersect = numstarts*(r^2*acos((d^2+r^2-R^2)/(2*d*r)) + ...
    R^2*acos((d^2+R^2-r^2)/(2*d*R)) - ...
    0.5*sqrt((-d+r-R)*(-d-r+R)*(-d+r+R)*(d+r+R)));
```

```

% solve for area of inner diameter and starts combined
A_internal = (A_starts - A_intersect) + A_inner;
% disp('Internal area (mm^2) is:')
% disp(A_internal)

% find outer diameter
OD = (d+r_starts)*2;
disp('Outer diameter (mm) is:')
disp(OD)

%% Solve for perimeter

% large circle arc
thetaR = acos((R^2 + d^2 - r^2) / (2*R*d));
betaR = (2*pi) - (2*thetaR);
arcR = betaR*R;
circR = 2*pi*R;

% small circle arc
alphan = asin((R*sin(thetaR)/r));
gammar = (2*pi) - (2*alphan);
arcr = gammar*r;
circular = 2*pi*r;

Pstarts = (numstarts*arcr);
Pinner = circR - (numstarts*(circR-arcR));
P = Pstarts+Pinner;
% disp('Outer perimeter (mm) is:')
% disp(P)

%% Factor in thickness

r2 = (D_start - (2*t_start))/2;
R2 = (ID-(2*t))/2;

A_inner2 = pi*R2^2;
A_starts2 = numstarts*pi*r2^2;

x2 = (d^2 - r2^2 + R2^2) / (2*d);
d1_2 = (d^2 - r2^2 + R2^2) / (2*d);
d2_2 = d-x;
y_in = ((4*d^2*R2^2 - (d^2 - r2^2 + R2^2)^2)) / (4*d^2);

A_intersect_in = numstarts*(r2^2*acos((d^2+r2^2-R2^2)/(2*d*r2)) + ...
    R2^2*acos((d^2+R2^2-r2^2)/(2*d*R2)) - ...
    0.5*sqrt((-d+r2-R2)*(-d-r2+R2)*(-d+r2+R2)*(d+r2+R2)));

A_internal_in = (A_starts2 - A_intersect_in) + A_inner2;
disp('Internal flow area (mm^2) is:')
disp(A_internal_in)

% Solve for internal shape perimeter

% large circle arc
thetaR2 = acos((R2^2 + d^2 - r2^2) / (2*R2*d));
betaR2 = (2*pi) - (2*thetaR2);
arcR2 = betaR2*R2;
circR2 = 2*pi*R2;

```

```

% small circle arc
alphan2 = asin((R2*sin(thetaR2)/r2));
gamm2 = (2*pi) - (2*alphan2);
arcr2 = gamm2*r2;
circr2 = 2*pi*r2;

Pstarts2 = (numstarts*arcr2);
Pinner2 = circr2 - (numstarts*(circr2-arcr2));
P2 = Pstarts2+Pinner2;
disp('Wetted perimeter (mm) is:')
disp(P2)

%% Pressure Drop

%% Specify flow conditions

density = 997; % kg/m^3
V_tube = 0.38; % volumetric flowrate, m^3/s
A_HX = 0.75^2; % maximum area of HX
mu = 769e-6; % kinematic viscosity, N*s/m^2
length = 3; % HX allowable length
roughness = 0; % relative roughness value

pressure_drop = deltaP(V_tube,density,OD,A_HX,P2,A_internal_in,mu,roughness,length);
disp('Pressure drop (kPa)')
disp(pressure_drop/1000)

function [deltaP] =
deltaP(volumetricflowrate,density,OD,HXarea,Pwet,A_crosssection,mu,epsilon,length)

tubeOD = OD / 1000; % convert to m
N_tubes = round(HXarea / (tubeOD^2))
A_c = A_crosssection / 1e6; % convert area to m
massflow = (volumetricflowrate * density) / N_tubes;
u_m = massflow / (density*A_c); % mean velocity

D_hydraulic = (4*A_c) / (Pwet/1000);

Re_D = (density * u_m * D_hydraulic) / mu;

f = 0.25 / log10((epsilon/3.7*D_hydraulic)+(5.74/(Re_D^0.9)))^2;

deltaP = f * (length/D_hydraulic) * ((density*(u_m^2))/2);

end

```

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX B. CFX COMMAND LINE (CCL) FILES

A. MASS FLOW INLET WITH ZERO PRESSURE OUTLET CONDITION

LIBRARY:

MATERIAL GROUP: Air Data

Group Description = Ideal gas and constant property air. Constant \
properties are for dry air at STP (0 C, 1 atm) and 25 C, 1 atm.

END

MATERIAL GROUP: CHT Solids

Group Description = Pure solid substances that can be used for conjugate \
heat transfer.

END

MATERIAL GROUP: Calorically Perfect Ideal Gases

Group Description = Ideal gases with constant specific heat capacity. \
Specific heat is evaluated at STP.

END

MATERIAL GROUP: Constant Property Gases

Group Description = Gaseous substances with constant properties. \
Properties are calculated at STP (0C and 1 atm). Can be combined with \
NASA SP-273 materials for combustion modelling.

END

MATERIAL GROUP: Constant Property Liquids

Group Description = Liquid substances with constant properties.

END

MATERIAL GROUP: Dry Peng Robinson

Group Description = Materials with properties specified using the built \
in Peng Robinson equation of state. Suitable for dry real gas modelling.

END

MATERIAL GROUP: Dry Redlich Kwong

Group Description = Materials with properties specified using the built \
in Redlich Kwong equation of state. Suitable for dry real gas modelling.

END

MATERIAL GROUP: Dry Soave Redlich Kwong

Group Description = Materials with properties specified using the built \
in Soave Redlich Kwong equation of state. Suitable for dry real gas \
modelling.

END

MATERIAL GROUP: Dry Steam

Group Description = Materials with properties specified using the IAPWS \ equation of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Gas Phase Combustion

Group Description = Ideal gas materials which can be use for gas phase \ combustion. Ideal gas specific heat coefficients are specified using \ the NASA SP-273 format.

END

MATERIAL GROUP: IAPWS IF97

Group Description = Liquid, vapour and binary mixture materials which use \ the IAPWS IF-97 equation of state. Materials are suitable for \ compressible liquids, phase change calculations and dry steam flows.

END

MATERIAL GROUP: Interphase Mass Transfer

Group Description = Materials with reference properties suitable for \ performing either Eulerian or Lagrangian multiphase mass transfer \ problems. Examples include cavitation, evaporation or condensation.

END

MATERIAL GROUP: Liquid Phase Combustion

Group Description = Liquid and homogenous binary mixture materials which \ can be included with Gas Phase Combustion materials if combustion \ modelling also requires phase change (eg: evaporation) for certain \ components.

END

MATERIAL GROUP: Particle Solids

Group Description = Pure solid substances that can be used for particle \ tracking

END

MATERIAL GROUP: Peng Robinson Dry Hydrocarbons

Group Description = Common hydrocarbons which use the Peng Robinson \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Peng Robinson Dry Refrigerants

Group Description = Common refrigerants which use the Peng Robinson \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Peng Robinson Dry Steam

Group Description = Water materials which use the Peng Robinson equation \

of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Peng Robinson Wet Hydrocarbons

Group Description = Common hydrocarbons which use the Peng Robinson \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Peng Robinson Wet Refrigerants

Group Description = Common refrigerants which use the Peng Robinson \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Peng Robinson Wet Steam

Group Description = Water materials which use the Peng Robinson equation \ of state. Suitable for condensing steam modelling.

END

MATERIAL GROUP: Real Gas Combustion

Group Description = Real gas materials which can be use for gas phase \ combustion. Ideal gas specific heat coefficients are specified using \ the NASA SP-273 format.

END

MATERIAL GROUP: Redlich Kwong Dry Hydrocarbons

Group Description = Common hydrocarbons which use the Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Redlich Kwong Dry Refrigerants

Group Description = Common refrigerants which use the Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Redlich Kwong Dry Steam

Group Description = Water materials which use the Redlich Kwong equation \ of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Redlich Kwong Wet Hydrocarbons

Group Description = Common hydrocarbons which use the Redlich Kwong \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Redlich Kwong Wet Refrigerants

Group Description = Common refrigerants which use the Redlich Kwong \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Redlich Kwong Wet Steam

Group Description = Water materials which use the Redlich Kwong equation \ of state. Suitable for condensing steam modelling.

END

MATERIAL GROUP: Soave Redlich Kwong Dry Hydrocarbons

Group Description = Common hydrocarbons which use the Soave Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Dry Refrigerants

Group Description = Common refrigerants which use the Soave Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Dry Steam

Group Description = Water materials which use the Soave Redlich Kwong \ equation of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Soave Redlich Kwong Wet Hydrocarbons

Group Description = Common hydrocarbons which use the Soave Redlich Kwong \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Wet Refrigerants

Group Description = Common refrigerants which use the Soave Redlich Kwong \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Wet Steam

Group Description = Water materials which use the Soave Redlich Kwong \ equation of state. Suitable for condensing steam modelling.

END

MATERIAL GROUP: Soot

Group Description = Solid substances that can be used when performing \ soot modelling

END

MATERIAL GROUP: User

Group Description = Materials that are defined by the user

END

MATERIAL GROUP: Water Data

Group Description = Liquid and vapour water materials with constant \ properties. Can be combined with NASA SP-273 materials for combustion \ modelling.

END

MATERIAL GROUP: Wet Peng Robinson

Group Description = Materials with properties specified using the built \
in Peng Robinson equation of state. Suitable for wet real gas modelling.

END

MATERIAL GROUP: Wet Redlich Kwong

Group Description = Materials with properties specified using the built \
in Redlich Kwong equation of state. Suitable for wet real gas modelling.

END

MATERIAL GROUP: Wet Soave Redlich Kwong

Group Description = Materials with properties specified using the built \
in Soave Redlich Kwong equation of state. Suitable for wet real gas \
modelling.

END

MATERIAL GROUP: Wet Steam

Group Description = Materials with properties specified using the IAPWS \
equation of state. Suitable for wet steam modelling.

END

MATERIAL: Air Ideal Gas

Material Description = Air Ideal Gas (constant Cp)

Material Group = Air Data, Calorically Perfect Ideal Gases

Option = Pure Substance

Thermodynamic State = Gas

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Molar Mass = 28.96 [kg kmol⁻¹]

Option = Ideal Gas

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 1.0044E+03 [J kg⁻¹ K⁻¹]

Specific Heat Type = Constant Pressure

END

REFERENCE STATE:

Option = Specified Point

Reference Pressure = 1 [atm]

Reference Specific Enthalpy = 0. [J/kg]

Reference Specific Entropy = 0. [J/kg/K]

Reference Temperature = 25 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 1.831E-05 [kg m⁻¹ s⁻¹]

Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 2.61E-2 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 0.01 [m⁻¹]

Option = Value

END

SCATTERING COEFFICIENT:

Option = Value

Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value

Refractive Index = 1.0 [m m⁻¹]

END

END

END

MATERIAL: Air at 25 C

Material Description = Air at 25 C and 1 atm (dry)

Material Group = Air Data, Constant Property Gases

Option = Pure Substance

Thermodynamic State = Gas

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 1.185 [kg m⁻³]

Molar Mass = 28.96 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 1.0044E+03 [J kg⁻¹ K⁻¹]

Specific Heat Type = Constant Pressure

END

REFERENCE STATE:

Option = Specified Point

Reference Pressure = 1 [atm]

Reference Specific Enthalpy = 0. [J/kg]

Reference Specific Entropy = 0. [J/kg/K]

Reference Temperature = 25 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 1.831E-05 [kg m⁻¹ s⁻¹]

Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 2.61E-02 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 0.01 [m⁻¹]

Option = Value

END

SCATTERING COEFFICIENT:

Option = Value

Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value

Refractive Index = 1.0 [m m⁻¹]

END

THERMAL EXPANSIVITY:

Option = Value

Thermal Expansivity = 0.003356 [K⁻¹]

END

END

END

MATERIAL: Aluminium

Material Group = CHT Solids, Particle Solids

Option = Pure Substance

Thermodynamic State = Solid

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 2702 [kg m⁻³]

Molar Mass = 26.98 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 9.03E+02 [J kg⁻¹ K⁻¹]

END

REFERENCE STATE:

Option = Specified Point

Reference Specific Enthalpy = 0 [J/kg]

Reference Specific Entropy = 0 [J/kg/K]

Reference Temperature = 25 [C]

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 237 [W m⁻¹ K⁻¹]

END

END

END

MATERIAL: Copper

Material Group = CHT Solids, Particle Solids

Option = Pure Substance

Thermodynamic State = Solid

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 8933 [kg m⁻³]

Molar Mass = 63.55 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 3.85E+02 [J kg⁻¹ K⁻¹]

END

REFERENCE STATE:

Option = Specified Point
 Reference Specific Enthalpy = 0 [J/kg]
 Reference Specific Entropy = 0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 401.0 [W m⁻¹ K⁻¹]
 END
 END
 END
 MATERIAL: Soot
 Material Group = Soot
 Option = Pure Substance
 Thermodynamic State = Solid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 2000 [kg m⁻³]
 Molar Mass = 12 [kg kmol⁻¹]
 Option = Value
 END
 REFERENCE STATE:
 Option = Automatic
 END
 ABSORPTION COEFFICIENT:
 Absorption Coefficient = 0 [m⁻¹]
 Option = Value
 END
 END
 END
 MATERIAL: Steel
 Material Group = CHT Solids, Particle Solids
 Option = Pure Substance
 Thermodynamic State = Solid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 7854 [kg m⁻³]

Molar Mass = 55.85 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 4.34E+02 [J kg⁻¹ K⁻¹]
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Specific Enthalpy = 0 [J/kg]
 Reference Specific Entropy = 0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 60.5 [W m⁻¹ K⁻¹]
 END
 END
 END
 MATERIAL: Water
 Material Description = Water (liquid)
 Material Group = Water Data, Constant Property Liquids
 Option = Pure Substance
 Thermodynamic State = Liquid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 997.0 [kg m⁻³]
 Molar Mass = 18.02 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 4181.7 [J kg⁻¹ K⁻¹]
 Specific Heat Type = Constant Pressure
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Pressure = 1 [atm]

Reference Specific Enthalpy = 0.0 [J/kg]

Reference Specific Entropy = 0.0 [J/kg/K]

Reference Temperature = 25 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 8.899E-4 [kg m⁻¹ s⁻¹]

Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 0.6069 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 1.0 [m⁻¹]

Option = Value

END

SCATTERING COEFFICIENT:

Option = Value

Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value

Refractive Index = 1.0 [m m⁻¹]

END

THERMAL EXPANSIVITY:

Option = Value

Thermal Expansivity = 2.57E-04 [K⁻¹]

END

END

END

MATERIAL: Water Ideal Gas

Material Description = Water Vapour Ideal Gas (100 C and 1 atm)

Material Group = Calorically Perfect Ideal Gases, Water Data

Option = Pure Substance

Thermodynamic State = Gas

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Molar Mass = 18.02 [kg kmol⁻¹]

Option = Ideal Gas

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 2080.1 [J kg⁻¹ K⁻¹]

Specific Heat Type = Constant Pressure

END

REFERENCE STATE:

Option = Specified Point

Reference Pressure = 1.014 [bar]

Reference Specific Enthalpy = 0. [J/kg]

Reference Specific Entropy = 0. [J/kg/K]

Reference Temperature = 100 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 9.4E-06 [kg m⁻¹ s⁻¹]

Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 193E-04 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 1.0 [m⁻¹]

Option = Value

END

SCATTERING COEFFICIENT:

Option = Value

Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value

Refractive Index = 1.0 [m m⁻¹]

END

END

END

END

FLOW: Flow Analysis 1

SOLUTION UNITS:

Angle Units = [rad]
 Length Units = [m]
 Mass Units = [kg]
 Solid Angle Units = [sr]
 Temperature Units = [K]
 Time Units = [s]
 END
 ANALYSIS TYPE:
 Option = Steady State
 EXTERNAL SOLVER COUPLING:
 Option = None
 END
 END
 DOMAIN: Default Domain
 Coord Frame = Coord 0
 Domain Type = Fluid
 Location = B1127, B122, B1322, B1517, B1712, B1907, B347, B542, B737, B932
 BOUNDARY: Default Fluid Fluid Interface Side 1
 Boundary Type = INTERFACE
 Location = \
 F1093.1127,F1110.1127,F126.122,F1288.1322,F1304.1322,F137.122,F313.347\
 ,F508.542,F703.737
 BOUNDARY CONDITIONS:
 MASS AND MOMENTUM:
 Option = Conservative Interface Flux
 END
 TURBULENCE:
 Option = Conservative Interface Flux
 END
 END
 END
 BOUNDARY: Default Fluid Fluid Interface Side 2
 Boundary Type = INTERFACE
 Location = \
 F1483.1517,F1499.1517,F1692.1712,F1873.1907,F332.347,F541.542,F720.737\
 ,F898.932,F931.932
 BOUNDARY CONDITIONS:
 MASS AND MOMENTUM:
 Option = Conservative Interface Flux

```

END
TURBULENCE:
    Option = Conservative Interface Flux
END
END
BOUNDARY: inlet
    Boundary Type = INLET
    Location = inlet
    BOUNDARY CONDITIONS:
        FLOW DIRECTION:
            Option = Normal to Boundary Condition
        END
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Mass Flow Rate = 0.3297 [kg s-1]
            Mass Flow Rate Area = As Specified
            Option = Mass Flow Rate
        END
        TURBULENCE:
            Option = Medium Intensity and Eddy Viscosity Ratio
        END
    END
END
BOUNDARY: outlet
    Boundary Type = OUTLET
    Location = outlet
    BOUNDARY CONDITIONS:
        FLOW REGIME:
            Option = Subsonic
        END
        MASS AND MOMENTUM:
            Option = Average Static Pressure
            Pressure Profile Blend = 0.05
            Relative Pressure = 0 [Pa]
        END
    PRESSURE AVERAGING:

```

```

Option = Average Over Whole Outlet
END
END
END
BOUNDARY: wall
Boundary Type = WALL
Location = \
F1094.1127,F1095.1127,F1096.1127,F1097.1127,F1098.1127,F1099.1127,F110\
0.1127,F1101.1127,F1102.1127,F1103.1127,F1104.1127,F1105.1127,F1106.11\
27,F1107.1127,F1108.1127,F1109.1127,F1111.1127,F1112.1127,F1113.1127,F\
1114.1127,F1115.1127,F1116.1127,F1117.1127,F1118.1127,F1119.1127,F1120\
.1127,F1121.1127,F1122.1127,F1123.1127,F1124.1127,F1125.1127,F1126.112\
7,F118.122,F119.122,F120.122,F121.122,F123.122,F124.122,F125.122,F127.\
122,F128.122,F1289.1322,F129.122,F1290.1322,F1291.1322,F1292.1322,F129\
3.1322,F1294.1322,F1295.1322,F1296.1322,F1297.1322,F1298.1322,F1299.13\
22,F130.122,F1300.1322,F1301.1322,F1302.1322,F1303.1322,F1305.1322,F13\
06.1322,F1307.1322,F1308.1322,F1309.1322,F131.122,F1310.1322,F1311.132\
2,F1312.1322,F1313.1322,F1314.1322,F1315.1322,F1316.1322,F1317.1322,F1\
318.1322,F1319.1322,F132.122,F1320.1322,F1321.1322,F133.122,F134.122,F\
135.122,F136.122,F138.122,F139.122,F140.122,F141.122,F142.122,F143.122\
,F144.122,F145.122,F146.122,F147.122,F148.122,F1484.1517,F1485.1517,F1\
486.1517,F1487.1517,F1488.1517,F1489.1517,F149.122,F1490.1517,F1491.15\
17,F1492.1517,F1493.1517,F1494.1517,F1495.1517,F1496.1517,F1497.1517,F\
1498.1517,F150.122,F1500.1517,F1501.1517,F1502.1517,F1503.1517,F1504.1\
517,F1505.1517,F1506.1517,F1507.1517,F1508.1517,F1509.1517,F151.122,F1\
510.1517,F1511.1517,F1512.1517,F1513.1517,F1514.1517,F1515.1517,F1516.\
1517,F152.122,F1678.1712,F1679.1712,F1680.1712,F1681.1712,F1682.1712,F\
1683.1712,F1684.1712,F1685.1712,F1686.1712,F1687.1712,F1688.1712,F1689\
.1712,F1690.1712,F1691.1712,F1694.1712,F1695.1712,F1696.1712,F1697.171\
2,F1698.1712,F1699.1712,F1700.1712,F1701.1712,F1702.1712,F1703.1712,F1\
704.1712,F1705.1712,F1706.1712,F1707.1712,F1708.1712,F1709.1712,F1710.\
1712,F1711.1712,F1874.1907,F1875.1907,F1876.1907,F1877.1907,F1878.1907\
,F1879.1907,F1880.1907,F1881.1907,F1882.1907,F1883.1907,F1884.1907,F18\
85.1907,F1886.1907,F1887.1907,F1888.1907,F1889.1907,F1890.1907,F1891.1\
907,F1892.1907,F1893.1907,F1894.1907,F1895.1907,F1896.1907,F1897.1907,\
F1898.1907,F1899.1907,F1900.1907,F1901.1907,F1903.1907,F1904.1907,F190\
5.1907,F1906.1907,F314.347,F315.347,F316.347,F317.347,F318.347,F319.34\
7,F320.347,F321.347,F322.347,F323.347,F324.347,F325.347,F326.347,F327.\
347,F328.347,F329.347,F330.347,F331.347,F333.347,F334.347,F335.347,F33\

```

6.347,F337.347,F338.347,F339.347,F340.347,F341.347,F342.347,F343.347,F\344.347,F345.347,F346.347,F509.542,F510.542,F511.542,F512.542,F513.542\F514.542,F515.542,F516.542,F517.542,F518.542,F519.542,F520.542,F521.5\42,F522.542,F523.542,F524.542,F525.542,F526.542,F527.542,F528.542,F529\542,F530.542,F531.542,F532.542,F533.542,F534.542,F535.542,F536.542,F5\37.542,F538.542,F539.542,F540.542,F704.737,F705.737,F706.737,F707.737,\F708.737,F709.737,F710.737,F711.737,F712.737,F713.737,F714.737,F715.73\7,F716.737,F717.737,F718.737,F719.737,F721.737,F722.737,F723.737,F724.\737,F725.737,F726.737,F727.737,F728.737,F729.737,F730.737,F731.737,F73\2.737,F733.737,F734.737,F735.737,F736.737,F899.932,F900.932,F901.932,F\902.932,F903.932,F904.932,F905.932,F906.932,F907.932,F908.932,F909.932\F910.932,F911.932,F912.932,F913.932,F914.932,F915.932,F916.932,F917.9\32,F918.932,F919.932,F920.932,F921.932,F922.932,F923.932,F924.932,F925\932,F926.932,F927.932,F928.932,F929.932,F930.932

BOUNDARY CONDITIONS:

MASS AND MOMENTUM:

Option = No Slip Wall

END

WALL ROUGHNESS:

Option = Smooth Wall

END

END

END

DOMAIN MODELS:

BUOYANCY MODEL:

Option = Non Buoyant

END

DOMAIN MOTION:

Option = Stationary

END

MESH DEFORMATION:

Option = None

END

REFERENCE PRESSURE:

Reference Pressure = 1 [atm]

END

END

FLUID DEFINITION: Fluid 1

Material = Water

```

Option = Material Library
MORPHOLOGY:
    Option = Continuous Fluid
END
END
FLUID MODELS:
    COMBUSTION MODEL:
        Option = None
    END
    HEAT TRANSFER MODEL:
        Fluid Temperature = 25 [C]
        Option = Isothermal
    END
    THERMAL RADIATION MODEL:
        Option = None
    END
    TURBULENCE MODEL:
        Option = SST
    END
    TURBULENT WALL FUNCTIONS:
        Option = Automatic
    END
END
DOMAIN INTERFACE: Default Fluid Fluid Interface
Boundary List1 = Default Fluid Fluid Interface Side 1
Boundary List2 = Default Fluid Fluid Interface Side 2
Interface Type = Fluid Fluid
INTERFACE MODELS:
    Option = General Connection
FRAME CHANGE:
    Option = None
END
MASS AND MOMENTUM:
    Option = Conservative Interface Flux
MOMENTUM INTERFACE MODEL:
    Option = None
END
END

```

```

PITCH CHANGE:
    Option = None
END
END
MESH CONNECTION:
    Option = Automatic
END
END
OUTPUT CONTROL:
RESULTS:
    File Compression Level = Default
    Option = Standard
END
END
SOLVER CONTROL:
    Turbulence Numerics = First Order
ADVECTION SCHEME:
    Option = High Resolution
END
CONVERGENCE CONTROL:
    Length Scale Option = Conservative
    Maximum Number of Iterations = 2000
    Minimum Number of Iterations = 1
    Timescale Control = Auto Timescale
    Timescale Factor = 1.0
END
CONVERGENCE CRITERIA:
    Residual Target = 1e-20
    Residual Type = RMS
END
DYNAMIC MODEL CONTROL:
    Global Dynamic Model Control = On
END
INTERRUPT CONTROL:
    Option = Any Interrupt
CONVERGENCE CONDITIONS:
    Option = Default Conditions
END
END

```

```

END
END
COMMAND FILE:
  Version = 22.1
END
SIMULATION CONTROL:
EXECUTION CONTROL:
  EXECUTABLE SELECTION:
    Double Precision = On
    Large Problem = On
  END
  INTERPOLATOR STEP CONTROL:
    Runtime Priority = Standard
  END
  PARTITIONER STEP CONTROL:
    Multidomain Option = Automatic
    Runtime Priority = Standard
  MEMORY CONTROL:
    Memory Allocation Factor = 10
  END
  PARTITION SMOOTHING:
    Maximum Partition Smoothing Sweeps = 100
    Option = Smooth
  END
  PARTITIONING TYPE:
    MeTiS Type = k-way
    Multipass Partitioning Option = None
    Option = MeTiS
    Partition Size Rule = Automatic
  END
END
RUN DEFINITION:
  Run Mode = Full
  Solver Input File = C:\Users\garth\OneDrive - Naval Postgraduate \
    School\202306_Byrne\004_CFD\2023 Simulations\Week of \
    20FEB\230427-finemeshes_files\dp0\CFX\CFX\Fluid Flow CFX.def
  END
  SOLVER STEP CONTROL:
    Runtime Priority = Standard

```

PARALLEL ENVIRONMENT:

Number of Processes = 128

Parallel Host List = RMKW1-2022-TCR1

Start Method = Intel MPI Local Parallel

END

END

END

END

B. IMPORTED VELOCITY PROFILE WITH ZERO PRESSURE OUTLET CONDITION

LIBRARY:

CEL:

FUNCTION: profile70mil

Argument Units = [m], [m], [m]

Option = Profile Data

Spatial Fields = x, y, z

DATA FIELD: Velocity u

Field Name = Velocity u

Parameter List = U, Velocity r Component, Wall U, Wall Velocity r \

Component

Result Units = [m s⁻¹]

END

DATA FIELD: Velocity v

Field Name = Velocity v

Parameter List = V, Velocity Theta Component, Wall V, Wall Velocity \

Theta Component

Result Units = [m s⁻¹]

END

DATA FIELD: Velocity w

Field Name = Velocity w

Parameter List = Velocity Axial Component, W, Wall Velocity Axial \

Component, Wall W

Result Units = [m s⁻¹]

END

DATA SOURCE:

File Name = C:/Users/garth/OneDrive - Naval Postgraduate \

School/202306_Byrne/004_CFD/2023 Simulations/Week of \

20FEB/VP70mil.csv

Option = From File

END

END

END

MATERIAL GROUP: Air Data

Group Description = Ideal gas and constant property air. Constant \

properties are for dry air at STP (0 C, 1 atm) and 25 C, 1 atm.

END

MATERIAL GROUP: CHT Solids

Group Description = Pure solid substances that can be used for conjugate \ heat transfer.

END

MATERIAL GROUP: Calorically Perfect Ideal Gases

Group Description = Ideal gases with constant specific heat capacity. \ Specific heat is evaluated at STP.

END

MATERIAL GROUP: Constant Property Gases

Group Description = Gaseous substances with constant properties. \ Properties are calculated at STP (0C and 1 atm). Can be combined with \ NASA SP-273 materials for combustion modelling.

END

MATERIAL GROUP: Constant Property Liquids

Group Description = Liquid substances with constant properties.

END

MATERIAL GROUP: Dry Peng Robinson

Group Description = Materials with properties specified using the built \ in Peng Robinson equation of state. Suitable for dry real gas modelling.

END

MATERIAL GROUP: Dry Redlich Kwong

Group Description = Materials with properties specified using the built \ in Redlich Kwong equation of state. Suitable for dry real gas modelling.

END

MATERIAL GROUP: Dry Soave Redlich Kwong

Group Description = Materials with properties specified using the built \ in Soave Redlich Kwong equation of state. Suitable for dry real gas \ modelling.

END

MATERIAL GROUP: Dry Steam

Group Description = Materials with properties specified using the IAPWS \ equation of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Gas Phase Combustion

Group Description = Ideal gas materials which can be use for gas phase \ combustion. Ideal gas specific heat coefficients are specified using \ the NASA SP-273 format.

END

MATERIAL GROUP: IAPWS IF97

Group Description = Liquid, vapour and binary mixture materials which use \ the IAPWS IF-97 equation of state. Materials are suitable for \ compressible liquids, phase change calculations and dry steam flows.

END

MATERIAL GROUP: Interphase Mass Transfer

Group Description = Materials with reference properties suitable for \ performing either Eulerian or Lagrangian multiphase mass transfer \ problems. Examples include cavitation, evaporation or condensation.

END

MATERIAL GROUP: Liquid Phase Combustion

Group Description = Liquid and homogenous binary mixture materials which \ can be included with Gas Phase Combustion materials if combustion \ modelling also requires phase change (eg: evaporation) for certain \ components.

END

MATERIAL GROUP: Particle Solids

Group Description = Pure solid substances that can be used for particle \ tracking

END

MATERIAL GROUP: Peng Robinson Dry Hydrocarbons

Group Description = Common hydrocarbons which use the Peng Robinson \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Peng Robinson Dry Refrigerants

Group Description = Common refrigerants which use the Peng Robinson \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Peng Robinson Dry Steam

Group Description = Water materials which use the Peng Robinson equation \ of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Peng Robinson Wet Hydrocarbons

Group Description = Common hydrocarbons which use the Peng Robinson \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Peng Robinson Wet Refrigerants

Group Description = Common refrigerants which use the Peng Robinson \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Peng Robinson Wet Steam

Group Description = Water materials which use the Peng Robinson equation \ of state. Suitable for condensing steam modelling.

END

MATERIAL GROUP: Real Gas Combustion

Group Description = Real gas materials which can be use for gas phase \ combustion. Ideal gas specific heat coefficients are specified using \ the NASA SP-273 format.

END

MATERIAL GROUP: Redlich Kwong Dry Hydrocarbons

Group Description = Common hydrocarbons which use the Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Redlich Kwong Dry Refrigerants

Group Description = Common refrigerants which use the Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Redlich Kwong Dry Steam

Group Description = Water materials which use the Redlich Kwong equation \ of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Redlich Kwong Wet Hydrocarbons

Group Description = Common hydrocarbons which use the Redlich Kwong \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Redlich Kwong Wet Refrigerants

Group Description = Common refrigerants which use the Redlich Kwong \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Redlich Kwong Wet Steam

Group Description = Water materials which use the Redlich Kwong equation \ of state. Suitable for condensing steam modelling.

END

MATERIAL GROUP: Soave Redlich Kwong Dry Hydrocarbons

Group Description = Common hydrocarbons which use the Soave Redlich Kwong \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Dry Refrigerants

Group Description = Common refrigerants which use the Soave Redlich Kwong \

equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Dry Steam

Group Description = Water materials which use the Soave Redlich Kwong \
equation of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Soave Redlich Kwong Wet Hydrocarbons

Group Description = Common hydrocarbons which use the Soave Redlich Kwong \
equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Wet Refrigerants

Group Description = Common refrigerants which use the Soave Redlich Kwong \
equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Soave Redlich Kwong Wet Steam

Group Description = Water materials which use the Soave Redlich Kwong \
equation of state. Suitable for condensing steam modelling.

END

MATERIAL GROUP: Soot

Group Description = Solid substances that can be used when performing \
soot modelling

END

MATERIAL GROUP: User

Group Description = Materials that are defined by the user

END

MATERIAL GROUP: Water Data

Group Description = Liquid and vapour water materials with constant \
properties. Can be combined with NASA SP-273 materials for combustion \
modelling.

END

MATERIAL GROUP: Wet Peng Robinson

Group Description = Materials with properties specified using the built \
in Peng Robinson equation of state. Suitable for wet real gas modelling.

END

MATERIAL GROUP: Wet Redlich Kwong

Group Description = Materials with properties specified using the built \
in Redlich Kwong equation of state. Suitable for wet real gas modelling.

END

MATERIAL GROUP: Wet Soave Redlich Kwong

Group Description = Materials with properties specified using the built \
in Soave Redlich Kwong equation of state. Suitable for wet real gas \
modelling.

END

MATERIAL GROUP: Wet Steam

Group Description = Materials with properties specified using the IAPWS \
equation of state. Suitable for wet steam modelling.

END

MATERIAL: Air Ideal Gas

Material Description = Air Ideal Gas (constant Cp)

Material Group = Air Data, Calorically Perfect Ideal Gases

Option = Pure Substance

Thermodynamic State = Gas

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Molar Mass = 28.96 [kg kmol⁻¹]

Option = Ideal Gas

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 1.0044E+03 [J kg⁻¹ K⁻¹]

Specific Heat Type = Constant Pressure

END

REFERENCE STATE:

Option = Specified Point

Reference Pressure = 1 [atm]

Reference Specific Enthalpy = 0. [J/kg]

Reference Specific Entropy = 0. [J/kg/K]

Reference Temperature = 25 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 1.831E-05 [kg m⁻¹ s⁻¹]

Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 2.61E-2 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 0.01 [m⁻¹]

Option = Value

END

SCATTERING COEFFICIENT:

Option = Value

Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value

Refractive Index = 1.0 [m m⁻¹]

END

END

END

MATERIAL: Air at 25 C

Material Description = Air at 25 C and 1 atm (dry)

Material Group = Air Data, Constant Property Gases

Option = Pure Substance

Thermodynamic State = Gas

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 1.185 [kg m⁻³]

Molar Mass = 28.96 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 1.0044E+03 [J kg⁻¹ K⁻¹]

Specific Heat Type = Constant Pressure

END

REFERENCE STATE:

Option = Specified Point

Reference Pressure = 1 [atm]

Reference Specific Enthalpy = 0. [J/kg]

Reference Specific Entropy = 0. [J/kg/K]

Reference Temperature = 25 [C]

END

DYNAMIC VISCOSITY:

Dynamic Viscosity = 1.831E-05 [kg m⁻¹ s⁻¹]

Option = Value

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 2.61E-02 [W m⁻¹ K⁻¹]

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 0.01 [m⁻¹]

Option = Value

END

SCATTERING COEFFICIENT:

Option = Value

Scattering Coefficient = 0.0 [m⁻¹]

END

REFRACTIVE INDEX:

Option = Value

Refractive Index = 1.0 [m m⁻¹]

END

THERMAL EXPANSIVITY:

Option = Value

Thermal Expansivity = 0.003356 [K⁻¹]

END

END

END

MATERIAL: Aluminium

Material Group = CHT Solids, Particle Solids

Option = Pure Substance

Thermodynamic State = Solid

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 2702 [kg m⁻³]

Molar Mass = 26.98 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 9.03E+02 [J kg⁻¹ K⁻¹]

END

REFERENCE STATE:

Option = Specified Point

Reference Specific Enthalpy = 0 [J/kg]

Reference Specific Entropy = 0 [J/kg/K]

Reference Temperature = 25 [C]

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 237 [W m⁻¹ K⁻¹]

END

END

END

MATERIAL: Copper

Material Group = CHT Solids, Particle Solids

Option = Pure Substance

Thermodynamic State = Solid

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 8933 [kg m⁻³]

Molar Mass = 63.55 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 3.85E+02 [J kg⁻¹ K⁻¹]

END

REFERENCE STATE:

Option = Specified Point

Reference Specific Enthalpy = 0 [J/kg]

Reference Specific Entropy = 0 [J/kg/K]

Reference Temperature = 25 [C]

END

THERMAL CONDUCTIVITY:

Option = Value

Thermal Conductivity = 401.0 [W m⁻¹ K⁻¹]

END

END

END

MATERIAL: Soot

Material Group = Soot

Option = Pure Substance

Thermodynamic State = Solid

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 2000 [kg m⁻³]

Molar Mass = 12 [kg kmol⁻¹]

Option = Value

END

REFERENCE STATE:

Option = Automatic

END

ABSORPTION COEFFICIENT:

Absorption Coefficient = 0 [m⁻¹]

Option = Value

END

END

END

MATERIAL: Steel

Material Group = CHT Solids, Particle Solids

Option = Pure Substance

Thermodynamic State = Solid

PROPERTIES:

Option = General Material

EQUATION OF STATE:

Density = 7854 [kg m⁻³]

Molar Mass = 55.85 [kg kmol⁻¹]

Option = Value

END

SPECIFIC HEAT CAPACITY:

Option = Value

Specific Heat Capacity = 4.34E+02 [J kg⁻¹ K⁻¹]

END

REFERENCE STATE:

Option = Specified Point

Reference Specific Enthalpy = 0 [J/kg]

Reference Specific Entropy = 0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 60.5 [W m⁻¹ K⁻¹]
 END
 END
 END
 MATERIAL: Water
 Material Description = Water (liquid)
 Material Group = Water Data, Constant Property Liquids
 Option = Pure Substance
 Thermodynamic State = Liquid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 997.0 [kg m⁻³]
 Molar Mass = 18.02 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 4181.7 [J kg⁻¹ K⁻¹]
 Specific Heat Type = Constant Pressure
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Pressure = 1 [atm]
 Reference Specific Enthalpy = 0.0 [J/kg]
 Reference Specific Entropy = 0.0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 DYNAMIC VISCOSITY:
 Dynamic Viscosity = 8.899E-4 [kg m⁻¹ s⁻¹]
 Option = Value
 END
 THERMAL CONDUCTIVITY:
 Option = Value

Thermal Conductivity = 0.6069 [W m⁻¹ K⁻¹]
 END
 ABSORPTION COEFFICIENT:
 Absorption Coefficient = 1.0 [m⁻¹]
 Option = Value
 END
 SCATTERING COEFFICIENT:
 Option = Value
 Scattering Coefficient = 0.0 [m⁻¹]
 END
 REFRACTIVE INDEX:
 Option = Value
 Refractive Index = 1.0 [m m⁻¹]
 END
 THERMAL EXPANSIVITY:
 Option = Value
 Thermal Expansivity = 2.57E-04 [K⁻¹]
 END
 END
 END
 MATERIAL: Water Ideal Gas
 Material Description = Water Vapour Ideal Gas (100 C and 1 atm)
 Material Group = Calorically Perfect Ideal Gases, Water Data
 Option = Pure Substance
 Thermodynamic State = Gas
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Molar Mass = 18.02 [kg kmol⁻¹]
 Option = Ideal Gas
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 2080.1 [J kg⁻¹ K⁻¹]
 Specific Heat Type = Constant Pressure
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Pressure = 1.014 [bar]

Reference Specific Enthalpy = 0. [J/kg]
 Reference Specific Entropy = 0. [J/kg/K]
 Reference Temperature = 100 [C]
 END
 DYNAMIC VISCOSITY:
 Dynamic Viscosity = 9.4E-06 [kg m⁻¹ s⁻¹]
 Option = Value
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 193E-04 [W m⁻¹ K⁻¹]
 END
 ABSORPTION COEFFICIENT:
 Absorption Coefficient = 1.0 [m⁻¹]
 Option = Value
 END
 SCATTERING COEFFICIENT:
 Option = Value
 Scattering Coefficient = 0.0 [m⁻¹]
 END
 REFRACTIVE INDEX:
 Option = Value
 Refractive Index = 1.0 [m m⁻¹]
 END
 END
 END
 END
 FLOW: Flow Analysis 1
 SOLUTION UNITS:
 Angle Units = [rad]
 Length Units = [m]
 Mass Units = [kg]
 Solid Angle Units = [sr]
 Temperature Units = [K]
 Time Units = [s]
 END
 ANALYSIS TYPE:
 Option = Steady State
 EXTERNAL SOLVER COUPLING:

```

Option = None
END
END
DOMAIN: Default Domain
Coord Frame = Coord 0
Domain Type = Fluid
Location = B1127, B122, B1322, B1517, B1712, B1907, B347, B542, B737, B932
BOUNDARY: Default Fluid Fluid Interface Side 1
Boundary Type = INTERFACE
Location = \
F1093.1127,F1110.1127,F126.122,F1288.1322,F1304.1322,F137.122,F313.347\
,F508.542,F703.737
BOUNDARY CONDITIONS:
MASS AND MOMENTUM:
Option = Conservative Interface Flux
END
TURBULENCE:
Option = Conservative Interface Flux
END
END
END
BOUNDARY: Default Fluid Fluid Interface Side 2
Boundary Type = INTERFACE
Location = \
F1483.1517,F1499.1517,F1692.1712,F1873.1907,F332.347,F541.542,F720.737\
,F898.932,F931.932
BOUNDARY CONDITIONS:
MASS AND MOMENTUM:
Option = Conservative Interface Flux
END
TURBULENCE:
Option = Conservative Interface Flux
END
END
END
BOUNDARY: inlet
Boundary Type = INLET
Location = inlet
Use Profile Data = On

```

```

BOUNDARY CONDITIONS:
  FLOW REGIME:
    Option = Subsonic
  END
  MASS AND MOMENTUM:
    Option = Cartesian Velocity Components
    U = profile70mil.Velocity u(x,y,z)
    V = profile70mil.Velocity v(x,y,z)
    W = profile70mil.Velocity w(x,y,z)
  END
  TURBULENCE:
    Option = Medium Intensity and Eddy Viscosity Ratio
  END
END
BOUNDARY PROFILE:
  Profile Name = profile70mil
END
END
BOUNDARY: outlet
  Boundary Type = OUTLET
  Location = outlet
  Use Profile Data = False
BOUNDARY CONDITIONS:
  FLOW REGIME:
    Option = Subsonic
  END
  MASS AND MOMENTUM:
    Option = Average Static Pressure
    Pressure Profile Blend = 0.05
    Relative Pressure = 0 [Pa]
  END
  PRESSURE AVERAGING:
    Option = Average Over Whole Outlet
  END
END
BOUNDARY: wall
  Boundary Type = WALL
  Location = \

```

F1094.1127,F1095.1127,F1096.1127,F1097.1127,F1098.1127,F1099.1127,F110\
 0.1127,F1101.1127,F1102.1127,F1103.1127,F1104.1127,F1105.1127,F1106.11\
 27,F1107.1127,F1108.1127,F1109.1127,F1111.1127,F1112.1127,F1113.1127,F\
 1114.1127,F1115.1127,F1116.1127,F1117.1127,F1118.1127,F1119.1127,F1120\
 .1127,F1121.1127,F1122.1127,F1123.1127,F1124.1127,F1125.1127,F1126.112\
 7,F118.122,F119.122,F120.122,F121.122,F123.122,F124.122,F125.122,F127.\
 122,F128.122,F1289.1322,F129.122,F1290.1322,F1291.1322,F1292.1322,F129\
 3.1322,F1294.1322,F1295.1322,F1296.1322,F1297.1322,F1298.1322,F1299.13\
 22,F130.122,F1300.1322,F1301.1322,F1302.1322,F1303.1322,F1305.1322,F13\
 06.1322,F1307.1322,F1308.1322,F1309.1322,F131.122,F1310.1322,F1311.132\
 2,F1312.1322,F1313.1322,F1314.1322,F1315.1322,F1316.1322,F1317.1322,F1\
 318.1322,F1319.1322,F132.122,F1320.1322,F1321.1322,F133.122,F134.122,F\
 135.122,F136.122,F138.122,F139.122,F140.122,F141.122,F142.122,F143.122\
 ,F144.122,F145.122,F146.122,F147.122,F148.122,F1484.1517,F1485.1517,F1\
 486.1517,F1487.1517,F1488.1517,F1489.1517,F149.122,F1490.1517,F1491.15\
 17,F1492.1517,F1493.1517,F1494.1517,F1495.1517,F1496.1517,F1497.1517,F\
 1498.1517,F150.122,F1500.1517,F1501.1517,F1502.1517,F1503.1517,F1504.1\
 517,F1505.1517,F1506.1517,F1507.1517,F1508.1517,F1509.1517,F151.122,F1\
 510.1517,F1511.1517,F1512.1517,F1513.1517,F1514.1517,F1515.1517,F1516.\
 1517,F152.122,F1678.1712,F1679.1712,F1680.1712,F1681.1712,F1682.1712,F\
 1683.1712,F1684.1712,F1685.1712,F1686.1712,F1687.1712,F1688.1712,F1689\
 .1712,F1690.1712,F1691.1712,F1694.1712,F1695.1712,F1696.1712,F1697.171\
 2,F1698.1712,F1699.1712,F1700.1712,F1701.1712,F1702.1712,F1703.1712,F1\
 704.1712,F1705.1712,F1706.1712,F1707.1712,F1708.1712,F1709.1712,F1710.\
 1712,F1711.1712,F1874.1907,F1875.1907,F1876.1907,F1877.1907,F1878.1907\
 ,F1879.1907,F1880.1907,F1881.1907,F1882.1907,F1883.1907,F1884.1907,F18\
 85.1907,F1886.1907,F1887.1907,F1888.1907,F1889.1907,F1890.1907,F1891.1\
 907,F1892.1907,F1893.1907,F1894.1907,F1895.1907,F1896.1907,F1897.1907,\
 F1898.1907,F1899.1907,F1900.1907,F1901.1907,F1903.1907,F1904.1907,F190\
 5.1907,F1906.1907,F314.347,F315.347,F316.347,F317.347,F318.347,F319.34\
 7,F320.347,F321.347,F322.347,F323.347,F324.347,F325.347,F326.347,F327.\
 347,F328.347,F329.347,F330.347,F331.347,F333.347,F334.347,F335.347,F33\
 6.347,F337.347,F338.347,F339.347,F340.347,F341.347,F342.347,F343.347,F\
 344.347,F345.347,F346.347,F509.542,F510.542,F511.542,F512.542,F513.542\
 ,F514.542,F515.542,F516.542,F517.542,F518.542,F519.542,F520.542,F521.5\
 42,F522.542,F523.542,F524.542,F525.542,F526.542,F527.542,F528.542,F529\
 .542,F530.542,F531.542,F532.542,F533.542,F534.542,F535.542,F536.542,F5\
 37.542,F538.542,F539.542,F540.542,F704.737,F705.737,F706.737,F707.737,\
 F708.737,F709.737,F710.737,F711.737,F712.737,F713.737,F714.737,F715.73\

7,F716.737,F717.737,F718.737,F719.737,F721.737,F722.737,F723.737,F724.\
737,F725.737,F726.737,F727.737,F728.737,F729.737,F730.737,F731.737,F73\
2.737,F733.737,F734.737,F735.737,F736.737,F899.932,F900.932,F901.932,F\
902.932,F903.932,F904.932,F905.932,F906.932,F907.932,F908.932,F909.932\
,F910.932,F911.932,F912.932,F913.932,F914.932,F915.932,F916.932,F917.9\
32,F918.932,F919.932,F920.932,F921.932,F922.932,F923.932,F924.932,F925\
.932,F926.932,F927.932,F928.932,F929.932,F930.932

Use Profile Data = False

BOUNDARY CONDITIONS:

MASS AND MOMENTUM:

Option = No Slip Wall

END

WALL ROUGHNESS:

Option = Smooth Wall

END

END

END

DOMAIN MODELS:

BUOYANCY MODEL:

Option = Non Buoyant

END

DOMAIN MOTION:

Option = Stationary

END

MESH DEFORMATION:

Option = None

END

REFERENCE PRESSURE:

Reference Pressure = 1 [atm]

END

END

FLUID DEFINITION: Fluid 1

Material = Water

Option = Material Library

MORPHOLOGY:

Option = Continuous Fluid

END

END

FLUID MODELS:

```

COMBUSTION MODEL:
    Option = None
END
HEAT TRANSFER MODEL:
    Fluid Temperature = 25 [C]
    Option = Isothermal
END
THERMAL RADIATION MODEL:
    Option = None
END
TURBULENCE MODEL:
    Option = SST
END
TURBULENT WALL FUNCTIONS:
    Option = Automatic
END
END
DOMAIN INTERFACE: Default Fluid Fluid Interface
    Boundary List1 = Default Fluid Fluid Interface Side 1
    Boundary List2 = Default Fluid Fluid Interface Side 2
    Interface Type = Fluid Fluid
INTERFACE MODELS:
    Option = General Connection
FRAME CHANGE:
    Option = None
END
MASS AND MOMENTUM:
    Option = Conservative Interface Flux
MOMENTUM INTERFACE MODEL:
    Option = None
END
END
PITCH CHANGE:
    Option = None
END
END
MESH CONNECTION:
    Option = Automatic

```

END
END
OUTPUT CONTROL:
RESULTS:
File Compression Level = Default
Option = Standard
END
END
SOLVER CONTROL:
Turbulence Numerics = First Order
ADVECTION SCHEME:
Option = High Resolution
END
CONVERGENCE CONTROL:
Length Scale Option = Conservative
Maximum Number of Iterations = 1500
Minimum Number of Iterations = 1
Timescale Control = Auto Timescale
Timescale Factor = 1.0
END
CONVERGENCE CRITERIA:
Residual Target = 1e-20
Residual Type = RMS
END
DYNAMIC MODEL CONTROL:
Global Dynamic Model Control = On
END
INTERRUPT CONTROL:
Option = Any Interrupt
CONVERGENCE CONDITIONS:
Option = Default Conditions
END
END
END
END
COMMAND FILE:
Version = 22.1
END
SIMULATION CONTROL:

```

EXECUTION CONTROL:
EXECUTABLE SELECTION:
    Double Precision = On
    Large Problem = On
END
INTERPOLATOR STEP CONTROL:
    Runtime Priority = Standard
END
PARTITIONER STEP CONTROL:
    Multidomain Option = Automatic
    Runtime Priority = Standard
MEMORY CONTROL:
    Memory Allocation Factor = 3
END
PARTITION SMOOTHING:
    Maximum Partition Smoothing Sweeps = 100
    Option = Smooth
END
PARTITIONING TYPE:
    MeTiS Type = k-way
    Multipass Partitioning Option = None
    Option = MeTiS
    Partition Size Rule = Automatic
END
END
RUN DEFINITION:
    Run Mode = Full
    Solver Input File = C:\Users\garth\OneDrive - Naval Postgraduate \
        School\202306_Byrne\004_CFD\2023 Simulations\Week of \
        20FEB\230427-finemeshes_files\dp0\CFX-2\CFX/70mil-importedV.def
END
SOLVER STEP CONTROL:
    Runtime Priority = Standard
PARALLEL ENVIRONMENT:
    Number of Processes = 128
    Parallel Host List = RMKW1-2022-TCR1
    Start Method = Intel MPI Local Parallel
END
END

```

END
END

C. HEAT TRANSFER SCENARIO

LIBRARY:

CEL:

EXPRESSIONS:

```
T hot out = massFlowAve(Temperature )@hot_out
shellbal = massFlow()@hot_in*maxVal(Specific Heat Capacity at Constant \
Volume)@hot_in* \
(massFlowAve(Temperature)@hot_in-massFlowAve(Temperature)@hot_out)
tubebal = massFlow()@cold_in*minVal(Specific Heat Capacity at Constant \
Volume)@cold_in* \
(areaAve(Temperature)@cold_out-areaAve(Temperature)@cold_in)
```

END

FUNCTION: profileTTFD

Argument Units = [m], [m], [m]

Option = Profile Data

Spatial Fields = x, y, z

DATA FIELD: Velocity u

Field Name = Velocity u

Parameter List = U, Velocity r Component, Wall U, Wall Velocity r \
Component

Result Units = [m s⁻¹]

END

DATA FIELD: Velocity v

Field Name = Velocity v

Parameter List = V, Velocity Theta Component, Wall V, Wall Velocity \
Theta Component

Result Units = [m s⁻¹]

END

DATA FIELD: Velocity w

Field Name = Velocity w

Parameter List = Velocity Axial Component, W, Wall Velocity Axial \
Component, Wall W

Result Units = [m s⁻¹]

END

DATA SOURCE:

File Name = ./TTFD.csv

Option = From File

END

END

END

MATERIAL GROUP: Air Data

Group Description = Ideal gas and constant property air. Constant \
properties are for dry air at STP (0 C, 1 atm) and 25 C, 1 atm.

END

MATERIAL GROUP: CHT Solids

Group Description = Pure solid substances that can be used for conjugate \
heat transfer.

END

MATERIAL GROUP: Calorically Perfect Ideal Gases

Group Description = Ideal gases with constant specific heat capacity. \
Specific heat is evaluated at STP.

END

MATERIAL GROUP: Constant Property Gases

Group Description = Gaseous substances with constant properties. \
Properties are calculated at STP (0C and 1 atm). Can be combined with \
NASA SP-273 materials for combustion modelling.

END

MATERIAL GROUP: Constant Property Liquids

Group Description = Liquid substances with constant properties.

END

MATERIAL GROUP: Dry Peng Robinson
 Group Description = Materials with properties specified using the built \ in Peng Robinson equation of state. Suitable for dry real gas modelling.

END

MATERIAL GROUP: Dry Redlich Kwong
 Group Description = Materials with properties specified using the built \ in Redlich Kwong equation of state. Suitable for dry real gas modelling.

END

MATERIAL GROUP: Dry Soave Redlich Kwong
 Group Description = Materials with properties specified using the built \ in Soave Redlich Kwong equation of state. Suitable for dry real gas \ modelling.

END

MATERIAL GROUP: Dry Steam
 Group Description = Materials with properties specified using the IAPWS \ equation of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Gas Phase Combustion
 Group Description = Ideal gas materials which can be use for gas phase \ combustion. Ideal gas specific heat coefficients are specified using \ the NASA SP-273 format.

END

MATERIAL GROUP: IAPWS IF97
 Group Description = Liquid, vapour and binary mixture materials which use \ the IAPWS IF-97 equation of state. Materials are suitable for \ compressible liquids, phase change calculations and dry steam flows.

END

MATERIAL GROUP: Interphase Mass Transfer
 Group Description = Materials with reference properties suitable for \ performing either Eulerian or Lagrangian multiphase mass transfer \ problems. Examples include cavitation, evaporation or condensation.

END

MATERIAL GROUP: Liquid Phase Combustion
 Group Description = Liquid and homogenous binary mixture materials which \ can be included with Gas Phase Combustion materials if combustion \ modelling also requires phase change (eg: evaporation) for certain \ components.

END

MATERIAL GROUP: Particle Solids
 Group Description = Pure solid substances that can be used for particle \ tracking

END

MATERIAL GROUP: Peng Robinson Dry Hydrocarbons
 Group Description = Common hydrocarbons which use the Peng Robinson \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Peng Robinson Dry Refrigerants
 Group Description = Common refrigerants which use the Peng Robinson \ equation of state. Suitable for dry real gas models.

END

MATERIAL GROUP: Peng Robinson Dry Steam
 Group Description = Water materials which use the Peng Robinson equation \ of state. Suitable for dry steam modelling.

END

MATERIAL GROUP: Peng Robinson Wet Hydrocarbons
 Group Description = Common hydrocarbons which use the Peng Robinson \ equation of state. Suitable for condensing real gas models.

END

MATERIAL GROUP: Peng Robinson Wet Refrigerants
 Group Description = Common refrigerants which use the Peng Robinson \

equation of state. Suitable for condensing real gas models.
 END
 MATERIAL GROUP: Peng Robinson Wet Steam
 Group Description = Water materials which use the Peng Robinson equation \ of state. Suitable for condensing steam modelling.
 END
 MATERIAL GROUP: Real Gas Combustion
 Group Description = Real gas materials which can be use for gas phase \ combustion. Ideal gas specific heat coefficients are specified using \ the NASA SP-273 format.
 END
 MATERIAL GROUP: Redlich Kwong Dry Hydrocarbons
 Group Description = Common hydrocarbons which use the Redlich Kwong \ equation of state. Suitable for dry real gas models.
 END
 MATERIAL GROUP: Redlich Kwong Dry Refrigerants
 Group Description = Common refrigerants which use the Redlich Kwong \ equation of state. Suitable for dry real gas models.
 END
 MATERIAL GROUP: Redlich Kwong Dry Steam
 Group Description = Water materials which use the Redlich Kwong equation \ of state. Suitable for dry steam modelling.
 END
 MATERIAL GROUP: Redlich Kwong Wet Hydrocarbons
 Group Description = Common hydrocarbons which use the Redlich Kwong \ equation of state. Suitable for condensing real gas models.
 END
 MATERIAL GROUP: Redlich Kwong Wet Refrigerants
 Group Description = Common refrigerants which use the Redlich Kwong \ equation of state. Suitable for condensing real gas models.
 END
 MATERIAL GROUP: Redlich Kwong Wet Steam
 Group Description = Water materials which use the Redlich Kwong equation \ of state. Suitable for condensing steam modelling.
 END
 MATERIAL GROUP: Soave Redlich Kwong Dry Hydrocarbons
 Group Description = Common hydrocarbons which use the Soave Redlich Kwong \ equation of state. Suitable for dry real gas models.
 END
 MATERIAL GROUP: Soave Redlich Kwong Dry Refrigerants
 Group Description = Common refrigerants which use the Soave Redlich Kwong \ equation of state. Suitable for dry real gas models.
 END
 MATERIAL GROUP: Soave Redlich Kwong Dry Steam
 Group Description = Water materials which use the Soave Redlich Kwong \ equation of state. Suitable for dry steam modelling.
 END
 MATERIAL GROUP: Soave Redlich Kwong Wet Hydrocarbons
 Group Description = Common hydrocarbons which use the Soave Redlich Kwong \ equation of state. Suitable for condensing real gas models.
 END
 MATERIAL GROUP: Soave Redlich Kwong Wet Refrigerants
 Group Description = Common refrigerants which use the Soave Redlich Kwong \ equation of state. Suitable for condensing real gas models.
 END
 MATERIAL GROUP: Soave Redlich Kwong Wet Steam
 Group Description = Water materials which use the Soave Redlich Kwong \ equation of state. Suitable for condensing steam modelling.
 END
 MATERIAL GROUP: Soot
 Group Description = Solid substances that can be used when performing \

```

    soot modelling
END
MATERIAL GROUP: User
    Group Description = Materials that are defined by the user
END
MATERIAL GROUP: Water Data
    Group Description = Liquid and vapour water materials with constant \
    properties. Can be combined with NASA SP-273 materials for combustion \
    modelling.
END
MATERIAL GROUP: Wet Peng Robinson
    Group Description = Materials with properties specified using the built \
    in Peng Robinson equation of state. Suitable for wet real gas modelling.
END
MATERIAL GROUP: Wet Redlich Kwong
    Group Description = Materials with properties specified using the built \
    in Redlich Kwong equation of state. Suitable for wet real gas modelling.
END
MATERIAL GROUP: Wet Soave Redlich Kwong
    Group Description = Materials with properties specified using the built \
    in Soave Redlich Kwong equation of state. Suitable for wet real gas \
    modelling.
END
MATERIAL GROUP: Wet Steam
    Group Description = Materials with properties specified using the IAPWS \
    equation of state. Suitable for wet steam modelling.
END
MATERIAL: Air Ideal Gas
    Material Description = Air Ideal Gas (constant Cp)
    Material Group = Air Data, Calorically Perfect Ideal Gases
    Option = Pure Substance
    Thermodynamic State = Gas
    PROPERTIES:
        Option = General Material
    EQUATION OF STATE:
        Molar Mass = 28.96 [kg kmol-1]
        Option = Ideal Gas
    END
    SPECIFIC HEAT CAPACITY:
        Option = Value
        Specific Heat Capacity = 1.0044E+03 [J kg-1 K-1]
        Specific Heat Type = Constant Pressure
    END
    REFERENCE STATE:
        Option = Specified Point
        Reference Pressure = 1 [atm]
        Reference Specific Enthalpy = 0. [J/kg]
        Reference Specific Entropy = 0. [J/kg/K]
        Reference Temperature = 25 [C]
    END
    DYNAMIC VISCOSITY:
        Dynamic Viscosity = 1.831E-05 [kg m-1 s-1]
        Option = Value
    END
    THERMAL CONDUCTIVITY:
        Option = Value
        Thermal Conductivity = 2.61E-2 [W m-1 K-1]
    END
    ABSORPTION COEFFICIENT:
        Absorption Coefficient = 0.01 [m-1]
        Option = Value

```

END
 SCATTERING COEFFICIENT:
 Option = Value
 Scattering Coefficient = 0.0 [m⁻¹]
 END
 REFRACTIVE INDEX:
 Option = Value
 Refractive Index = 1.0 [m m⁻¹]
 END
 END
 MATERIAL: Air at 25 C
 Material Description = Air at 25 C and 1 atm (dry)
 Material Group = Air Data, Constant Property Gases
 Option = Pure Substance
 Thermodynamic State = Gas
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 1.185 [kg m⁻³]
 Molar Mass = 28.96 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 1.0044E+03 [J kg⁻¹ K⁻¹]
 Specific Heat Type = Constant Pressure
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Pressure = 1 [atm]
 Reference Specific Enthalpy = 0. [J/kg]
 Reference Specific Entropy = 0. [J/kg/K]
 Reference Temperature = 25 [C]
 END
 DYNAMIC VISCOSITY:
 Dynamic Viscosity = 1.831E-05 [kg m⁻¹ s⁻¹]
 Option = Value
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 2.61E-02 [W m⁻¹ K⁻¹]
 END
 ABSORPTION COEFFICIENT:
 Absorption Coefficient = 0.01 [m⁻¹]
 Option = Value
 END
 SCATTERING COEFFICIENT:
 Option = Value
 Scattering Coefficient = 0.0 [m⁻¹]
 END
 REFRACTIVE INDEX:
 Option = Value
 Refractive Index = 1.0 [m m⁻¹]
 END
 THERMAL EXPANSIVITY:
 Option = Value
 Thermal Expansivity = 0.003356 [K⁻¹]
 END
 END
 END

MATERIAL: Aluminium
 Material Group = CHT Solids, Particle Solids
 Option = Pure Substance
 Thermodynamic State = Solid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 2702 [kg m⁻³]
 Molar Mass = 26.98 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 9.03E+02 [J kg⁻¹ K⁻¹]
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Specific Enthalpy = 0 [J/kg]
 Reference Specific Entropy = 0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 237 [W m⁻¹ K⁻¹]
 END
 END
 MATERIAL: Copper
 Material Group = CHT Solids, Particle Solids
 Option = Pure Substance
 Thermodynamic State = Solid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 8933 [kg m⁻³]
 Molar Mass = 63.55 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 3.85E+02 [J kg⁻¹ K⁻¹]
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Specific Enthalpy = 0 [J/kg]
 Reference Specific Entropy = 0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 401.0 [W m⁻¹ K⁻¹]
 END
 END
 MATERIAL: Soot
 Material Group = Soot
 Option = Pure Substance
 Thermodynamic State = Solid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:

Density = 2000 [kg m⁻³]
 Molar Mass = 12 [kg kmol⁻¹]
 Option = Value
 END
 REFERENCE STATE:
 Option = Automatic
 END
 ABSORPTION COEFFICIENT:
 Absorption Coefficient = 0 [m⁻¹]
 Option = Value
 END
 END
 MATERIAL: Steel
 Material Group = CHT Solids, Particle Solids
 Option = Pure Substance
 Thermodynamic State = Solid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 7854 [kg m⁻³]
 Molar Mass = 55.85 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 4.34E+02 [J kg⁻¹ K⁻¹]
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Specific Enthalpy = 0 [J/kg]
 Reference Specific Entropy = 0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 60.5 [W m⁻¹ K⁻¹]
 END
 END
 MATERIAL: Water
 Material Description = Water (liquid)
 Material Group = Water Data, Constant Property Liquids
 Option = Pure Substance
 Thermodynamic State = Liquid
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Density = 997.0 [kg m⁻³]
 Molar Mass = 18.02 [kg kmol⁻¹]
 Option = Value
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 4181.7 [J kg⁻¹ K⁻¹]
 Specific Heat Type = Constant Pressure
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Pressure = 1 [atm]
 Reference Specific Enthalpy = 0.0 [J/kg]

Reference Specific Entropy = 0.0 [J/kg/K]
 Reference Temperature = 25 [C]
 END
 DYNAMIC VISCOSITY:
 Dynamic Viscosity = 8.899E-4 [kg m⁻¹ s⁻¹]
 Option = Value
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 0.6069 [W m⁻¹ K⁻¹]
 END
 ABSORPTION COEFFICIENT:
 Absorption Coefficient = 1.0 [m⁻¹]
 Option = Value
 END
 SCATTERING COEFFICIENT:
 Option = Value
 Scattering Coefficient = 0.0 [m⁻¹]
 END
 REFRACTIVE INDEX:
 Option = Value
 Refractive Index = 1.0 [m m⁻¹]
 END
 THERMAL EXPANSIVITY:
 Option = Value
 Thermal Expansivity = 2.57E-04 [K⁻¹]
 END
 END
 END
 MATERIAL: Water Ideal Gas
 Material Description = Water Vapour Ideal Gas (100 C and 1 atm)
 Material Group = Calorically Perfect Ideal Gases, Water Data
 Option = Pure Substance
 Thermodynamic State = Gas
 PROPERTIES:
 Option = General Material
 EQUATION OF STATE:
 Molar Mass = 18.02 [kg kmol⁻¹]
 Option = Ideal Gas
 END
 SPECIFIC HEAT CAPACITY:
 Option = Value
 Specific Heat Capacity = 2080.1 [J kg⁻¹ K⁻¹]
 Specific Heat Type = Constant Pressure
 END
 REFERENCE STATE:
 Option = Specified Point
 Reference Pressure = 1.014 [bar]
 Reference Specific Enthalpy = 0. [J/kg]
 Reference Specific Entropy = 0. [J/kg/K]
 Reference Temperature = 100 [C]
 END
 DYNAMIC VISCOSITY:
 Dynamic Viscosity = 9.4E-06 [kg m⁻¹ s⁻¹]
 Option = Value
 END
 THERMAL CONDUCTIVITY:
 Option = Value
 Thermal Conductivity = 193E-04 [W m⁻¹ K⁻¹]
 END
 ABSORPTION COEFFICIENT:

```

Absorption Coefficient = 1.0 [m^-1]
Option = Value
END
SCATTERING COEFFICIENT:
Option = Value
Scattering Coefficient = 0.0 [m^-1]
END
REFRACTIVE INDEX:
Option = Value
Refractive Index = 1.0 [m m^-1]
END
END
END
MATERIAL: seawater
Material Group = Constant Property Liquids
Option = Pure Substance
Thermodynamic State = Liquid
PROPERTIES:
Option = General Material
EQUATION OF STATE:
Density = 1021 [kg m^-3]
Molar Mass = 18 [kg kmol^-1]
Option = Value
END
SPECIFIC HEAT CAPACITY:
Option = Value
Specific Heat Capacity = 4005.7 [J kg^-1 K^-1]
Specific Heat Type = Constant Pressure
END
DYNAMIC VISCOSITY:
Dynamic Viscosity = 0.00082744 [Pa s]
Option = Value
END
THERMAL CONDUCTIVITY:
Option = Value
Thermal Conductivity = 0.6179 [W m^-1 K^-1]
END
END
END
MATERIAL: ti64
Material Group = CHT Solids
Option = Pure Substance
Thermodynamic State = Solid
PROPERTIES:
Option = General Material
EQUATION OF STATE:
Density = 4410 [kg m^-3]
Molar Mass = 46.74 [kg kmol^-1]
Option = Value
END
SPECIFIC HEAT CAPACITY:
Option = Value
Specific Heat Capacity = 523 [J kg^-1 K^-1]
END
THERMAL CONDUCTIVITY:
Option = Value
Thermal Conductivity = 5.5 [W m^-1 K^-1]
END
END
END
END

```

```

FLOW: Flow Analysis 1
SOLUTION UNITS:
  Angle Units = [rad]
  Length Units = [m]
  Mass Units = [kg]
  Solid Angle Units = [sr]
  Temperature Units = [K]
  Time Units = [s]
END
ANALYSIS TYPE:
  Option = Steady State
EXTERNAL SOLVER COUPLING:
  Option = None
END
END
DOMAIN: shell
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = B120
BOUNDARY: Default Fluid Fluid Interface Side 1
  Boundary Type = INTERFACE
  Location = \
    F121.120,F122.120,F123.120,F124.120,F125.120,F126.120,F127.120,F128.12\
    0,F129.120,F130.120,F131.120,F132.120,F133.120,F134.120,F135.120,F136.\
    120,F137.120,F138.120,F139.120,F140.120,F141.120,F142.120,F143.120,F14\
    4.120,F145.120,F146.120,F147.120,F148.120,F149.120,F150.120,F151.120,F\
    152.120
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
END
BOUNDARY: Domain Interface 1 Side 1
  Boundary Type = INTERFACE
  Location = sym11
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:
    Option = Conservative Interface Flux
  END
END
END
BOUNDARY: Domain Interface 1 Side 2
  Boundary Type = INTERFACE
  Location = sym12
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
END

```

```

TURBULENCE:
  Option = Conservative Interface Flux
END
END
BOUNDARY: Domain Interface 2 Side 1
Boundary Type = INTERFACE
Location = sym21
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:
    Option = Conservative Interface Flux
  END
END
BOUNDARY: Domain Interface 2 Side 2
Boundary Type = INTERFACE
Location = sym22
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:
    Option = Conservative Interface Flux
  END
END
BOUNDARY: Domain Interface 3 Side 1
Boundary Type = INTERFACE
Location = sym31
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:
    Option = Conservative Interface Flux
  END
END
BOUNDARY: Domain Interface 3 Side 2
Boundary Type = INTERFACE
Location = sym32
BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  MASS AND MOMENTUM:
    Option = Conservative Interface Flux
  END
  TURBULENCE:

```

```

    Option = Conservative Interface Flux
  END
END
BOUNDARY: hot_in
  Boundary Type = INLET
  Location = hot_in
  Use Profile Data = False
  BOUNDARY CONDITIONS:
    FLOW DIRECTION:
      Option = Normal to Boundary Condition
    END
    FLOW REGIME:
      Option = Subsonic
    END
    HEAT TRANSFER:
      Option = Total Temperature
      Total Temperature = 355 [K]
    END
    MASS AND MOMENTUM:
      Mass Flow Rate = 0.312 [kg s-1]
      Mass Flow Rate Area = As Specified
      Option = Mass Flow Rate
    END
    TURBULENCE:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
BOUNDARY: hot_out
  Boundary Type = OPENING
  Location = F154.120
  Use Profile Data = False
  BOUNDARY CONDITIONS:
    FLOW DIRECTION:
      Option = Normal to Boundary Condition
    END
    FLOW REGIME:
      Option = Subsonic
    END
    HEAT TRANSFER:
      Opening Temperature = 348.9 [K]
      Option = Opening Temperature
    END
    MASS AND MOMENTUM:
      Option = Opening Pressure and Direction
      Relative Pressure = 0 [Pa]
    END
    TURBULENCE:
      Option = Medium Intensity and Eddy Viscosity Ratio
    END
  END
END
DOMAIN MODELS:
  BUOYANCY MODEL:
    Option = Non Buoyant
  END
  DOMAIN MOTION:
    Option = Stationary
  END
  MESH DEFORMATION:

```

```

Option = None
END
REFERENCE PRESSURE:
  Reference Pressure = 1 [atm]
END
END
FLUID DEFINITION: Fluid 1
  Material = Water
  Option = Material Library
MORPHOLOGY:
  Option = Continuous Fluid
END
END
FLUID MODELS:
COMBUSTION MODEL:
  Option = None
END
HEAT TRANSFER MODEL:
  Option = Thermal Energy
END
THERMAL RADIATION MODEL:
  Option = None
END
TURBULENCE MODEL:
  Option = SST
END
TURBULENT WALL FUNCTIONS:
  Option = Automatic
END
END
INITIALISATION:
  Option = Automatic
INITIAL CONDITIONS:
  Velocity Type = Cartesian
CARTESIAN VELOCITY COMPONENTS:
  Option = Automatic
END
STATIC PRESSURE:
  Option = Automatic
END
TEMPERATURE:
  Option = Automatic
END
TURBULENCE INITIAL CONDITIONS:
  Option = Medium Intensity and Eddy Viscosity Ratio
END
END
END
DOMAIN: tube
  Coord Frame = Coord 0
  Domain Type = Fluid
  Location = B363
BOUNDARY: Default Fluid Fluid Interface Side 2
  Boundary Type = INTERFACE
  Location = \
    F371.363,F372.363,F373.363,F374.363,F380.363,F381.363,F382.363,F383.36\
    3,F384.363,F385.363,F386.363,F387.363,F388.363,F389.363,F390.363,F391.\
    363,F392.363,F393.363,F394.363,F395.363,F396.363,F397.363,F398.363,F39\
    9.363,F400.363,F401.363,F402.363,F403.363,F404.363,F405.363,F406.363,F\
    407.363

```

```

BOUNDARY CONDITIONS:
  HEAT TRANSFER:
    Option = Conservative Interface Flux
  END
  WALL ROUGHNESS:
    Option = Smooth Wall
  END
END
BOUNDARY: cold_in
Boundary Type = INLET
Location = cold_in
Use Profile Data = On
BOUNDARY CONDITIONS:
  FLOW REGIME:
    Option = Subsonic
  END
  HEAT TRANSFER:
    Option = Total Temperature
    Total Temperature = 305 [K]
  END
  MASS AND MOMENTUM:
    Option = Cartesian Velocity Components
    U = profileTTFD.Velocity u(x,y,z)
    V = profileTTFD.Velocity v(x,y,z)
    W = profileTTFD.Velocity w(x,y,z)
  END
  TURBULENCE:
    Option = Medium Intensity and Eddy Viscosity Ratio
  END
END
BOUNDARY PROFILE:
  Profile Name = profileTTFD
END
BOUNDARY: cold_out
Boundary Type = OUTLET
Location = cold_out
Use Profile Data = False
BOUNDARY CONDITIONS:
  FLOW REGIME:
    Option = Subsonic
  END
  MASS AND MOMENTUM:
    Option = Average Static Pressure
    Pressure Profile Blend = 0.05
    Relative Pressure = 0 [Pa]
  END
  PRESSURE AVERAGING:
    Option = Average Over Whole Outlet
  END
END
BOUNDARY: tube Default
Boundary Type = WALL
Location = \
  F364.363,F365.363,F366.363,F367.363,F368.363,F369.363,F370.363,F376.36\
  3,F377.363,F378.363
Use Profile Data = False
BOUNDARY CONDITIONS:
  HEAT TRANSFER:

```

```

    Option = Adiabatic
END
MASS AND MOMENTUM:
    Option = No Slip Wall
END
WALL ROUGHNESS:
    Option = Smooth Wall
END
END
END
DOMAIN MODELS:
BUOYANCY MODEL:
    Option = Non Buoyant
END
DOMAIN MOTION:
    Option = Stationary
END
MESH DEFORMATION:
    Option = None
END
REFERENCE PRESSURE:
    Reference Pressure = 1 [atm]
END
END
FLUID DEFINITION: Fluid 1
    Material = seawater
    Option = Material Library
MORPHOLOGY:
    Option = Continuous Fluid
END
END
FLUID MODELS:
COMBUSTION MODEL:
    Option = None
END
HEAT TRANSFER MODEL:
    Option = Thermal Energy
END
THERMAL RADIATION MODEL:
    Option = None
END
TURBULENCE MODEL:
    Option = SST
END
TURBULENT WALL FUNCTIONS:
    Option = Automatic
END
END
INITIALISATION:
    Option = Automatic
INITIAL CONDITIONS:
    Velocity Type = Cartesian
CARTESIAN VELOCITY COMPONENTS:
    Option = Automatic
END
STATIC PRESSURE:
    Option = Automatic
END
TEMPERATURE:
    Option = Automatic
END

```

```

TURBULENCE INITIAL CONDITIONS:
  Option = Medium Intensity and Eddy Viscosity Ratio
END
END
END
END
DOMAIN INTERFACE: Default Fluid Fluid Interface
Boundary List1 = Default Fluid Fluid Interface Side 1
Boundary List2 = Default Fluid Fluid Interface Side 2
Interface Type = Fluid Fluid
INTERFACE MODELS:
  Option = General Connection
FRAME CHANGE:
  Option = None
END
HEAT TRANSFER:
  Option = Conservative Interface Flux
HEAT TRANSFER INTERFACE MODEL:
  Material = ti64
  Option = Thin Material
  Thickness = 1 [mm]
END
END
MASS AND MOMENTUM:
  Option = No Slip Wall
END
PITCH CHANGE:
  Option = None
END
END
MESH CONNECTION:
  Option = GGI
END
END
DOMAIN INTERFACE: Domain Interface 1
Boundary List1 = Domain Interface 1 Side 1
Boundary List2 = Domain Interface 1 Side 2
Interface Type = Fluid Fluid
INTERFACE MODELS:
  Option = Translational Periodicity
MASS AND MOMENTUM:
  Option = Conservative Interface Flux
MOMENTUM INTERFACE MODEL:
  Option = None
END
END
END
MESH CONNECTION:
  Option = Automatic
END
END
DOMAIN INTERFACE: Domain Interface 2
Boundary List1 = Domain Interface 2 Side 1
Boundary List2 = Domain Interface 2 Side 2
Interface Type = Fluid Fluid
INTERFACE MODELS:
  Option = Translational Periodicity
MASS AND MOMENTUM:
  Option = Conservative Interface Flux
MOMENTUM INTERFACE MODEL:
  Option = None

```

```

    END
  END
END
MESH CONNECTION:
  Option = Automatic
END
END
DOMAIN INTERFACE: Domain Interface 3
  Boundary List1 = Domain Interface 3 Side 1
  Boundary List2 = Domain Interface 3 Side 2
  Interface Type = Fluid Fluid
INTERFACE MODELS:
  Option = Translational Periodicity
MASS AND MOMENTUM:
  Option = Conservative Interface Flux
MOMENTUM INTERFACE MODEL:
  Option = None
END
END
MESH CONNECTION:
  Option = Automatic
END
END
OUTPUT CONTROL:
MONITOR OBJECTS:
MONITOR BALANCES:
  Option = Full
END
MONITOR FORCES:
  Option = Full
END
MONITOR PARTICLES:
  Option = Full
END
MONITOR POINT: hotoutmon
  Coord Frame = Coord 0
  Expression Value = T hot out
  Option = Expression
END
MONITOR POINT: shellbalmon
  Coord Frame = Coord 0
  Expression Value = shellbal
  Option = Expression
END
MONITOR POINT: tubebalmon
  Coord Frame = Coord 0
  Expression Value = tubebal
  Option = Expression
END
MONITOR RESIDUALS:
  Option = Full
END
MONITOR TOTALS:
  Option = Full
END
RESULTS:
  File Compression Level = Default
  Option = Standard
  Output Equation Residuals = All

```

```

END
END
SOLVER CONTROL:
  Turbulence Numerics = First Order
ADVECTION SCHEME:
  Option = High Resolution
END
CONVERGENCE CONTROL:
  Length Scale Option = Conservative
  Maximum Number of Iterations = 2000
  Minimum Number of Iterations = 1
  Timescale Control = Auto Timescale
  Timescale Factor = 1.0
END
CONVERGENCE CRITERIA:
  Residual Target = 1e-20
  Residual Type = RMS
END
DYNAMIC MODEL CONTROL:
  Global Dynamic Model Control = On
END
INTERRUPT CONTROL:
  Option = Any Interrupt
CONVERGENCE CONDITIONS:
  Option = Default Conditions
END
END
END
EXPERT PARAMETERS:
  tbulk for htc = 303.8
END
END
COMMAND FILE:
  Version = 22.1
END
SIMULATION CONTROL:
EXECUTION CONTROL:
  EXECUTABLE SELECTION:
    Double Precision = On
    Large Problem = On
  END
  INTERPOLATOR STEP CONTROL:
    Runtime Priority = Standard
  END
  PARTITIONER STEP CONTROL:
    Multidomain Option = Automatic
    Runtime Priority = Standard
  MEMORY CONTROL:
    Memory Allocation Factor = 1.5
  END
  PARTITION SMOOTHING:
    Maximum Partition Smoothing Sweeps = 100
    Option = Smooth
  END
  PARTITIONING TYPE:
    MeTiS Type = k-way
    Multipass Partitioning Option = None
    Option = MeTiS
    Partition Size Rule = Automatic
  END
END

```

```
RUN DEFINITION:
  Run Mode = Full
  Solver Input File = ./TT-redo.def
END
SOLVER STEP CONTROL:
  Runtime Priority = Standard
MEMORY CONTROL:
  Memory Allocation Factor = 1.5
END
PARALLEL ENVIRONMENT:
  Number of Processes = 256
  Parallel Host List = RMKW1-2022-TCR1
  Start Method = Intel MPI Local Parallel
END
END
END
END
```

APPENDIX C. SAMPLE BATCH SHELL SUBMISSION SCRIPT

```
#!/usr/bin/csh
#SBATCH --job-name=BYRNE19mil
#SBATCH --error=19mil.err.%j
#SBATCH --output=19mil.out.%j
#SBATCH --time=500:10:00
#SBATCH --nodes=2
#SBATCH --ntasks=128
#SBATCH --tasks-per-node=64
#SBATCH --mem-per-cpu=1024M
#SBATCH --mail-type=END
#SBATCH --no-requeue
###SBATCH --exclude=compute-2-39,compute-7-9

###. /etc/profile
module purge
module load app/ansys/20.2

set MYFILEHOSTS="hosts.$SLURM_JOB_ID"
srun hostname | sort > $MYFILEHOSTS
###echo $MYFILEHOSTS > test.hostlist.txt
###echo $MYFILEHOSTS

###set PARLIST=`cat $MYFILEHOSTS`
###echo $PARLIST

wait

foreach i (`scontrol show hostname "$SLURM_NODELIST" | tr "\n" " ")
    set compnode=`grep "^${i}.hamming.cluster" ~/.ssh/known_hosts`
    if ("${compnode}" == "" ) then
        ssh-keyscan -t ecdsa "${i}.hamming.cluster" | tee -a ~/.ssh/known_hosts;
    endif
end

sed -z 's/\n/,g;s/,,$/\n/' $MYFILEHOSTS > tmp.hosts

wait

set PARLIST=`cat tmp.hosts`
echo $PARLIST

wait

cfx5solve -size 2 -def 19mil-element.def -parallel -par-dist $PARLIST -start-method "Intel MPI Distributed Parallel"
```

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF REFERENCES

- [1] NRC, “NRC: Package ML11171A500 -Westinghouse AP1000 Design Control Document Rev. 19.” Accessed: Mar. 28, 2023. Available: <https://www.nrc.gov/docs/ML1117/ML11171A500.html>
- [2] USNI, Shellbourne, M., “Cost estimates for lead boat in Columbia-class program grow by \$637M,” *USNI News*, Jun. 07, 2021. Available: <https://news.usni.org/2021/06/07/cost-estimates-for-lead-boat-in-columbia-class-program-grow-by-637m> (accessed Mar. 28, 2023).
- [3] Caputo, A. C., Pelagagge, P. M., and Salini, P., “Heat exchanger design based on economic optimisation,” *Appl. Therm. Eng.*, vol. 28, no. 10, pp. 1151–1159, Jul. 2008. Available: <https://doi.org/10.1016/j.applthermaleng.2007.08.010>.
- [4] Kakaç, S., Liu, H., and Pramuanjaroenkij, A., *Heat Exchangers: Selection, Rating, and Thermal Design, Fourth Edition*, 4th ed. Boca Raton: CRC Press, 2020. Available: <https://doi.org/10.1201/9780429469862>.
- [5] Thulukkanam, K., *Heat Exchanger Design Handbook*, 2nd ed. Boca Raton: CRC Press, 2013. Available: <https://doi.org/10.1201/b14877>.
- [6] Barba, A., Rainieri, S., and Spiga, M., “Heat transfer enhancement in a corrugated tube,” *Int. Commun. Heat Mass Transf.*, vol. 29, pp. 313–322, Apr. 2002. Available: [https://doi.org/10.1016/S0735-1933\(02\)00321-4](https://doi.org/10.1016/S0735-1933(02)00321-4).
- [7] Wu, Z., Qian, C., Liu, G., Liu, Z., and Sheng, P., “Mechanical properties and heat transfer performance of conically corrugated tube,” *Materials*, vol. 14, no. 17, p. 4902, Aug. 2021. Available: <https://doi.org/10.3390/ma14174902>.
- [8] Chiou, J. P., “Experimental investigation of the augmentation of forced convection heat transfer in a circular tube using spiral spring inserts,” *J. Heat Transf.*, vol. 109, no. 2, pp. 300–307, May 1987. Available: <https://doi.org/10.1115/1.3248080>.
- [9] Vicente, P. G., García, A., and Viedma, A., “Experimental investigation on heat transfer and frictional characteristics of spirally corrugated tubes in turbulent flow at different Prandtl numbers,” *Int. J. Heat Mass Transf.*, vol. 47, no. 4, pp. 671–681, Feb. 2004. Available: <https://doi.org/10.1016/j.ijheatmasstransfer.2003.08.005>.
- [10] Qian, J., Yang, C., Chen, M., and Jin, Z., “Thermohydraulic performance evaluation of multi-start spirally corrugated tubes,” *Int. J. Heat Mass Transf.*, vol. 156, p. 119876, Aug. 2020. Available: <https://doi.org/10.1016/j.ijheatmasstransfer.2020.119876>.

- [11] Zimparov, V., “Enhancement of heat transfer by a combination of three-start spirally corrugated tubes with a twisted tape,” *Int. J. Heat Mass Transf.*, vol. 44, no. 3, pp. 551–574, Feb. 2001. Available: [https://doi.org/10.1016/S0017-9310\(00\)00126-5](https://doi.org/10.1016/S0017-9310(00)00126-5).
- [12] Wiley, “Fundamentals of heat and mass transfer, 8th edition.” Accessed Apr. 26, 2023. Available: <https://www.wiley.com/en-be/Fundamentals+of+Heat+and+Mass+Transfer%2C+8th+Edition-p-9781119353881>
- [13] Jones, W. P., “The Prediction of laminarization with a two-equation model of turbulence,” Available: https://www.researchgate.net/publication/313117966_The_Prediction_of_Laminarization_with_a_Two-Equation_Model_of_Turbulence (accessed Mar. 08, 2023).
- [14] Menter, F. R., “Influence of freestream values on k-omega turbulence model predictions,” *AIAA J.*, vol. 30, no. 6, pp. 1657–1659, Jun. 1992. Available: <https://doi.org/10.2514/3.11115>.
- [15] Wilcox, D. C., “Formulation of the k-w turbulence model revisited,” *AIAA J.*, vol. 46, no. 11, pp. 2823–2838, Nov. 2008. Available: <https://doi.org/10.2514/1.36541>.
- [16] Menter, F. R., “Two-equation eddy-viscosity turbulence models for engineering applications,” *AIAA J.*, vol. 32, no. 8, pp. 1598–1605, Aug. 1994. Available: <https://doi.org/10.2514/3.12149>.
- [17] Bredberg, J., “On the wall boundary condition for turbulence models.” Chalmers University of Technology. Gotēborg, Sweden. Available: https://www.tfd.chalmers.se/~lada/postscript_files/jonas_report_WF.pdf
- [18] Salim, M. S., and Cheah, S. C., “Wall y^+ strategy for dealing with wall-bounded turbulent flows,” presented at the Lecture Notes in Engineering and Computer Science, Jan. 2009.
- [19] Tang, X., Dai, X., and Zhu, D., “Experimental and numerical investigation of convective heat transfer and fluid flow in twisted spiral tube,” *Int. J. Heat Mass Transf.*, vol. 90, pp. 523–541, Nov. 2015. Available: <https://doi.org/10.1016/j.ijheatmasstransfer.2015.06.068>.
- [20] Bernardin, J. D., Ferguson, K., Sattler, D., and Kim, S. J., “The Design, Analysis, and Fabrication of an Additively Manufactured Twisted Tube Heat Exchanger,” in *ASME 2017 Heat Transfer Summer Conference*, Bellevue, Washington, USA: American Society of Mechanical Engineers, Jul. 2017, p. V002T10A002. Available: <https://doi.org/10.1115/HT2017-4866>.

- [21] Bernardin, J. D., Ferguson, K., and Sattler, D., “The Testing and Model Validation of an Additively Manufactured Twisted Tube Heat Exchanger,” presented at the ASME 2019 Heat Transfer Summer Conference collocated with the ASME 2019 13th International Conference on Energy Sustainability, American Society of Mechanical Engineers Digital Collection, Dec. 2019. Available: <https://doi.org/10.1115/HT2019-3500>.
- [22] Gerstler, W. D., and Erno, D., “Introduction of an Additively Manufactured multi-Furcating Heat Exchanger,” in *2017 16th IEEE Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm)*, May 2017, pp. 624–633. Available: <https://doi.org/10.1109/ITHERM.2017.7992545>.
- [23] Arie, M. A., Shooshtari, A. H., Dessiatoun, S. V., and Ohadi, M. M., “Performance Characterization of an Additively Manufactured Titanium (Ti64) Heat Exchanger for an Air-Water Cooling Application,” in *ASME 2016 Heat Transfer Summer Conference*, 2016. Available: <https://doi.org/10.1115/HT2016-1059>.
- [24] Zhang, X., Tiwari, R., Shooshtari, A. H., and Ohadi, M. M., “An Additively Manufactured Metallic Manifold-microchannel Heat Exchanger for High Temperature Applications,” *Appl. Therm. Eng.*, vol. 143, pp. 899–908, Oct. 2018. Available: <https://doi.org/10.1016/j.applthermaleng.2018.08.032>.
- [25] Velo3D, “Sapphire XC and Sapphire XC 1MZ Printer,” *Velo3D | Leading Provider of Metal 3D Printing Solutions*, Accessed: May 1, 2023, Available: <https://velo3d.com/sapphire-xc-sapphire-xc-1mz-printer/>
- [26] EOS, “EOS M 400–4 - Ultra Fast 3D Printing.” Accessed: Apr 18, 2023. Available: <https://www.eos.info/en/industrial-3d-printer/metal/eos-m-400-4>
- [27] Jin, Z., Chen, F., Gao, F., Gao, X., and Qian, J., “Effects of Pitch and Corrugation Depth on Heat Transfer Characteristics in Six-start Spirally Corrugated Tube,” *Int. J. Heat Mass Transf.*, vol. 108, pp. 1011–1025, May 2017, Available: <https://doi.org/10.1016/j.ijheatmasstransfer.2016.12.091>.
- [28] NASA, “Sea Surface Temperature (AQUA/MODIS),” NASA database. Accessed Jun. 15, 2023. Available: <https://neo.gsfc.nasa.gov/view.php?datasetId=MYD28W&date=2022-04-01>
- [29] Weisstein, E. W., “Circle-Circle Intersection.” From *MathWorld*--A Wolfram Web Resource. Available: <https://mathworld.wolfram.com/Circle-CircleIntersection.html>
- [30] Clonts, R., email, Feb. 2023.
- [31] ANSYS Inc., Ansys® Meshing User’s Guide, Release 2021 R2.

- [32] ANSYS Inc., Ansys® CFX-Pre User's Guide, Release 2021 R2.
- [33] Nayar, K. G., Sharqawy, M. H., Banchik, L. D., and Lienhard V, J. H., "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>.
- [34] Sharqawy, M. H., Lienhard V., J. H., and Zubair, S. M., "The Thermophysical Properties of Seawater: A Review of Existing Correlations and Data," *Prof Lienhard Angie Locknar*, Apr. 2010. Accessed: May 01, 2023. Available: <https://dspace.mit.edu/handle/1721.1/69157>
- [35] ASM, "ASM Material Data Sheet," Accessed: May 1, 2023. Available: <https://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MTP641>
- [36] Proto3000, "Titanium (Ti64)." Accessed: May 1, 2023. Available: <https://proto3000.com/materials/dmls-titanium-ti64/>
- [37] Bartsch, K. Bossen, B., Chaudhary, W., Landry, M., and Herzog, D., 2022, "Thermal Conductivity of Ti-6Al-4V in Laser Powder Bed Fusion." *Frontiers in Mechanical Engineering*. Available: <https://doi.org/10.3389/fmech.2022.830104>
- [38] Mierzejewska, Z. A., Hudák, R., and Sidun, J., "Mechanical Properties and Microstructure of DMLS Ti6Al4V Alloy Dedicated to Biomedical Applications," *Materials*, vol. 12, no. 1, p. 176, Jan. 2019. Available: <https://doi.org/10.3390/ma12010176>

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