3D-Printed Heat Exchanger

Johannes Ljungman

DIVISION OF PRODUCT DEVELOPMENT | DEPARTMENT OF DESIGN SCIENCES
FACULTY OF ENGINEERING LTH | LUND UNIVERSITY
2019

MASTER THESIS

3D printed heat exchanger

Johannes Ljungman

3D-Printed Heat Exchanger

Copyright © 2019 Johannes Ljungman

Published by

Department of Design Sciences
Faculty of Engineering LTH, Lund University
P.O. Box 118, SE-221 00 Lund, Sweden

Subject: Product Development (MMKM05)
Division: Division of Product Development
Supervisor: Martin Andersson
Examiner: Olaf Diegel

Abstract

Additive manufacturing, or 3D-printing, is due to its many advantages an increasingly used form of manufacturing. The design freedom, reduced material waste, shortened process from drawing to finished product, and the ability to print moving parts make it extremely interesting for many applications. One application where not much research has been done, but there is an increasing interest, is the field of heat exchangers.

Heat exchangers are used in countless applications such as industry, vehicles, computers, power plants etc. The process of designing a heat exchanger is complex and even the smallest geometric changes can have a great impact of their performance. Heat transfer area is one of the limiting factors for heat transfer and adding area with complex geometries is made available with the design freedom additive manufacturing brings.

A Shell-and-Tube heat exchanger was designed using CAD, for the purpose of 3D-printing. Calculations were first made to determine the area increase compared to a similar conventional heat exchanger. The two heat exchangers were also compared, through CFD simulations to determine the possibilities for increased heat transfer. Parameters such as pressure drop, temperature and fluid flow were analyzed. From theory, adding area will increase heat transfer, but from the results of the CFD analysis no such conclusion can be drawn. No real temperature difference could be seen when comparing complex to conventional heat exchangers.

The work was also a study in how well modern 3D-printers, for metal components, handle complex surfaces and narrow tube passages. The Shell-and-Tube was designed to minimize support material but powder removal also a factor that was investigated.

Keywords: Additive manufacturing, CAD, CFD, heat exchanger, heat transfer

Sammanfattning

Additiv tillverkning, eller 3D-printing, är på grund av sina många fördelar en ständigt ökande tillverkningsmetod. Designfriheten, det reducerade materialspillet, förkortad tid från ritning till färdig produkt och förmågan att kunna printa rörliga delar gör 3D-printing intressant för många olika användningsområden. Ett användningsområde där relativt lite forskning gjorts men som har ett ökat intresse är värmeväxlare.

Värmeväxlare används idag i oräkneliga applikationer, bland annat i industrin, fordon, datorer, kraftverk etc. Att designa en värmeväxlare är en komplex process där de minsta ändringarna på geometri kan ha stora konsekvenser för prestandan. Värmeöverföringsarea är en av de begränsande faktorerna för värmeöverföring. En möjlighet som designfriheten additiv tillverkning medför är att man kan addera till arean genom komplexa geometrier.

En Shell-and-Tube värmeväxlare ämnad för 3D-printing designades med CAD. Först gjordes beräkningar som jämförde den areaökning som kunde erhållas jämfört med en konventionell liknande värmeväxlare. De båda värmeväxlarna jämfördes även med CFD analys för att utvärdera möjligheterna för ökad värmeöverföring. Även parametrar som tryck, temperatur och flöde analyserades. Enligt teorin ska ökad area innebära mer värmeöverföring men från resultaten av CFD kan inga generella slutsatser dras. Inga direkta temperaturskillnader syntes vid jämförelsen av de båda värmeväxlarna.

Arbetet var även en studie i hur väl moderna 3D-printrar ämnade för metallkomponenter kan hantera komplexa ytor och smala passager. Shell-and-Tube värmeväxlaren designades för att minimera mängden supportmaterial och hur väl pulver kunde avlägsnas undersöktes också.

Nyckelord: Additiv tillverkning, CAD, CFD, värmeväxlare, värmeöverföring

Acknowledgments

I wish to express my gratitude towards some people without whom this project would not have gotten far.

First, I would like to thank my supervisor, Martin Andersson, for help steering me in the right direction when nothing seemed to work.

I would also like to thank Axel Nordin for always taking the time to answer questions and discuss my work.

Thanks to Hanna Gothäll at COMSOL Technical Support for helping me when I was getting nowhere.

Thanks to Rudolf Abelin at DDG for setting up the simulation server for me.

Lastly, thanks to my family for support through the work with this thesis.

Lund, May 2019

Johannes Ljungman

Table of contents

List of acronyms and abbreviations

AM	additive manufacturing
CFD	computational fluid dynamics
CAD	computer aided design
EBM	electron beam melting
HEX	heat exchanger
PHE	plate heat exchanger
SLM	selective laser melting
SLS	selective laser sintering
S&T	shell-and-tube

1 Introduction

1.1 Background

Heat exchangers (HEX) are used in countless applications. Areas of use can be vehicles, industries, computer cooling, air conditioners etc. Since the use of heat exchangers is so extensive there is a desire to optimize their effectiveness and geometric properties.

Additive manufacturing (AM) is becoming more and more used in the industry to deliver on narrower set design parameters. By manufacturing with AM much more complex and previously impossible designs can be produced. When looking at HEX, even very small design changes can have a large impact of the overall efficiency of the product. However, there are some drawbacks that need to be considered. When AM is applied there is always some residual material such as support material, particles and feature sizes that need to be removed. There is also a surface roughness that can be quite extensive, especially on some surfaces. Physical tests are necessary since many of these factors are hard to simulate.

This master thesis will be part of the project "ReLed-3D, Resource efficient flexible production in the automotive industry through additive metal manufacturing", funded by VINNOVA FFI – Strategic Vehicle Research and Innovation [1].

1.2 Aim and purpose

The aim and purpose of this master thesis is to design, simulate, and test 3D-printed HEX. The belief is that the results will show the advantage of AM and give an idea of feasible designs that will help optimize heat exchangers for commercial and industrial use.

The approach will be to do a comparable study of heat exchangers and through this evaluate if a different design is easy to realize for the same operating conditions.

1.3 State of the art

There has already been some research done in the field of additive manufactured heat exchangers and what the benefits of such techniques are [2–5]. The great benefits of AM are that many steps in the manufacturing process of the part can be skipped. It is possible to manufacture complex and working parts, with only a computer aided design (CAD) model as instruction. The part is then built through selective laser melting (SLM) and all joints are created as it is printed. For a HEX the freedom of design is of great interest since even the smallest design changes can have big impacts on the performance of the HEX. In this section some state of the art in this field is presented.

1.3.1 Design characterization

As mentioned previously, it is of great interest to look at the complexity of design made possible through AM. Brandon J. et al. [3] manufactured a hydraulic oil cooler with the use of AM to compare with a stock finned HEX of the same dimensions. The results of their study showed some difference in performance of the two heat exchangers, especially when it came to pressure drop. The purpose of the study was not, however, to design a heat exchanger with a superior heat transfer but rather to see the limitations and possibilities of AM in this application. They concluded the pressure drop, which was greater than the specifications, to be adherent to lesser convective heat transfer. This was due to excess material not being removed after manufacturing, which illustrates the importance of design rules/guidelines when applying AM. The HEX they designed had complex geometries, such as lenticular shaped manifolds (instead of circular) and offset strip fins inside tubes. They also had plate fins on the air side printed at a 45° angle, compared to the tubes, instead of more conventional horizontal fins. This was compared to a microchannel aluminium oil cooler with plain fins on the air side and offset strip fins on the oil side.

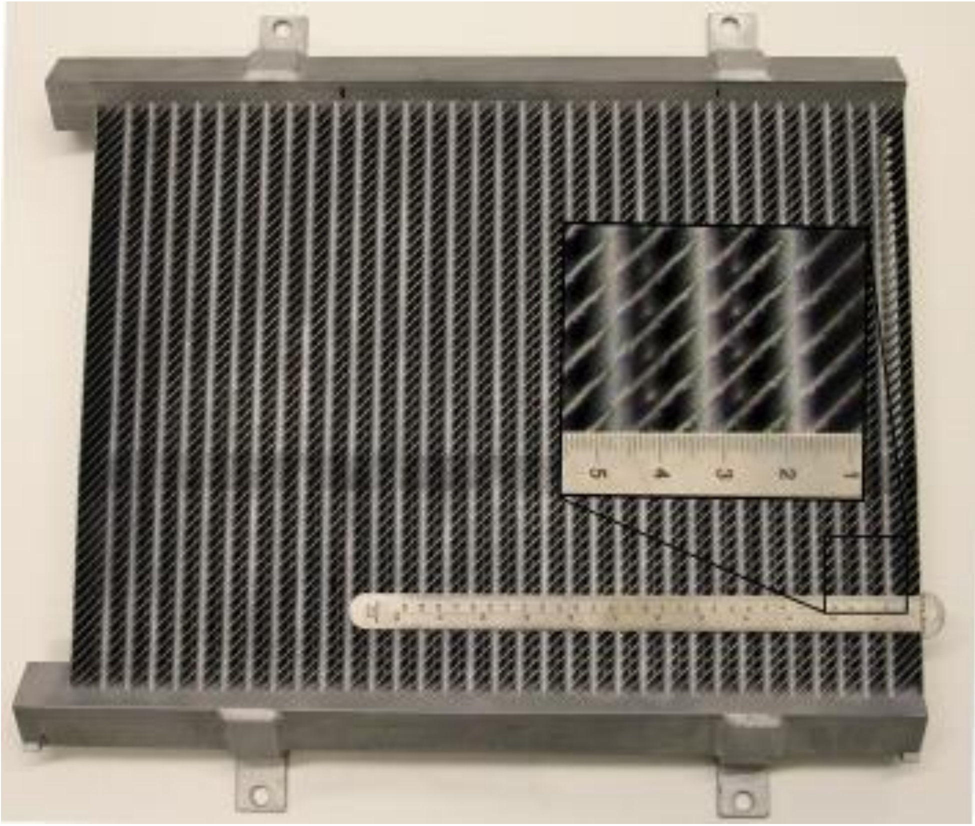

Figure 1 Photograph of the printed heat exchanger with inset close-up of three rows of tubes and accompanying scale[3]**. Copyright © 2018 Elsevier & Copyright Clearance Center.**

Kirsch and Thole [4] investigated the heat transfer and pressure loss performance of additive manufactured wavy channels. This is another example of design freedom possible with AM. Their result shows that short wavelength channels yield high pressure losses, without corresponding increases in heat transfer, due to the flow structure promoted by the waves. Longer wavelength offers less of a penalty in pressure losses with good heat transfer performance.

Other fields of interest that have rapidly been improved are 3D printing of porous beds. AM with the help of CAD instructions can now print extremely fine geometries, and one such is porous bed, which up until now have been layered randomly in their columns. However, Conan Fee et al. [5] discusses the drawbacks and limitations of available equipment. Firstly, the rate of printing is currently too slow to enable enough rapid manufacturing for even the smallest mesostructured[1] porous beds. An object of 20 cm printed in 6 μm layers at 1 min per layer would take 23 days to complete. Secondly, he further discusses the limitations of CAD software, claiming the CAD files describing such an object comprise many gigabytes of information and that the file-size is beyond what is possible for commercial CAD software as well as the graphics processing capability.

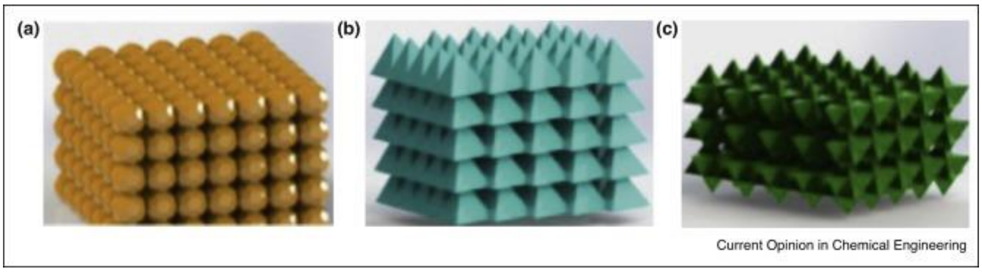

Figure 2 Simple cubic lattices of (a) spheres, (b) tetrahedra, (c) stella octangulae. Copyright © 2017 Elsevier & Copyright Clearance Center.

A field where 3D printing is already widely used is within aerospace technology. General Electrics, who among many other things, are invested in aviation have been researching 3D printed heat exchangers. They have been developing a "super-compact" heat exchanger, which would not be possible to manufacture in any other way than through additive manufacturing [2][2]. They are with the university of Maryland exploring more intricate biological shapes and designs. The human lung is one of the most efficient heat exchangers known [6] and if one could design even slightly towards such intricate shapes it could have a huge benefit for heat exchangers. This is of course extremely expensive and is mostly at research level, but it could become more common in highly technological fields such as mentioned aerospace and within medicine.

[1] Solid and fluid phase geometric feature i.e. particle packing arrangement, particle shape or extraparticle void shape.

[2] Although the reference is questionable, it was still decided interesting to mention. There are no scientific papers to be found on the subject, which is reasonable due to trade secrets.

1.4 Scope and delimitations

After considering what was mentioned above the scope of the thesis materializes. Although porous beds would be interesting to research further the methods for production available still make it rather difficult to realize. The focus will instead be on geometry changes that could be printed and simulated with relative ease. For this it will be of interest looking to the work by Brandon J. et al. [3] did on hydraulic oil cooler or designing in a similar manner something that has not been tested.

The work will not go into too much depth on heat transfer theory, where governing equations etc are set up. For the interested, this theory can be found in Introduction to heat transfer [7]. Instead, some theory on heat exchangers will be explained and analytical calculations, where possible, shall be done. More focus will be on numerical calculations since, especially for the more complex geometries, analytical calculations will be difficult.

Some delimitations will be needed to get the project moving forward. The heat exchanger will be modelled with the assumption it is insulated. For the most accurate results it would have been best to model according to ambient temperature and calculating heat flux (heat transferred energy to environment), but information was not made available.

2 Method

2.1 Methodology selection

The study of heat transfer is a complex field analytically and considering the complex nature of the project CFD will be one of the main methods of choice. Computational Fluid Dynamics (CFD) is a finite volume method used to simulate heat transfer and other forms of energy transfer in fluids. This is the cheaper and easier option compared to physical testing and a proven tool for engineering applications.

CAD as a method is necessary for the work with this thesis. Most CFD developers offer the option to build the design using their software, but they are generally much less user friendly and do not offer the same opportunities to refine the model as CAD dedicated software do. A reason to use the simulation dedicated software for modelling would be to decrease the probability of import errors but it is still preferable to use CAD dedicated software for the design.

When designing the method is to, with restraints formulated previously, use an iterative approach for both CAD modelling and CFD simulations. Reasoning for the method being iterative is that as limitations to the design are made obvious, changes will need to be performed to enable progress in the work with the thesis. Sometimes there is a need to go two steps back to move one step forward. The aim is to design a few different models that can be simulated to see the effect of design choices made. It might be necessary to design a smaller model of the end design to have something reasonable to simulate since a larger design will require too much computing power.

Concept generation is to be done as a reflection of the results of similar studies. What has been the outcomes of these studies and what can be improved to further enhance the test results. Ulrich and Eppinger point out the importance of external research and benchmarking [8], and these have been done through the work of pre-study. The product specification will be limited according to design rules for AM of aluminum components and the previous testing that have been done on the current heat exchanger (Appendix B) can be used as input parameters for calculations.

2.2 Process

After limitations and ideas have been evaluated according to criterions from section 2.1 the next phase of the project commences. For this step it is designing the geometries that should be analyzed. The different models to design are one standardized heat exchanger and one that is meant for 3D-printing and doing a comparison analysis on them. It is here important to check the design for weaknesses according to design rules for 3D-printing, specifically, for 3D-printing in aluminum material. When the design is adequate the next phase is computational fluid dynamics (CFD) simulations. The simulations should be done in a suitable manner, where simplifications should be done wherever possible to shorten the amount of simulation hours.

2.3 Tools

Some calculations will be done analytically but considering the complexity of heat transfer most of the calculations will be done using numerical methods through commercially available software. For this master thesis it has been decided that the main method of use will be simulations through CFD. To do the CFD simulations the software of choice has, after recommendations from associate professor and supervisor Martin Andersson, landed on COMSOL version 5.4. Other possible tools for simulation are Ansys Fluid or Autodesk CFD, among others.

To have something to run simulations on it is necessary to have designs that will be created with the use of Computer Aided Design (CAD), Creo Parametric 5.0.3.0 being the software mainly used.

3 Theory

3.1 Additive Manufacturing

AM has been available as method for manufacturing for some time. Up until recently it has been used mainly for rapid prototyping and rapid tooling, but it is also of great use for production of complicated parts where space and efficiency are of critical importance. Unlike conventional manufacturing where material is removed to form the desired part AM, as the name suggests, adds material and is thus the option with less material waste. T. DebRoy et al. [9] bring up numerous AM techniques, including Electron Beam Melting (EBM), Selective Laser Sintering (SLS), and Selective Laser Melting (SLM).

3.1.1 **Selective laser melting**

SLM is one of the more common methods of additive manufacturing. The technique has sprung from SLS, where a part is printed in powder but where sintering of the part is needed to get the metal powder to bond, forming a solid. The process of SLS is thus divided in the printing part and the sintering. SLM melts the metal powder with high focused laser onto the part until it is completed, making SLM the faster and more advanced option.

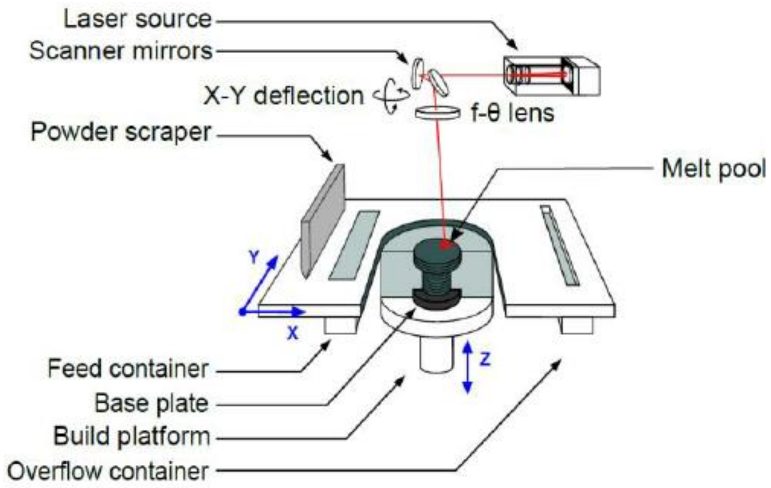

Figure 3 Schematic overview of the SLM process [10] ***reused with permission from the authors*****.**

Figure 3 shows a schematic view of a SLM machine. When the part is being built the powder is, as mentioned, melted and bonds are created. After the laser has melted the powder material together the platform will lower itself and new powder is added on top to form a new layer to be selectively sintered and fused to the part. The process repeats itself until the part is finished.

The material of choice is the material used in the machine at the Department of Design Sciences, AlSi10Mg. AlSi10Mg is particularly good for AM because it has good weldability and hardening while maintaining high thermal conductivity [11].

When regards are taken to possible wall thickness A. You et al. [12] have done a number of test geometries where they found that a wall thickness of 0.2-0.3 mm was bent due to thermally induced stresses. They found that a wall thickness greater or equal to 0.4 mm to be successfully built with no warping. However as Hathaway et al. [3] discuss their research showed that, when designing small tubes, the tube inner diameter of 6 mm causes problems. They discussed that it might have been because the tube length was set at 360 mm some of the powder was not removed in postprocessing. This goes against experiences at the Department for Design Sciences. To get a better idea of printability and narrow passages some test samples have been included which is discussed later in the report.

H.K. Rafi et al. [13] present research of the difference between SLM and EBM in regard to material properties and surface roughness and also mention the different environment in which the part is being built, where EBM build chamber temperature is kept at an elevated temperature of 700˚C. Their findings is that there is a difference between surface of the two techniques where SLM produce the finer surfaces. According to Axel Nordin at the Department for Design Sciences this will affect the powder removal if sections are made very narrow. The EBM process is more likely to create semi-sintered powder if the distance between walls is small and if the part is not printed satisfactory with SLM, it will not work with any other AM method.

It is also worth mentioning the importance of print direction. As P. Herzog discuss how, especially for surfaces printed at an angle, the surface roughness will vary [14]. If the print direction is in the z-direction, surfaces that will be on the downfacing side will have a considerably greater surface roughness than surfaces facing upwards. This is a factor that might be able to circumvent when doing the print. By rotating the part before printing many surfaces and features can be built to a better quality than if the part is placed in a less optimized position. This can also be avoided by careful design when the CAD object is being created.

3.2 Heat transfer

Heat transfer is a form of energy which is always transferred from the hot part to the cold part within a substance or from a body at a high temperature to another body at a lower temperature. The bodies do not need to be in contact but a difference in temperature must exist. In other words, they need to be in thermal contact.

Heat can be transferred in three different forms, namely, heat radiation, conduction and convection.

Heat radiation is a form of heat transfer where no medium is required to propagate the heat. It is the heat transferred between surfaces, or from a surface to surrounding medium, such as the sun radiates heat into space.

Conduction is heat transfer through a solid or in a fluid, such as gas or liquids. The requirement for conduction is transfer from a region at a high temperature to a low temperature region and it is governed by molecular motion at rest, and by movement of electrons as in the case of metals.

Convection is heat transfer that appears as a fluid is flowing along an exterior surface or similar and as the temperature of the fluid and the surface a different, the amount of heat being exchanged is affected by the macroscopic fluid motion. There are two types of convection, forced and free (natural). Forced convection is the result of using fans or pumps, etc. Natural convection occurs from the temperature difference in the fluid results in a movement of the fluid. These convections can also be combined, simply resulting in mixed convection [7].

When considering heat transfer in a HEX some things are of importance. Firstly, one needs to consider the fluids used and the dynamic mechanical work being done in the HEX. When regarding fluid dynamics one speaks of laminar or turbulent flow. For a greater heat transfer the desired flow form is always turbulent due to its superior performance. It is not always the case that turbulent flows are present, and, in some applications, laminar flow is desired. Turbulence (if possible) does however have the better heat transfer qualities. The fluids need to be in the turbulent phase so that there is better transport of heat from walls through the fluid. If the flow is laminar the heat from the wall will mostly stay in the boundary layers closest to the wall section and thus not having optimal heat transfer. One possibility of ensuring turbulence in pipes is to have obstructions in the flow direction. These inner obstructions (fins) are quite possible to create using AM but would be very difficult or impossible to include using conventional methods.

It should be mentioned that phase-change has an even greater heat transfer then forced convection but for this thesis focus will be on forced convection and laminar or turbulent flows.

When creating a 3D-printed part there is also some surface roughness that can be quite coarse after manufacturing. Especially on the bottoms surface, seen from print direction, the surface created is relatively rough, as mentioned in the SLM section. This could however be an advantage and not a disadvantage when it comes to heat transfer. Having a rough surface can help create turbulence and thus enhancing heat transfer. Kandilakar et al. made experiments of narrow tube diameter designs to test heat transfer and pressure drop characteristics with regards to surface roughness and found that especially for small diameters the effect of surface roughness was significant [15].

3.2.1 Heat Exchanger

There are basically two ways to increase the heat transfer for a HEX. These two are divided into passive and active techniques [16]. Passive means special geometries or additives to liquids. For the active, one must add external power and is thus not as desirable from a design perspective. As previously mentioned, this thesis will focus on design perspective so only the passive approach is considered. Passive approach is usually the direction one chooses to go in due to the added cost and energy needed for active techniques.

In thermal analysis of heat exchangers, the total heat flux $\dot{Q}$ (W) is of primary importance. This heat flow is a function of the overall heat transfer coefficient, U, the heat transferring area, A and the proper average temperature[3]:

$$\dot{Q} = UA \cdot \Delta t_m = \frac{1}{TR} \cdot \Delta t_m \tag{3.1}$$

[3] All equations can be found in Introduction to heat transfer [7]

Where TR is the total thermal resistance. TR should take into account all the different resistances between the hot and cold fluids. These in series gives the TR function

$$TR = \frac{1}{\alpha_i A_i} + \frac{1}{\alpha_{F_i} A_i} + \frac{b_w}{\lambda_w A_{v1}} + \frac{1}{\alpha_{F_o} A_o} + \frac{1}{\alpha_o A_o} \quad (3.2)$$

In this function one finds a lot of different parameters that will be of interest for this work. α_i and A_i are adherent to the inner surface where convective heat transfer is present, and α_0, A_0 adhere to the outer surface. b_w is the wall thickness, λ_w is the walls thermal conductivity and A_{v1} is the heat conducting area. The F factors stand for fouling factors, which depend on the fluid on either side of the wall.

This gives an idea that heat transfer is thus coherent to many different parameters and it is not a simple thing to do easy analytical calculations. The heat transfer coefficient, α, changes with geometry, flow field, and the physical properties of the fluid.

Energy balance for the hot side/cold side is given as

$$\dot{Q}_h = \dot{Q}_c \quad (3.3)$$

Where the function for heat transfer is the same for hot and cold side

$$\dot{Q} = (\dot{m}c_p)_c \Delta t_c = C_c \cdot \Delta t_c \quad (3.4)$$

When calculating the temperature difference Δt_m in a heat exchanger Δt_m is the log-mean-temperature-difference (LMTD) and it is usually calculated for counterflow heat exchangers as

$$LMTD = \Delta t_m = \frac{\left(t_{h_{out}} - t_{c_{in}}\right) - \left(t_{h_{in}} - t_{c_{out}}\right)}{ln \frac{\left(t_{h_{out}} - t_{c_{in}}\right)}{\left(t_{h_{in}} - t_{c_{out}}\right)}} \quad (3.5)$$

Since this equation is for counterflow heat exchangers a correction factor F is introduced if one does not have a counterflow HEX. To calculate this correction factor further information can be found in *Introduction to heat transfer* which has been referenced earlier.

3.2.2 Compact heat exchanger

Sundén [7] writes in the Introduction to heat transfer what defines a HEX as being compact. Compactness is defined as the heat transferring area to volume ratio. This ratio is $\frac{A}{V} > 700 \frac{m^2}{m^3}$ and for some radiators in cars the ratio is $\frac{A}{V} > 1100 \frac{m^2}{m^3}$. The compactness of a heat exchanger can also be defined by the hydraulic diameter being $D_h \leq$ 6 mm [6]. When regarding Shell-and-Tube (S&T) HEX, who are mainly used in industry, this ratio is often 70-500 $\frac{m^2}{m^3}$ making them considered not compact.

Hydraulic diameter being the other way to classify how compact a heat exchanger can be made is particularly useful when considering plate heat exchangers and other similar heat exchangers. The hydraulic diameter is defined as 4 times cross sectional area, divided by the wetted perimeter or;

$$D_h = \frac{4 \cdot minimum\ free\ flow\ area}{Wetted\ perimeter} = \frac{4 \cdot H}{\Phi} \tag{3.6}$$

Where H is the height of the inter-plate channel [17].

3.2.3 Heat transferring area

The heat transferring area is, in the case of a Shell-and-Tube, the outer area of the pipes which is in contact with the coolant. Calculating this area for the conventional HEX is straightforward geometry and no further explanation is necessary for this. Calculating the area of helical, oval shapes is however not as intuitive.

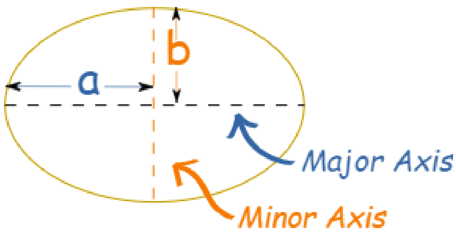

Figure 4 Parameters used for equation 3.9 [26]**.**

There is no exact solution for calculating the circumference of an oval but a good approximation is given in equation 3.9 [26], along with calculations for helical pipe length, 3.10, where d is diameter of the helix and Z is the pitch (length in Z direction):

$$C = \pi \cdot [3 \cdot (a + b) - \sqrt{(3a + b)(a + 3b)}] \tag{3.9}$$

$$L = \sqrt{(\pi d)^2 + Z^2} \tag{3.10}$$

3.2.4 **Flow characteristics**

It is important, as mentioned, that there is turbulence in the flow medium to enhance the heat transfer. When a fluid enters a circular pipe one can determine if the fluid flow is turbulent by calculating the Reynolds number (Re) which is a dimensionless number of inertial and viscous forces, indicating turbulence, where Re > 2300 is usually the indicator for turbulence. This is in other words a quick indication if there is turbulence which is a simple check one should use before starting complicated calculations.

$$Re = \frac{\rho U D}{\mu} = \frac{UD}{\nu} \tag{3.7}$$

Where ρ is density of the fluid, *U* the velocity. *D* is characteristic length, which in the case of pipes is the pipe inner diameter. μ is the dynamic viscosity. When dealing with calculations of heat transfer, especially μ can cause some difficulties. As temperature differences are introduced the fluid will have a decreased viscosity for temperature rise, and increased viscosity if the temperature is decreased. This behavior is nonlinear. This can be coupled in CFD but it should be mentioned as the nonlinearity may cause some difficulties for numerical calculations [18].

3.3 Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the analysis of systems involving heat transfer, fluid flow and other similar dynamic processes using computer-based simulation.

CFD is based on the governing equations of fluid dynamics. The laws that are adopted to create these equations are [19; 20]:

- Mass is conserved for a fluid
- Newtons 2nd law; the rate of change of momentum equals the sum of forces acting on the fluid
- Thermodynamics 1st law; the rate of change of energy equals the sum of rate of heat addition and the rate of work done on the fluid.

While performing numerical calculations, most of the simulations were done with turbulent flow. When solving for turbulent flow for this thesis COMSOL 5.4 was set to use Reynolds-Average Navier-Stokes equation [21].

$$\rho(\boldsymbol{U}\cdot\nabla\boldsymbol{U})+\nabla\cdot(\mu_T\left(\nabla\boldsymbol{U}+(\nabla\boldsymbol{U})^T\right)-\frac{2}{3}\mu_T(\nabla\cdot\boldsymbol{U})\boldsymbol{I}\Big)$$
$$=-\nabla P+\nabla\cdot(\mu\left(\nabla\boldsymbol{U}+(\nabla\boldsymbol{U})^T\right)-\frac{2}{3}\mu(\nabla\boldsymbol{U})\boldsymbol{I}\Big)+\boldsymbol{F} \qquad (3.8)$$

No further explanation will be given on the equation but a simplified explanation is given by Persson and Nilvé [22].

3.3.1 Modelling

When modelling for CFD there are plenty of parameters, boundary conditions, initial conditions etc. that need to be included in the study. Boundary conditions applied to the model may be seen in table 3.1.

While setting up which solver used for turbulent flow it is important to think about the model setup. For example, for flow calculations there are different boundary layers at different distances from the walls in which the fluid flow. So closest to the wall there is a viscous sublayer and on top of this there is a buffer layer and on top of this a log-law layer [23]. This means that a very fine mesh would be required to do simulations close to the walls of the HEX. To come around this very time-consuming modelling one can use a wall function. One of these wall functions is the k-ε method.

The k-ε method is the most widely used in engineering applications. k is described as the turbulent kinetic energy while ε is the dissipation rate of the turbulent energy [22; 24].

Solver settings are important for modeling CFD simulations. COMSOL has recommended solver settings for the physics applied to the model and these were followed for the most part with some smaller deviations. To hasten the process and be able to validate the model, it is possible to change numerous settings. The easiest to change for faster simulations is relative tolerance which can be lowered, with the consequence of a less reliable simulation. Auxiliary sweep may also be added for study extension, which is a method to get around the problem of non-convergence cause by nonlinearity mentioned in 3.2.3

Table 3.1 Boundary conditions applied to the model.

Physics	***Fluid properties***	***Pressure***	***Wall conditions***	***Inlet/Inflow***	***Outlet/Outflow***	***Heat Flux***	***Thermal Insulation***
Turbulent flow shell (spf)	Ht dependant	1 atm	No slip	User defined constant	0 Pa	-	-
Heat transfer fluid shell (ht) T	spf dependant u & p	1 atm reference	-	361.15 K	-	spf p & ht2 T, cylinder in crossflow	-
Heat transfer solid (ht2) T2	-	-	-	-	-	T outer pipe & baffle, T3inside pipe	Outer shell
Turbulent flow pipes (spf2)	ht3 dependant	1 atm	No slip	User defined const	0 Pa	-	-
Heat transfer fluid pipes (ht3) T3	spf2 dependant	1 atm	-	393.15 K	-	spf2 & ht2 T, internal forced convection	-

Grid independence study is a method of evaluating the validity of a simulation. A grid independence can and should be performed to validate that the solution reaches the same monitoring conditions independent of mesh applied to the model. To perform an independent study, one refines the mesh to see if the model still converges and the monitored conditions reach the same values as previously. If it does, the solution is accurate[25].

Table 3.2 Material properties used when setting up model in COMSOL.

Material properties.	***Aluminum***	***Coolant***	***Oil***
Density [kg/m^3]	2640	1113,2	869
Heat capacity [J/(kgC)]	915	3140	2010
Thermal conductivity [W/(mC)]	110	0,5	0,145
Dynamic viscosity [Pa·s]	1	0,0161	0,004345
Ratio of specific heat	1	1,1	1,2

4 Results

4.1 Test specimens

The test specimens were designed to be 10 cm long but for technical reason with the print the designs were shortened to 4 cm. As seen at the back of the tube farthest to the left there was a problem with printing and the specimen got damage. This can also be seen at the bottom right corner of the slit profile in figure 5. Some warpage was present as a result of other parts in the print.

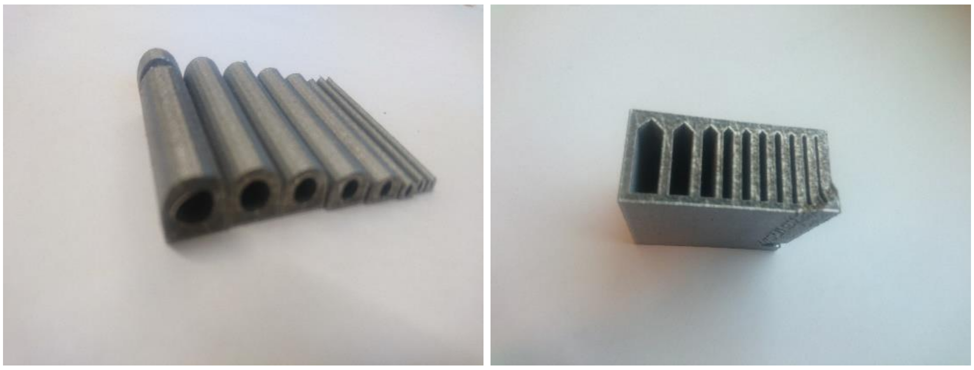

Figure 5 Lenticular shaped test specimens (left) and vertical slits (right).

Figure 6 Polygons and twisted pipe.

In figure 6 the test for the more complex geometries such as polygons and banana shape can be seen. The "banana" (top right in figure 6) is very narrow, and it is difficult to see if the powder was removed sufficiently in this test. Warpage can be seen in this specimen as well as for the slit specimen.

4.2 Design

There was not much to go on when this thesis was conducted so some initiatives had to be taken to get the project moving forward. Firstly; only some information about the current heat exchanger used in the Volvo Truck was given, so assumptions had to be made. From literature some help was found, as mentioned in the theory section, of what is a compact HEX is. Some innovative thinking was needed as not a lot of information was given. One idea was to do rough calculations of what was needed to reach the A/V > 1100 $\frac{m^2}{m^3}$ ratio. Considering the specification to reduce the volume by 25% and using an arbitrary volume, there now was some direction to continue the design process. Firstly, some rough calculations were made to see where optimizations were needed. The volume was set to 50x100x200 mm making the volume 1 dm^3. After this an area (making the HEX compact) could be calculated to 0,825 m^2. When calculating on a S&T heat exchanger, not including baffles, and with the inner diameter set at 5 mm and wall thickness at 1 mm, it was found that

188 tubes were needed. The distance between the pipes were set to 3 mm to see how many of these pipes could fit in the volume and the result was a mere 50 pipes. Even though a S&T HEX cannot be considered a compact HEX this was quite far from the target.

$$V_0 = 0{,}1 \cdot 0{,}2 \cdot 0{,}05 = 0{,}001\ m^3 = 1\ dm^3 \tag{4.1}$$

$$\frac{A}{0{,}75 \cdot V_0} > 1100\ \frac{m^2}{m^3} \implies A = 0{,}875\ m^2 \tag{4.2}$$

$$A_{pipe} = \pi \cdot d_0 \cdot L = \pi \cdot 0{,}007\ m^2 = 0{,}0044\ m^2 \tag{4.3}$$

$$n_{pipe} \cdot A_{pipe} = A \iff n_{pipe} = \frac{A}{A_{pipe}} = 187{,}6\ pipes \tag{4.4}$$

The final design of the simplified HEX was chosen as a Shell-and-Tube with an inner diameter of 100 mm and a length of 200 mm. The reasoning behind this was to keep it simple when designing as the design is done with arbitrary parameters there was no reason to complicate it. It was also done in the hopes to make the Shell-and-Tube compact or at least closer to it. Dimensions for the conventional Shell-and-Tube are given in table 4.1. As seen from the table the design was a decent as Shell-and-Tube HEX usually fall in the range of 70-500 $\frac{m^2}{m^3}$. It could be made more compact just by adding a more pipes, which would be possible, but for the purpose of this report it was not considered necessary for the comparison.

The area and volume are calculated as follows:

$$A_{pipes} = N_{pipes} \cdot \pi \cdot d_0 \cdot L \tag{4.5}$$

$$V_{shell} = \pi \cdot (\frac{D_0}{2})^2 \cdot L \tag{4.6}$$

Table 4.1 Conventional HEX dimensions.

Geometries/dimensions	***abbreviation***	***mm***
Shell outer dia	Do	110
Shell inner dia	Di	100
Total Length	L	200
Baffle thickness	th	2
Baffle distance	BD	26
Baffles number	Baffles	6
Pipe outer dia	do	5
Pipe inner dia	di	3
Pipe Length	Lpipe	200
Ring A radius	-	11.1
Ring B radius	-	22.2
Ring C radius	-	33.3
Rind D radius	-	44.4
Mid pipe nbr	-	1
Ring A nbr	-	6
Ring B nbr	-	12
Ring C nbr	-	20
Ring D nbr	-	30
Area [m^2]	-	0.21677
Volume [m^3]	-	0.00190
A/V	-	114.0596

The design for 3D-printing needed to take print direction into consideration to avoid support material as mentioned in theory. The design can be seen in figure 7. The reason for the conical baffles is the angle aspect when printing.

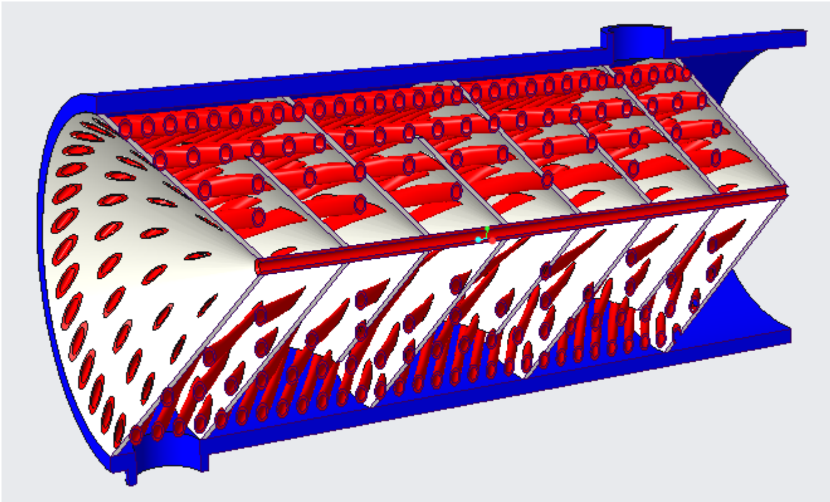

Figure 7 Cross section view of the complex geometry heat exchanger

When analytical calculations were done, using equation 3.9 and 3.10, for the HEX meant for 3D printing the result showed an increase in area with roughly 24 % compared to the standard HEX with straight pipes. Going back to equation 3.1 this is obviously good regarding heat transfer. The results of the calculations can be seen in table 4.2.

When these values are obtained the area and volume is solved according to equation 4.5 and 4.6.

Table 4.2 Analytical calculations of complex HEX for 3D-printing.

Pipe dimensions	***mm***	***Pipe area [m^2]***	***Total area/pipe sets***
a	2		
B	3		
Pipe d	341	0.0054	0.162
Pipe c	288	0.0046	0.091
Pipe b	243	0.0039	0.046
Pipe a	212	0.0034	0.020
Mid pipe	200	0.003	0.003
Area [m^2]	0.358		
Volume [m^3]	0.002		
A/V	**170.146**		
Ratio conventional/3D	**1.243**		

Comparing mass and volume of the material for conventional and complex HEX is seen below. Since the material of use is aluminum and the material in the original HEX (appendix B) is steel, a quick comparison was made for mass properties. It was chosen to do the calculations as if the conventional HEX was steel and the complex is made from aluminum.

Table 4.3 Calculations of mass for different materials.

Units	
$\boldsymbol{\rho_{Stainless\ Steel}}\ [\frac{\boldsymbol{kg}}{\boldsymbol{m^3}}]$	7744
$\boldsymbol{V_{conventional\ HEX}}\ [\boldsymbol{dm^3}]$	0.571
$\boldsymbol{m_{SS,conventional}}\ [\boldsymbol{kg}]$	4.423
$\boldsymbol{\rho_{AlSi10Mg}}\ [\frac{\boldsymbol{kg}}{\boldsymbol{m^3}}]$	2680
$\boldsymbol{V_{Complex\ HEX}}\ [\boldsymbol{dm^3}]$	0.563
$\boldsymbol{m_{Al,complex}}\ [\boldsymbol{kg}]$	1.509

As seen by the calculations above in table 4.3, the specification to decrease the mass by 50 % should be easy to meet and the difficulty is volume reduction.

4.3 Modeling

When setting up the model in COMSOL, if the modeling process is used as recommended, the first step is to set up parameters and materials. The most important parameter to get turbulence was calculating Reynolds number according to equation 3.7. To get turbulence in the coolant domain and the oil domain Re is thus calculated to:

$$Re_{Oil} = \frac{\rho UD}{\mu} = \frac{869 \cdot 5 \cdot 0{,}003}{0{,}004345} = 3000$$

$$Re_{Coolant} = \frac{1113{,}2 \cdot 2 \cdot 0{,}02}{0{,}01610} = 2766$$

These values are to be classified as weakly turbulent and they were merely calculated to show an estimate of requirements to meet turbulence. By using these values as a pointer, the inlet velocity can be increased to reach similar pressure drops as the compared heat exchanger, seen in appendix B.

The first successful simulation of the heat exchanger for turbulent flow was considered and only for the fluid on the shell side of the HEX. This was a very useful first result as it showed a problem in the design. As figure 8 shows, there is no flow near the outlet (left side of the heat exchanger) for the coolant as the design is thus not adequate. This results in no cooling over the pipes with the hot fluid in this section. A redesign had to be performed, adding an extra baffle to get circulation through the whole shell pass and this can be seen in figure 9.

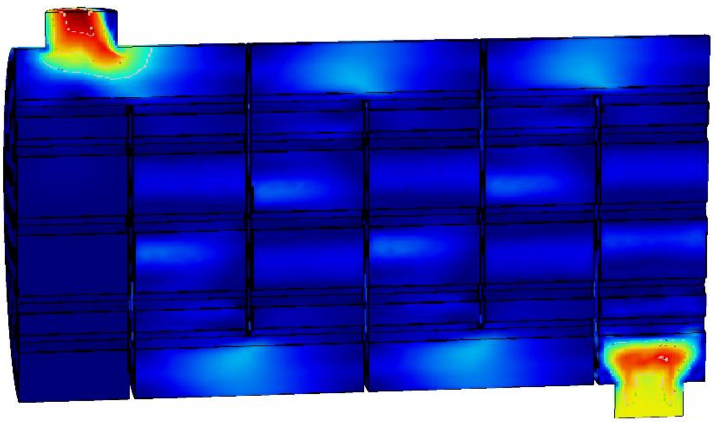

Figure 8 First simulation of simple HEX

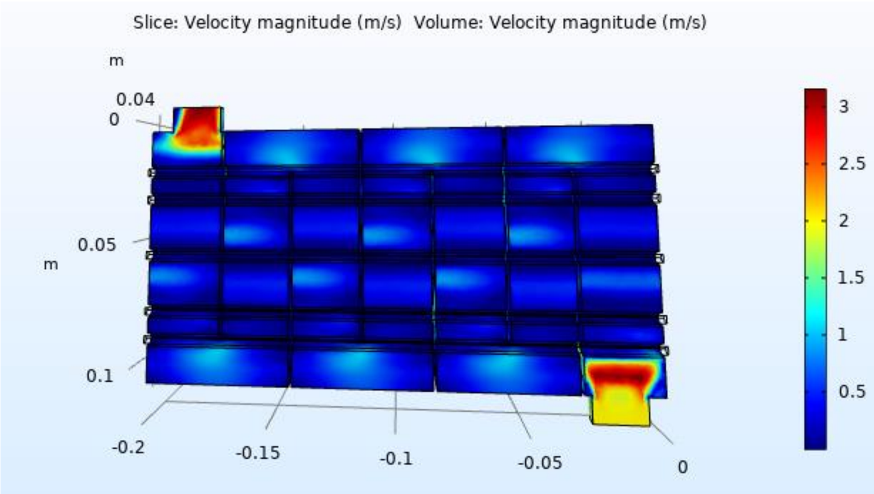

Figure 9 Modified heat exchanger with one extra baffle. The figure shows that there is flow in all regions which was not the case in the first design.

This was, as mentioned, only a simulation of the fluid on the shell side of the heat exchanger so next step was to assimilate the fluid on the tube side. The result of this and the following step to include the walls (baffles, pipes and shell) can be seen below.

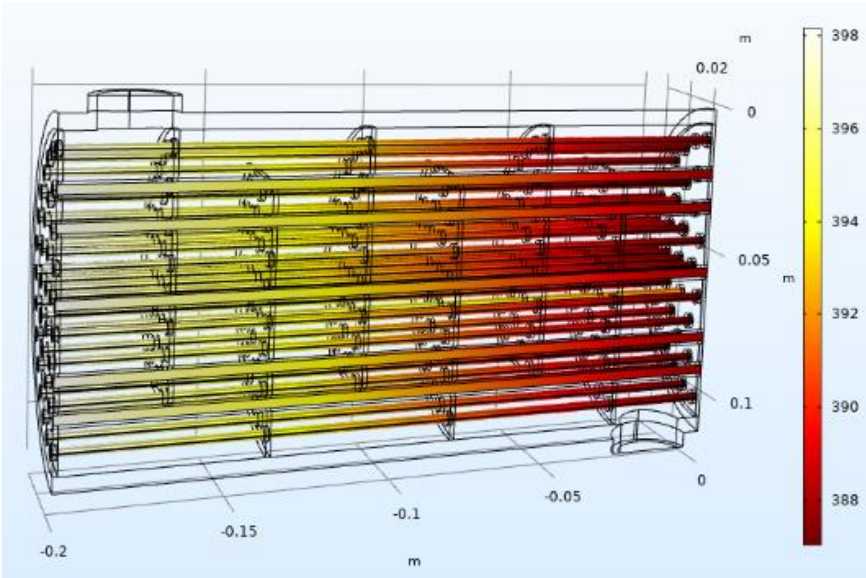

Figure 10 Temperature distribution in fluid, pipe side.

The second to last stage was to include everything except the heat flux (W). The simulation here showed that the pressure drops were similar to the pressure drops from the table of data received, which can be seen in appendix B. The scaling was done from the limited information received, assuming a volume of approximatively 300x150x100 mm. With the parameters chosen for the Shell-and-Tube design the scale factor is roughly 1.5 which would make a pressure drop of 125 kPa for the coolant scale to approximately 88 kPa. Hence the pressure drops received, 90 kPa, seems reasonable.

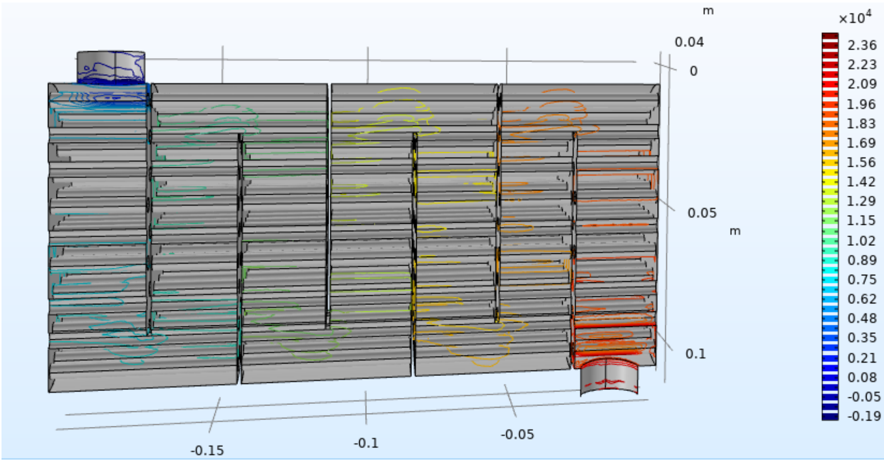

Figure 11 Pressure drop shell side at inlet velocity 2 m/s

The pressure drop on the pipe side was a bit lower than that of the data table and so simulations were run where inlet oil velocity was increased. The tolerance for the solver was also lowered so that the solutions would quicker simulation. This caused problems with convergence for the turbulence variables, especially for tube side turbulence, which was not converging. A solution to convergence issues is suggested in discussion.

The simulations that resulted in the pressure drops closest to that of the data received can be seen in figure 11. This corresponds to the simulation with the high inlet velocity for pipe side.

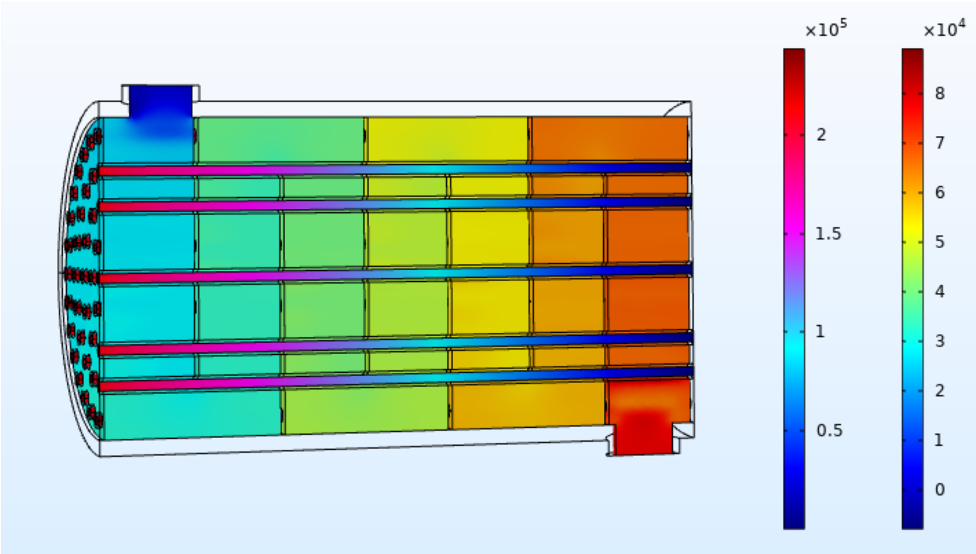

Figure 12 Pressure drop pipe and coolant for high velocity simulation.

The temperature distribution in figure 13 shows good temperature difference for the oil but the temperature rise in the coolant is very low. Thus, adding more pipes would not be a problem.

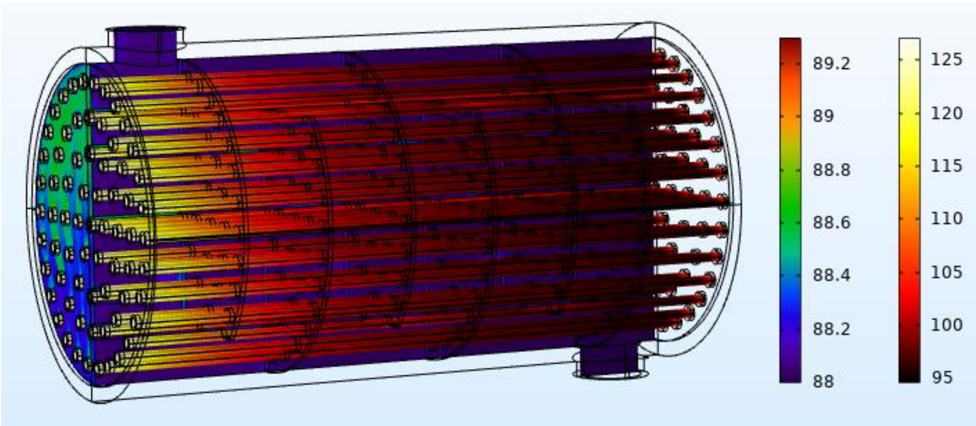

Figure 13 Temperature for pipe and shell side.

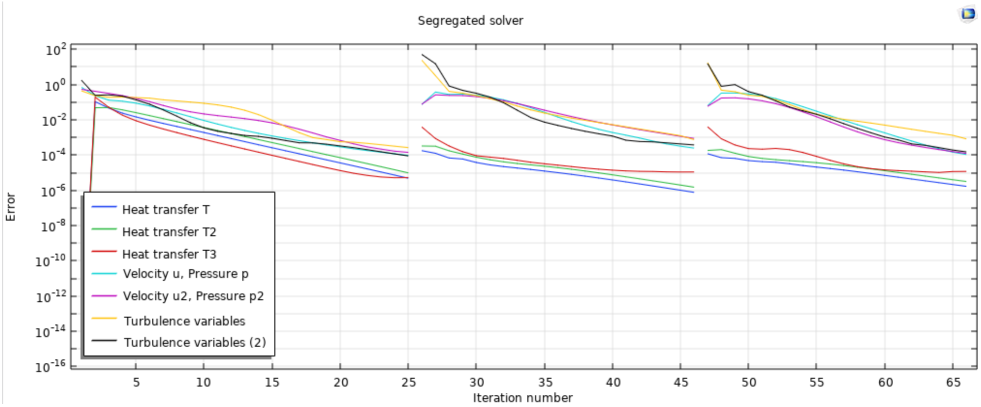

Figure 14 Convergence plot using viscosity ramping

As a last evaluation a grid independence study was performed to see the validity of the simulation. This and the other results can be seen in table 4.4. The simulation marked bold at the bottom is the simulation for grid independence study where it is evident all parameters except pressure in pipes are similar.

Table 4.4 Results of the simulation where inlet and velocity field were user defined.

Velocity field shell (m/s)	*Velocity field tube*	*Inlet vel shell (m/s)*	*Inlet tube*	*ΔP shell (kPa)*	*ΔP2 tube*	*ΔT shell*	*ΔT2 tube*
0.5	0.5	2	5	23.6	92.6	1	3
0.5	0.5	2	1	23.6	7.3	0.3	10
0.5	0.5	3	4	49.7	33.8	0.7	18
0.5	0.5	1	1	7.15	4.41	0.5	30
1	**4**	**4**	**5**	**87.8**	**49.1**	**0.4**	**25**
1	6.5	5	6	134	51.5	1	32
1.5	10	4.5	10	110	122	1	28
1	15	4	15	87.8	241	2.5	16
1	**4**	**4**	**5**	**86.6**	**32.4**	**0.1**	**25**

4.4 Model for 3D-printing

The whole model designed as a comparison to the conventional Shell-and-Tube was not necessary to print to see if the design could be printed. The model was scaled down so that only three helical pipes and the straight center pipe was included for the print. The number of pipes in the inner ring was also reduced to three and a section of the shell was cut to see how the pipes surfaces turned out after printing.

The model can be seen below. Support material was required to be able to print this model and this support is needed at the baffles as the print direction is as the arrow shows in figure 15. The baffles that are adjacent to the cut section of the shell will thus be printed in a void, which is impossible as mentioned earlier.

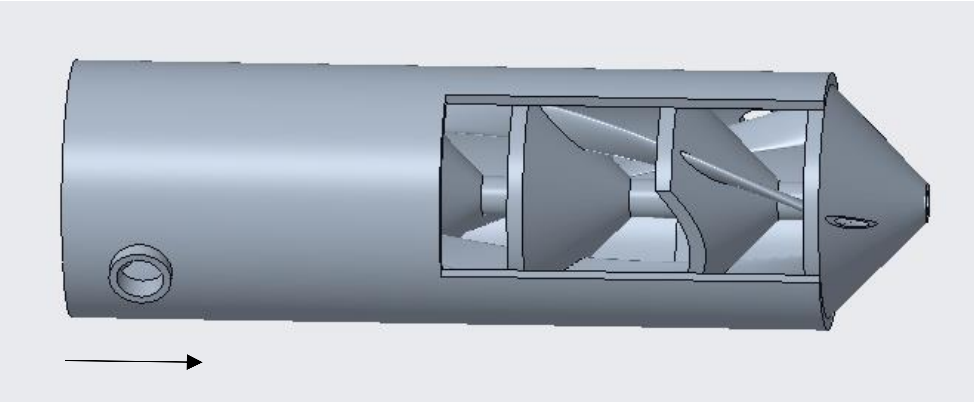

Figure 15 Model for printing. The cut section is to see how baffles and pipes are printed inside the shell. The inner pipes have a narrower passage than the pipes farther from the center, which makes them the more interesting to see printability of. Arrow shows the print direction.

The model was printed successfully and all powder, as could be determined was removed satisfactory. Even for the small diameter of the helical pipes the powder was removed easily, which was not the case in a plastic reference model that was printed. Some manual post printing work needed to be done to remove support material, but this post processing would not be needed as the cut-out wall section is not present in the design actually thought to be used. The surfaces were not a problem either, except for where mentioned support was present.

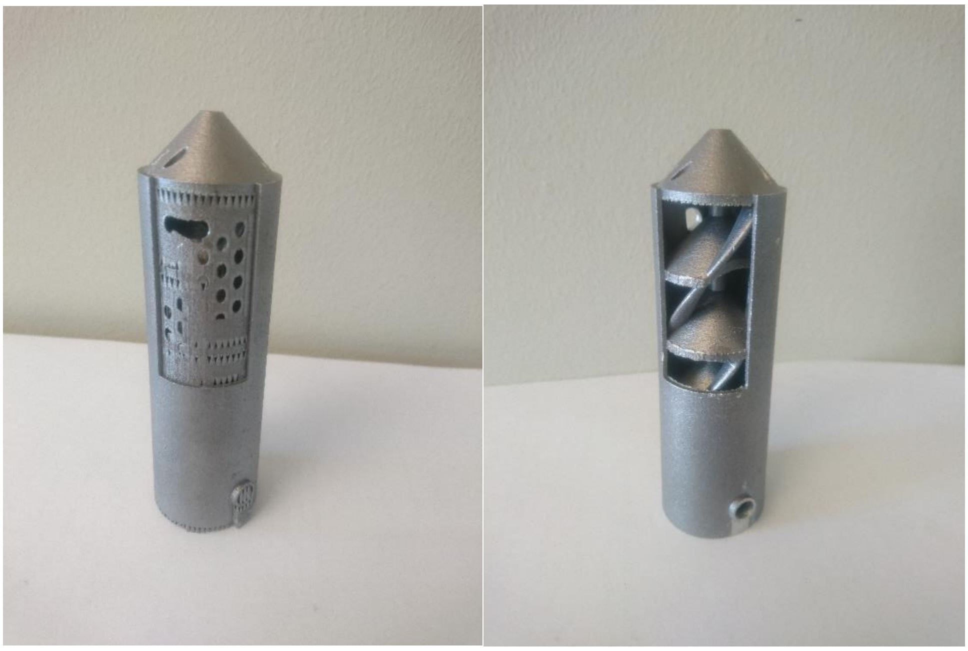

Figure 16 The left figure is as printed with support material and to the right the post processed model.

Due to time constraints numerous simulations were not possible to run on the large complex HEX with 69 pipes. Since it needed to be scaled down it would be difficult to compare results with the conventional HEX that most simulations were run on. A comparison simulation was instead done on another simple model with straight pipes, also with just four pipes like in the complex scaled down HEX.

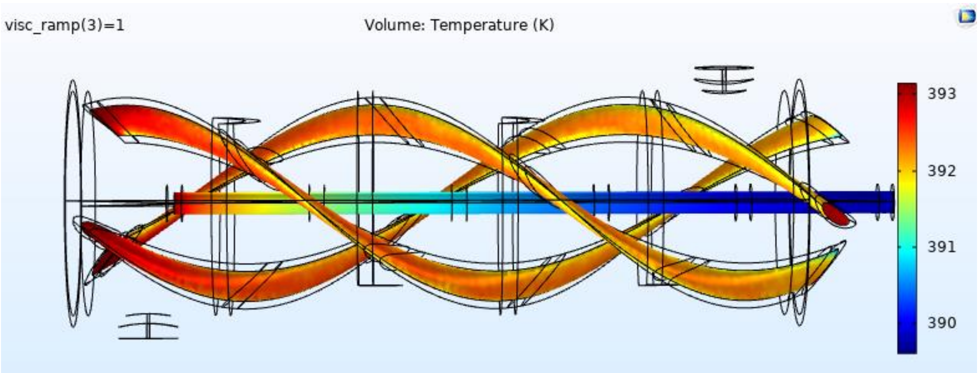

Figure 17 Temperature difference pipe side for complex heat exchanger.

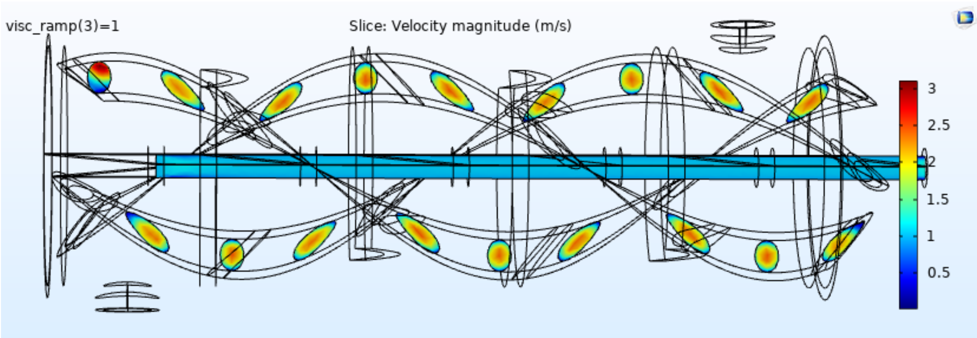

Figure 18 Velocity profile showing velocity ramping up considerably more in twisted pipes compared to the straight middle pipe.

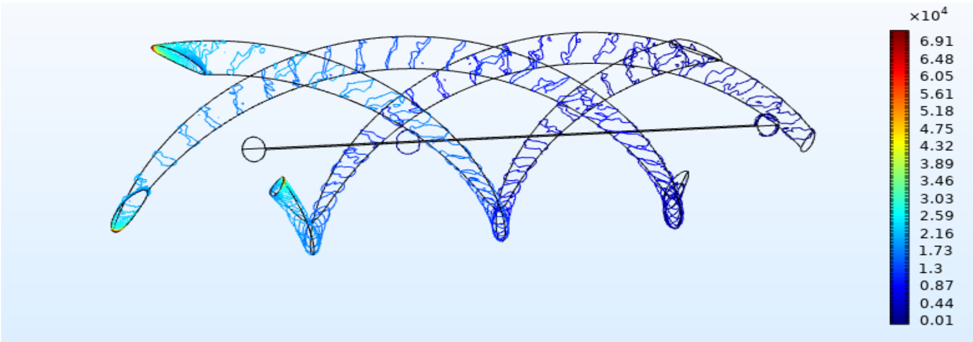

Figure 19 Pressure drop in pipes.

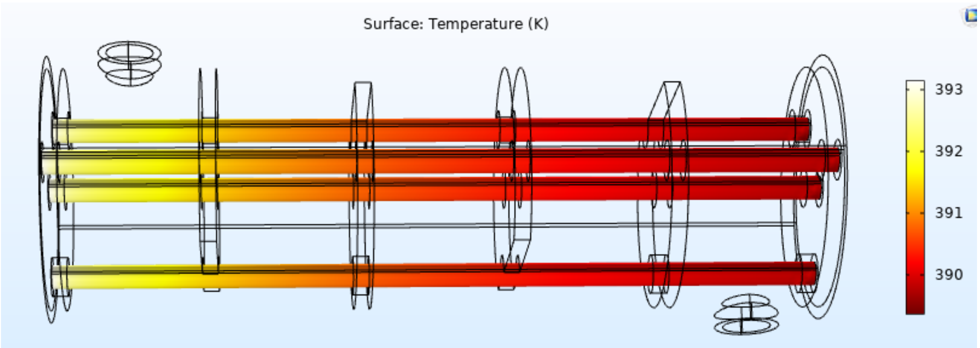

Figure 20 Temperature drop conventional.

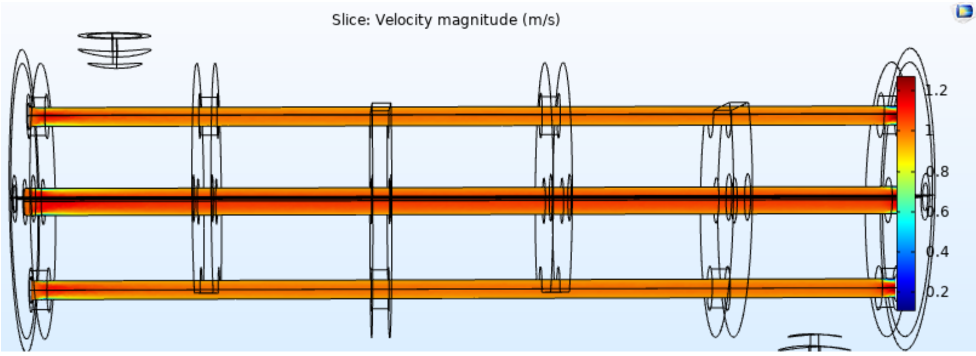

Figure 21 Velocity profile conventional.

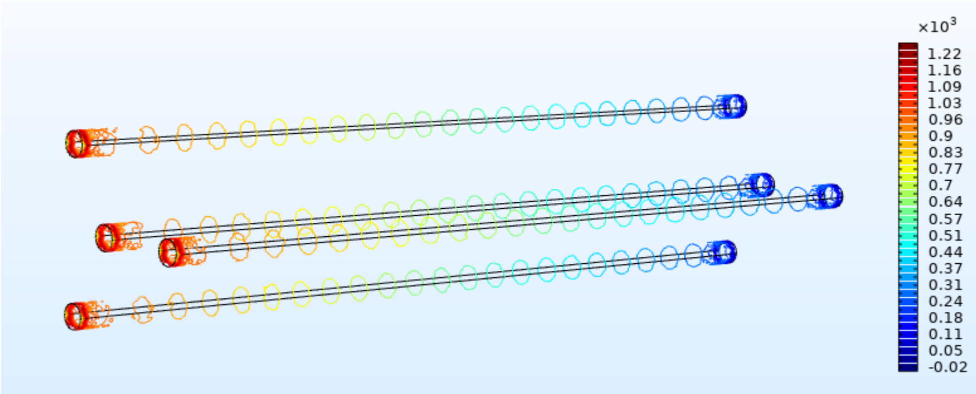

Figure 22 Pressure drop pipes for the smaller HEX compared with the smaller complex.

Table 4.5 Number of elements for the different simulations.

	Number of elements
Most simulations conventional HEX	2 989 049
Independence study	7 781 333
Small conventional HEX	1 215 263
Complex HEX	751 859

Notice how the number of elements for the complex HEX is fewer than for its comparable model. The mesh settings were set to the same quality.

5 Discussion

5.1 Discussion on the methods used

AM is with no doubt a very interesting and useful manufacturing technique. Though AM at present day is a relatively slow manufacturing technique, it definitely can be worth using for complex or lightweight parts and tools. For low production volumes it is also the cheaper option, compared to having a site for production of various components. It is possible it can be worth the cost for manufacturing of heat exchangers. The energy cost just to press plates for a plate heat exchanger is huge and this could definitely be reduced with AM. As mentioned earlier, there is research being done in the field and it progresses constantly. It is of common use in space and air industry where weight is of utmost importance.

Simulations with COMSOL or any other similar software is a great tool to evaluate a design or construction. However, just like experiments, there are a lot of reasons to discuss results received.

First, the tolerance set when importing files can be a source of faulty results and the mesh can also be a strong source of error. When meshing one wants to have the most efficient mesh while still having the most accurate results. For the case of a HEX the mesh in the solid might be the least important to have a good quality mesh on since the calculations for heat transfer by conduction through the solid is less difficult (less computational power needed) than that of the fluids, especially for turbulent cases. When the fluids are introduced the mesh will become much more complicated as solids and fluids behave differently. Boundary layers near walls need to be created in several layers for a good simulation due to the difficulty of calculating velocity profile, as mentioned in *3.3.1*.

At the test simulations the layers were reduced to only three boundary layers to quicken the calculations. The relative tolerance was also reduced from 0.01% error to 0.1% error to quicken the process. As has been mentioned, by reducing the relative tolerance, one chooses to allow greater margin of error. The tolerance was increased again to 0.01% to get the most accurate results when the simulations worked properly.

5.2 Test specimens

As mentioned in the theory section for SLM some controversies arose regarding narrow passages/channels and how easily the powder could be removed after printing. To get a better idea of the capabilities of additive manufacturing some test samples were created with CAD and printed. The design of the test specimens was chosen to see how the powder could be removed and to see how the non-circular geometries would be printed. When designing it was important to consider the guidelines mentioned by A. You et al. [12]. Besides wall thicknesses it was important to consider how surfaces in the design are oriented regarding the print direction. Printing a horizontal surface will not be possible without support material and printing anything below a 45˚ angle will result in very rough surfaces.

This information was relevant as the design of the HEX was built on the findings of the test specimens printability. It gave a guideline for how small the pipe inner diameter could be made.

Unfortunately, when the test specimens (figure 5 & 6) were created, they were made only 40 mm long as mentioned in results. This was due to technical reasons which was not given, and it left the results not as good of a guideline as hoped. It was, however, interesting and good to see how easily the powder could be removed even for the very narrow passages below 1 mm. There was also a problem with the test specimen with the more complex hole geometries. Two of the other parts that were printed at the same time got warped, probably due to thermally introduced residual stresses added in the process. This resulted in the scraper being damaged and powder not applied evenly on the parts after one layer was melted. Still, it gave a somewhat better idea of the printability of the SLM machine.

5.3 Design

The design for the conventional shell & tube was done as a model to have something to compare with. There were no real difficulties in choosing the design as simplifications were made and not all features that could be included in a Shell-and-Tube were accounted for. For example, there is no inlet added for the oil and the oil was instead modeled as inlet being where the pipes started. This might have been a mistake since adding 69 inlets and outlets proved to take some time with all boundary conditions, but the idea was to not add too much to the geometry and through this reducing software error and computational requirements.

As seen by the calculations in the result section the A/V ratio is quite far from the definition of a compact heat exchanger. What was done after the calculations for the conventional Shell-and-Tube heat exchanger was to see how much more area could be obtained by twisting the pipe and making it oval instead of circular. The thought was, that if a Shell-and-Tube could be made compact just by changing geometries, this would be of great interest for further research.

When deciding on the diameter for the pipes to allow powder removal the choice landed on an inner diameter of 3 mm. After talking with Axel Nordin and Jonny Nyman at design sciences, it was decided this should be more than enough. Especially for the relatively short length of the pipes this should not be a problem and the test specimens back that assumption as even for the very narrow 0,5 mm passage, the powder was removed satisfactory. It should be mentioned that the inner diameter of the helical pipes will be less than 3 mm. This is because the pipes twist and depending on distance to center axle of the shell the pipe diameter will decrease the closer to the center it is placed.

The result of the analytical calculations showed increased heat transferring are of around 25 % which according to equation 3.1 would make the complex heat exchanger transfer more heat than a conventional one. In this regard the use of AM to manufacture a heat exchanger is believed to be of interest. This all, naturally, depends on the cost of manufacturing with AM.

Initially, when designing the heat exchanger for 3D-printing, attempts were made free forming the pipe design. This proved to be quite difficult and the CAD software was running rather slow, which is why a more traditional approach felt necessary. Using helical sweep, the same way as one creates a screws pitch, the design was easier to handle. The first iteration was very similar to the conventional HEX, besides pipe geometry. However, with design rules for 3D printing in metal the baffles also had to be redesigned. If the print direction is chosen in the pipes axial direction the baffles would be printed horizontally, which would make the use of support material necessary. If the HEX is tilted 45˚, and printed with that tilt, some parts of the pipes will also, as for baffles previously, be printed horizontally. This is why the conical shaped baffles were created. As for the pipes the pitch was calculated, with a safety factor, so the angle of the pipes would never be above 45˚ of the print-direction.

5.4 Modelling and simulations

The parameters used for the calculations in COMSOL are taken from the table in Appendix B as no other real values could be received. When setting up the model in COMSOL some other parameters are required. Parameters such as ratio of specific heat was impossible to calculate with the information given for the oil and coolant and thus a mean value of typical similar fluids was taken. The parameters used can be found in table 3.2.

Since the interesting HEX is very complex there was a desire to model the simpler, conventional, design first. This was done to learn the software and get a model to compare results with. When modelling for CFD there are numerous of selections to make, such as materials, material properties, boundary conditions, initial conditions, etc. The belief was thus, that finding errors would be easier in a simpler model than going for the complex model first. This proved to be a correct assumption as the simple of the two was still difficult to get results for. Many issues occurred during the modelling process making it quite time-consuming. In appendix C the process is described more closely.

When the model for the last simplified HEX mentioned in appendix C was done it was time to couple the dependent variables together. Since flow will affect temperature and temperature will affect pressure, these three variables (T, u, p) are the variables to be coupled. The simple HEX with results of simulation can be seen in figure 23.

One important variable, which caused problems for the more difficult simulations is the coupling between velocity fields with pressure and temperature. As was discussed earlier COMSOL takes care of the velocity profile close to the wall with wall-functions included in the k-ε method. The information how exactly it is done is limited in the software and it could be a source of faulty results. Choosing the correct values for velocity is thus difficult because it is hard to decide if it is the velocity close to the wall or at the wall. At the wall the velocity is zero and so is the velocity in the solid. This could be the reason why an error occurred when the exact same settings were applied on the larger 1:1 scale model.

The only model that the velocity could be coupled was thus the simplest of the models, with only one pipe. It could be coincidence it worked for the simple model with only one pipe and not the more difficult models because there was no problem when the velocity was set to a constant for the more demanding designs. This result is thus a reason for discussion as a mean value had to be taken after looking at the velocity images for the simulation. The velocity varies and is much higher at baffle bends, just like the velocity for water in a river is higher at bends.

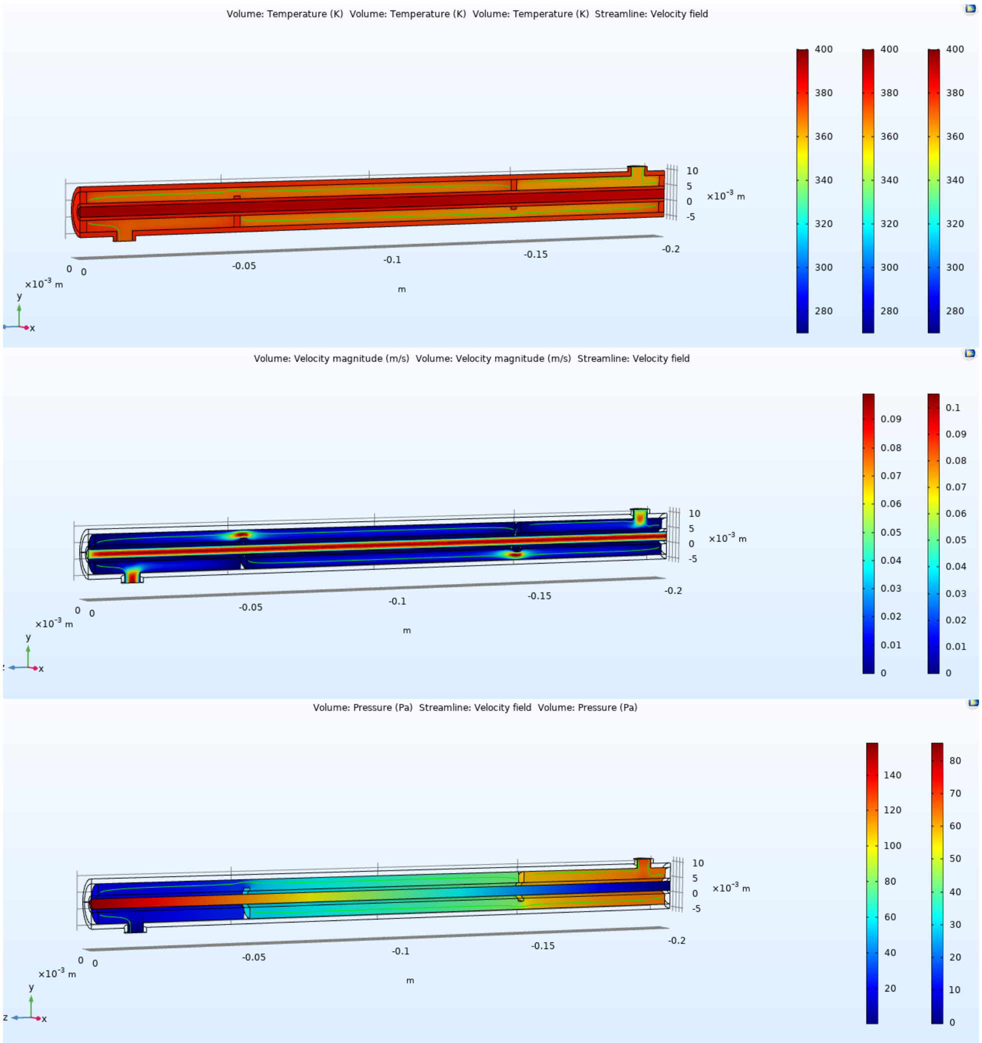

Figure 23 Top: Temperature distribution in Shell-and-Tube. Mid: Velocity profile. Bottom: Pressure distribution. As seen the model is simplified using one pipe and two internal baffles.

When everything else worked (except u coupling) the 1:1 scale model was the focus once again and reasonable results could now be calculated.

A result that was not expected was the very low temperature difference in the coolant on the shell side. The temperature difference from inlet to outlet was only 0,5 Kelvin but the temperature difference in the pipes, where the oil flows, was roughly 10 Kelvin. The largest temperature difference is around two Kelvin. This could be due the very large volume on the shell side compared to the tube side, but it could also be because of the insulation boundary condition set on the shell as it makes the heat transfer to surroundings smaller. Previous results were done with an assumed temperature of the shell, which was not ideal. The best result would come if one could get the surrounding temperature of where the HEX is placed and calculate a heat flux to use on the shell surface, and the second-best alternative is to simulate as thermal insulation, which was done here. Below is a comparison of the temperature simulated with temperature versus insulation. One can see that heat is transferring only slightly into the baffles from the shell and on the surface inside the shell, the rest being at a almost uniform temperature, where with the insulated HEX the temperature is more evenly distributed.

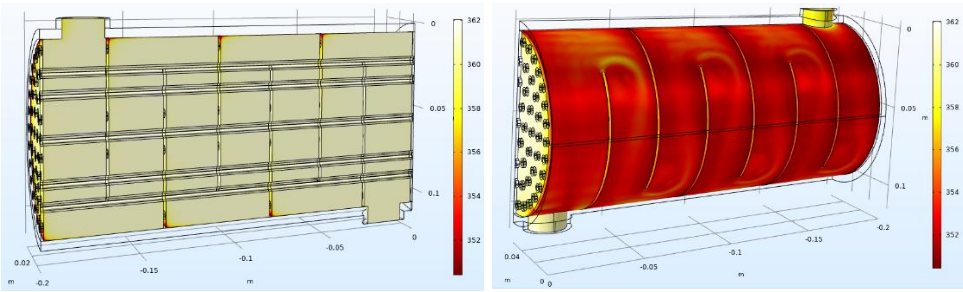

Figure 24 Shell modelled as a fixed temperature

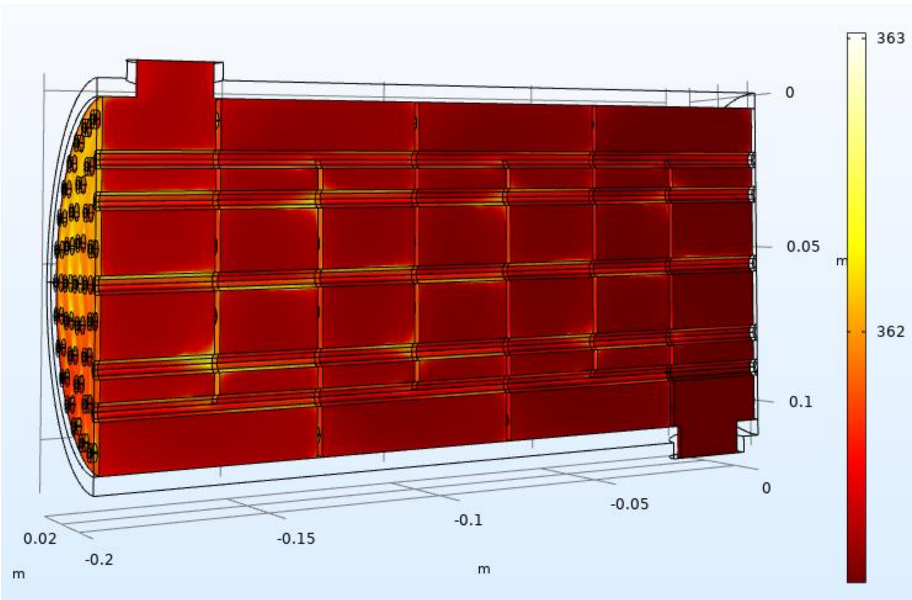

Figure 25 Temperature distribution with insulated shell

Another reason for this temperature could be the thickness of the pipes. If the pipes were made thinner there should be more heat transferred and thus a greater temperature rise in the coolant fluid. If the temperature is accurate, which as mentioned is possible, it shows that the HEX can be made more compact by adding more pipes.

When attempts were made to ramp up the velocity in the pipes and fluid to ensure greater turbulence there were convergence issues. This is a common issue for highly viscous fluids when solving for turbulence and especially the oil dynamic viscosity used is highly viscous. These convergence issues are caused by the nonlinearities in the turbulence model as the *Reynolds-Average Navier-Stokes* equation (3.8) becomes highly non-linear. This can sometimes be solved by viscosity ramping which solves a series of equations, starting at a higher viscosity (lower Reynolds number) and using the solution of the higher Re as starting value for the following iteration. The numerical calculation then works itself down to the correct value of the dynamic viscosity [18]. This technique was applied, and the model could be made to converge.

The non-convergence could also be caused by insufficient mesh resolution or boundary conditions being inappropriate. It could also be caused by sharp corners in the geometry. In this case it is hard to deduce which of these factors was the reason for the non-converging simulation since it could be any or all the stated reasons.

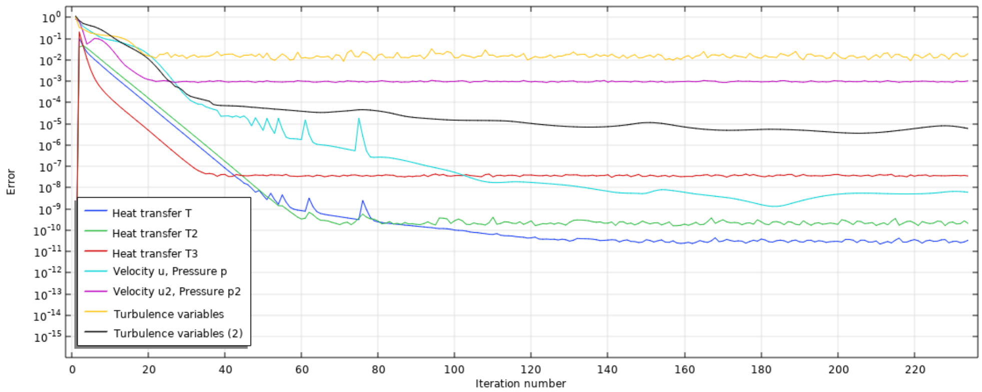

Figure 26 Convergence plot showing how the turbulence variables get close to 1 % error but here encounters instability and the simulation is not converging.

When a mesh was created to validate results of previous simulations it showed that temperature plots and pressure was most likely adequate on the shell side of the HEX. For tube side it did however differ somewhat for pressure, table 4.4, which shows that a refined mesh was necessary for the tubes and oil domain. It might be possible that the mesh required further refinement but considering the mesh independent study took 43 hours to simulate, it was not possible to test more.

The almost non-existing ΔT_{shell} indicates that there is room for optimization. There is room for a higher temperature rise which can be achieved by adding more pipes and/or adding more baffles. Adding baffles will also increase pressure drop which there also is more than enough room for. For higher heat transferred one may also be increased by ramping up velocity, as equation *3.4* shows. Increasing velocity will increase mass flow and consequently heat transfer. One positive thing that comes from low ΔT_{shell} is that the need to cool the coolant is made easier.

Ramping up the fluid velocity could get similar pressure drops as in the HEX tests that were done according to appendix B. The velocity would however have to be ramped up quite a lot for this study and that is why it is not recommended to use for the same application.

5.4.1 **Model for 3D-printing**

It was mentioned in results that mode pipes could be added to make the heat exchanger more compact. This was however not considered necessary to evaluate if the design was good for the intent of 3D-printing.

What was considered more interesting was to see how much more area could be achieved just by changing geometry features instead of adding pipes. Considering the design freedom when using AM an idea arose to twist the pipes in the shell. This is a manufacturing technique not possible, or at least difficult, with conventional methods and it would also show the capabilities of AM as well as adding to the area of the pipes. Another change was to make the pipes oval instead of circular with the same average diameter and thus adding area to the pipe.

The results received on the complex HEX show a large pressure drop on the tube side compare to that of the conventional HEX, figure 19 & 22. As can be seen there is roughly 70 kPa higher pressure drop in the complex model. However, most of this pressure drop is in a corner of the inlet which might be due to a singularity. The more trustworthy result is of around 3 kPa, which is still three times greater than that of the conventional HEX This is most likely caused by the pipes being very narrow and pressure building up and the increased length also adding to pressure drop.

As for the temperature the result shows that there was no real difference between the complex and the conventional HEX. This is most likely because the area increase of the inner ring of pipes is just very slightly larger for the helical compared to the straight designed pipes.

The pitch and distance from center of the HEX being relatively small, and the helix only doing one revolution is the reason the pipes become slimmer the closer to the center of the shell they are placed. As can be seen by the sliced figure 18, the velocity in the twist builds up considerably more than the straight pipe. This causes the fluid to pass through more quickly and creates more turbulence. This should have resulted in more temperature drop but it can not be seen clearly from the results.

The print results were very good, and this was a result of the carefully followed design rules. As mentioned, there was no problem removing the powder and the support material would not be needed in the actual model. Worth considering is if the good surfaces are needed. As was mentioned in theory, a rough surface could enhance heat transfer as it helps create turbulence. Testing this might be difficult with simulations as the rough surface could be tough modelling. It might be easier to tests this with physical tests. Not following the 45˚ angle rule would still print the part but the surface being printed would be very rough. This would have been interesting to test.

When considering the use of the chosen design one can discuss if it is useful for applications such as cars or trucks. One reasons Shell-and-Tube are not used in these applications is because, as mentioned, they cannot be considered compact. It was therefore interesting to see if the Shell-and-Tube design could be made compact. Shell-and-Tube heat exchangers can handle very high pressures better than plate heat exchangers. Many plate heat exchanger manufacturers have welded PHE as well in their stock to make the PHE handle higher pressure ranges.

5.5 Mesh

As has been mentioned, meshing is of great importance when doing simulations and is therefore a parameter worth discussing on its own. Building the mesh with predefined settings is often the easiest and, in some cases, also adequate for results. When the mesh was done for the larger model of the conventional HEX it ended up at around 6 million elements, which is equal to very time-consuming calculations. When building the mesh for the small complex HEX, depending on settings (fine, coarse, etc.) it varied between 300k and 9 million elements.

Besides these predefined sizes one also has the possibility to set multiple parameters individually for a better user-controlled mesh. It was especially around sharp corners that a huge number of elements were automatically created since there were so called singularities. So, user defined mesh was needed, and that is a good thing, since one may control and refine mesh at areas that are estimated to be of greater importance.

As was mentioned regarding grid independence study, a good check when dealing with CFD calculations is to do one or more alternative meshes to see if the results are the same, or close, to what the first simulation gave. That way the mesh is validated to be accurate.

An example can be seen below where one simulation is compared with a different mesh quality. As seen the temperature difference with the first, unrefined mesh (fig. 28), is around 0,5 Kelvin. The simulation with the higher resolution and boundary layers on surfaces shows a more plausible result, as was shown in the result section, fig. 17, where ΔT was around 3 Kelvin.

The use of tetrahedral mesh can also be discussed. For simulation purposes it is the fastest mesh, but it might not always be the best for the different physics. The most important areas are areas close to boundaries and so the mesh to the right in figure 27 is the better mesh as it includes boundary layers.

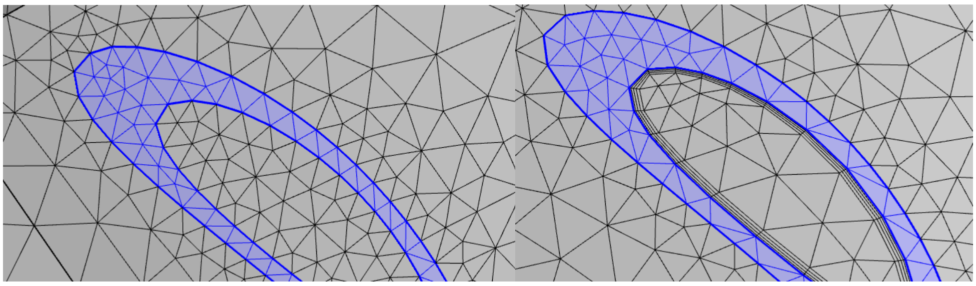

Figure 27 Left figure shows mesh without boundary layers and the right figure is with boundary layers.

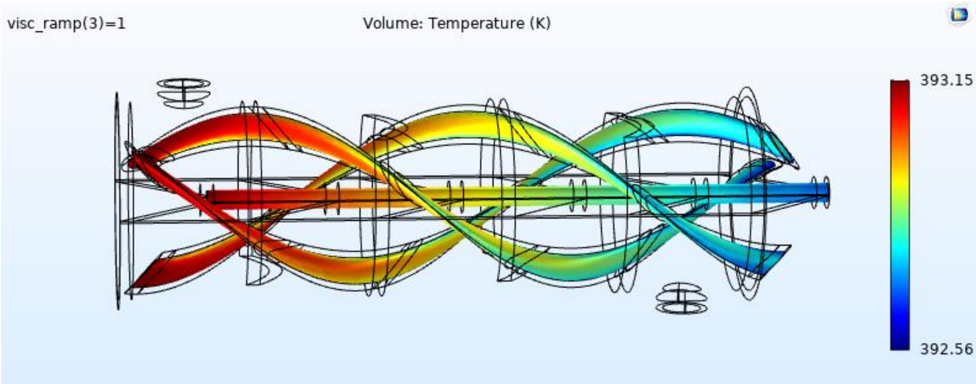

Figure 28 Temperature difference for lower quality mesh. Result of left-hand side mesh in figure 27.

6 Conclusions

The results indicate that using the chosen design should be possible, if not constraints make it unsuitable for a given task. Using additive manufacturing could most likely help reach the goal in the project to reduce the size by 25 % and the mass by 50 % is easy to achieve if aluminum is used instead of steel.

The pressure drops of the conventional heat exchanger were decently accurate to the compared HEX, which was a success. The inlet velocity was however necessary to be increased considerably, which makes the chosen design not the best for the application of cooling engine oil for trucks as much more energy would be needed for pumps. When regarding heat transfer and that heat transferring area is increased quite easily the belief is that a highly efficient HEX could be constructed with the use of AM.

The temperature difference for the two compared scaled down models was very similar. This is most likely due to the area increase of the inner ring of pipes is not nearly as great as that of the outer most pipes.

Unfortunately, it was not possible to try more designs or get a number of simulations on complex heat exchangers. The main limitation in the project was time, and some of the geometries causing numerous problems with software was not expected. A conclusion here is to build the model extremely simple at the initiation phase, and if possible, building from there. This was a good experience development and if a similar project will be done there is a lot to take from the work with this thesis.

6.1 Suggestions for future work

Due to time constraints not all ideas that came up in the beginning of the thesis were possible to explore further. It would have been interesting to design a compact heat exchanger that was based more on the design of present compact HEX, for example a plate heat exchanger. There are different designs that could be of interest, such as cross flow, cellular flow, and adding to area using fins or other protrusions.

If one really wants to model an interesting design, it would also be very interesting to design with regards to biological design. As was mentioned earlier in the report, lungs are one of the most efficient heat exchangers known and although doing an exact replica would be difficult it would be very interesting to designing more towards it. Other interesting design to look at would be that of gills on fish which was an idea that came up earlier in the project.

If someone would like to go further with the choice of design in this report it would be interesting to see how the design could be optimized. How many tubes can be added, how narrow can one make the channels, how thin walls can be printed, distance between baffles, etc. Doing simulations to find the heat transfer coefficient, α, would also be interesting as it would make analytical evaluation easier.

If further work is done on similar geometries and using COMSOL it is recommended to try smaller shapes from the beginning. A lot of time can be saved for simulations if the geometry can be easily set up in COMSOL. Consider using Boolean operations in CAD software before exporting and importing the assembly to simulation software. Just defeaturing the relatively small complex HEX took a very long time. It would also be interesting to simulate vibration in the Shell-and-Tube to get an idea of the structural stability.

To verify that all powder is removed completely some ideas came up. Firstly, if the simulations are believed to be accurate, one can do physical tests and see what pressure drops come from these tests. Another way to see could be to collect all powder with a fine mesh and simply compare the volume with the volume inside the pipes.

A solid mechanics evaluation would also be good as there could be a risk for heat induced fatigue in the material, especially since a heat exchanger cools down and gets heated up repeatedly.

References

[1] Nordin A. Development of test rig and 3D printed heat excchanger [Internet]. . Retrieved 2018 Jan 15, . From: http://www.product.lth.se/education/master-thesis/available-projects/

[2] GE Research and Project Partners Using Metal 3D Printing to Make a More Efficient Heat Exchanger [Internet]. . Retrieved 2019 Apr 19, . From: https://3dprint.com/241712/ge-research-metal-3d-printing-efficient-heat-exchanger/

[3] Hathaway BJ, Garde K, Mantell SC, Davidson JH. Design and characterization of an additive manufactured hydraulic oil cooler [Internet]. International Journal of Heat and Mass Transfer. 2018 . Retrieved 2019 Jan 11, . From: https://doi.org/10.1016/j.ijheatmasstransfer.2017.10.013

[4] Kirsch KL, Thole KA. Heat Transfer and Pressure Loss Measurements in Additively Manufactured Wavy Microchannels [Internet]. Journal of Turbomachinery. 2016 . Retrieved 2019 Jan 20, . From: http://turbomachinery.asmedigitalcollection.asme.org/article.aspx?doi=10.1115/1.4034342

[5] Fee C. 3D-printed porous bed structures [Internet]. Vol. 18, Current Opinion in Chemical Engineering. Elsevier Ltd; 2017 . Retrieved 2019 Jan 10, . From: http://dx.doi.org/10.1016/j.coche.2017.07.003

[6] Li Q, Flamant G, Yuan X, Neveu P, Luo L. Compact heat exchangers: A review and future applications for a new generation of high temperature solar receivers [Internet]. Renewable and Sustainable Energy Reviews. 2011 . Retrieved 2019 Feb 20, . From: https://linkinghub.elsevier.com/retrieve/pii/S136403211100311X

[7] Sundén B. Introduction to heat transfer. Southampton WIT Press; 2012. 344 p.

[8] Ulrich KT, Eppinger SD. Produktutveckling Konstruktion och Design. Fifth. Lund Sweden: Studentlitteratur AB; 2012. 524 p.

[9] DebRoy T, Wei HL, Zuback JS, Mukherjee T, Elmer JW, Milewski JO, et al. Additive manufacturing of metallic components – Process, structure and properties. Prog Mater Sci. 2018 Mar . Retrieved 2019 May 15, ;(vol. 92) p. 112–224. From: https://linkinghub.elsevier.com/retrieve/pii/S0079642517301172

[10] Kempen K., Thijs L., Yasa E., Badrossamay M., Verheecke W. KJP. Process Optimization and Microstructural Analysis for Selective Laser [Internet]. 2014. . Retrieved 2019 Jan 10, . From: https://www.researchgate.net/profile/Karolien_Kempen/publication/262698754_Process_Optimization_and_Microstructural_Analysis_for_Selective_Laser_Melting_of_AlSi10Mg/links/0f31753886ff9bd340000000/Process-Optimization-and-Microstructural-Analysis-for-Selec

[11] Thijs L, Kempen K, Kruth J, Humbeeck J Van. Fine-structured aluminium products with controllable texture by selective laser melting of pre-alloyed AlSi10Mg powder. Acta Mater. 2013;(vol. 61)(5 ed.) p. 1809–19. From: http://dx.doi.org/10.1016/j.actamat.2012.11.052

[12] You A, Be MAY, In I. Design guidelines for laser additive manufacturing of lightweight structures in. 2015;(vol. 14001)(June 2014 ed.). From: https://lia.scitation.org/doi/10.2351/1.4885235

[13] Rafi HK, Karthik N V., Gong H, Starr TL, Stucker BE. Microstructures and mechanical properties of Ti6Al4V parts fabricated by selective laser melting and electron beam melting. J Mater Eng Perform. 2013;(vol. 22)(12 ed.) p. 3872–83.

[14] Herzog PK. Surface characterization in Additive Manufacturing An emperical study of angular dependency [Internet]. Master Thesis. 2018 . Retrieved 2019 Feb 14, . From: http://lup.lub.lu.se/luur/download?func=downloadFile&recordOId=8963731&fileOId=8963732

[15] Kandlikar SG, Joshi S, Tian S. Effect of Surface Roughness on Heat Transfer and Fluid Flow Characteristics at Low Reynolds Numbers in Small Diameter Tubes [Internet]. 2010. . Retrieved 2019 Feb 17, . From: https://www.tandfonline.com/doi/abs/10.1080/01457630304069

[16] Webb RL, Nae-Hyun Kim. Principles of edhanced heat transfer. 2nd ed. New York: Taylor & Francis Group 2005; 2005. 3–10 p.

[17] Sundén B, Wang L, Manglik RM. Plate heat exchangers: Design, applications and performance. WIT Press; 2007. 366 p.

[18] Straccia A. Viscosity ramping [Internet]. 2016 . Retrieved 2019 May 7, . From: https://www.comsol.com/blogs/viscosity-ramping-improves-the-convergence-of-cfd-models/

[19] Tu J, Yeoh GH, Liu C. Computational fluid dynamics A practical approach. Third. Katey Birtcher; 2018. 467 p.

[20] Versteeg HK, Malalasekera W. An Introduction to Computational Fluid Dynamics The Finite Volume Method. Second. Vol. 2. Pearson Education Limited; 2007. 517 p.

[21] COMSOL Multiphysics. Navier Stokes equations [Internet]. January 15. 2015 . Retrieved 2019 May 14, . From: https://www.comsol.com/multiphysics/navier-stokes-equations

[22] Nilvé E, Persson D. Exterior water flow simulation. 2018; From:

http://lup.lub.lu.se/luur/download?func=downloadFile&recordOId=8951011&fileOId=8951012

[23] Simulating turbulent flow in comsol multiphysics [Internet]. COMSOL Multiphysics; 2019 . Retrieved 2019 Apr 15, . From: https://www.comsol.com/video/simulating-turbulent-flow-in-comsol-multiphysics

[24] Launder BE, Sharma BI. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. Lett Heat Mass Transf. 1974 Nov . Retrieved 2019 May 14, ;(vol. 1)(2 ed.) p. 131–7. From: https://linkinghub.elsevier.com/retrieve/pii/0094454874901507

[25] Convergence and mesh independant study [Internet]. 2012 . Retrieved 2019 May 24, . From: https://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/

[26] Perimeter of an ellipse [Internet]. . Retrieved 2019 Mar 10, . From: https://www.mathsisfun.com/geometry/ellipse-perimeter.html

Appendix A Time Plan

A.1 Project Plan

- The differences in literature comes from literature being studied throughout the work with the thesis.
- The difference in design is due to the numerous re-designs that had to be done to be able to simulate.
- Simulations took a very long time, hence the increased time for them and they were performed until the end of the project
- Printing got extended as the printer was serviced and a plastic model was also printed.

Activity	Planned Start	Planned duration	Actual start	Actual duration	Percent finished
Phase 1	1	5	1	11	0%
Pre-Study	1	1	1	3	0%
Litterature study	2	2	2	10	0%
Concept generation	3	3	3	3	0%
Phase 2	5	10	5	15	0%
Design	5	8	5	13	0%
Simulation	7	8	8	12	0%
Phase 3	15	6	15	6	0%
Printing	17	2	15	6	0%
Update lit study	12	2	12	2	0%
Report	15	6	0	0	0%
Populärvetenskaplig artikel	20	1	20	1	0%
Opposition	19	1	19	1	0%
Presentation	20	1	20	1	0%

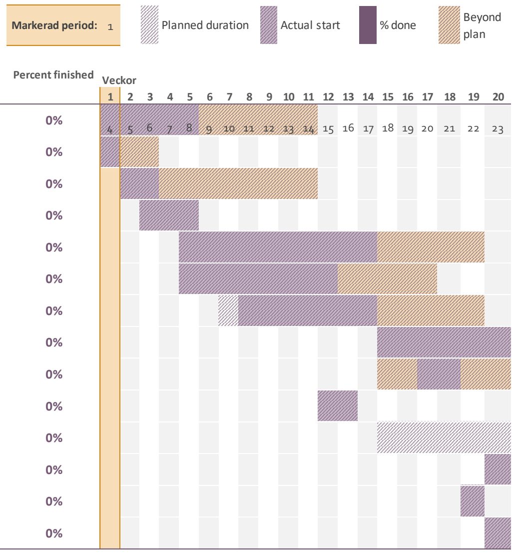

Figure A.1 Project plan and outcome.

Appendix B Current Heat Exchanger

B.1 Current

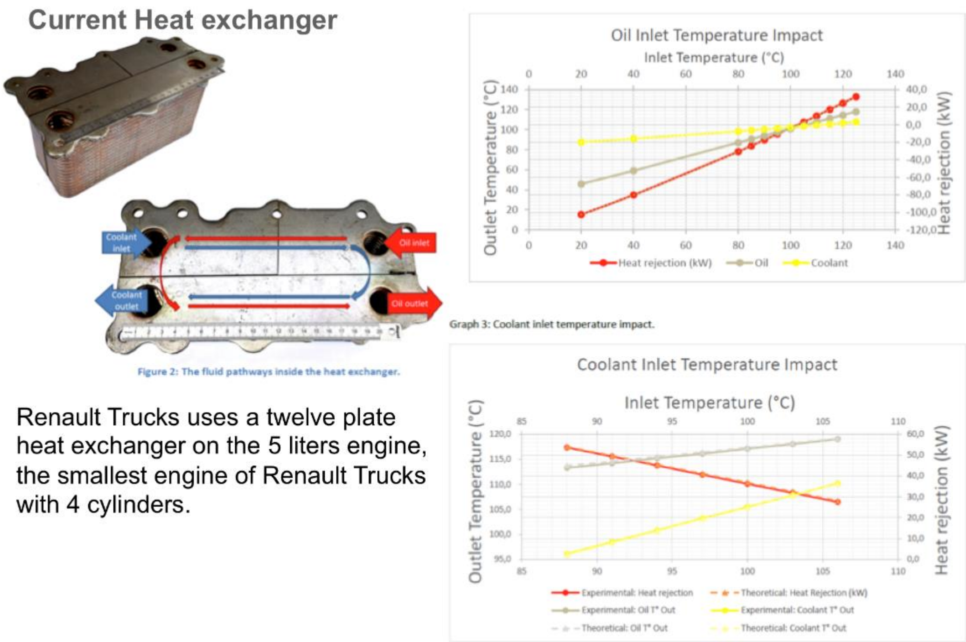

Figure B.1 Image of the current HEX. This were the only dimensions made available.

B.2 Tables

Table B.1 Table of data from tests run on current HEX in volvo trucks.

	Oil Inlet temperature (°C)	Flow (lpm) - Q_{oil}	Pressure drop (kPa) - Δp oil	Oil temperature out (°C)	Inlet temperature (°C)	Flow rate (lpm) - $Q_{coolant}$	Pressure drop (kPa) - Δp coolant	Coolant temperature out (°C)	Heat rejection (kW)
First experience	20	150	495	46	103	110	125	87,5	-102,5
	40	150	414	59,1	103	110	125	90,9	-80,1
	80	150	346	87	103	110	125	98,3	-30,8
	85	150	341	90,5	103	110	125	99,3	-24,2
	90	150	336	93,9	103	110	125	100,4	-17,5
	95	150	330	97,4	103	110	125	101,4	-10,8
	100	150	327	100,8	103	110	125	102,4	-3,7
	105	150	325	104,3	103	110	125	103,5	3,0
	110	150	320	107,8	103	110	125	104,5	10,1
	115	150	315	111,2	103	110	125	105,6	17,4
	120	150	312	114,6	103	110	125	106,7	24,6
	125	150	308	118,0	103	110	125	107,8	31,9
Second experience	125	40	39	112,8	103	110	125	105,3	14,9
	125	70	94	115,4	103	110	125	106,1	20,6
	125	100	164	116,7	103	110	125	106,8	25,3
	125	130	247	117,6	103	110	125	107,5	29,4
	125	160	341	118,2	103	110	125	108,0	33,1
	125	190	445	118,7	103	110	125	108,5	36,6
	125	220	559	119,1	103	110	125	109,0	39,8
Third experience	125	150	320	113,3	88	110	125	96,1	53,6
	125	150	318	114,2	91	110	125	98,5	49,3
	125	150	316	115,2	94	110	125	100,8	45,0
	125	150	314	116,1	97	110	125	103,2	40,6
	125	150	312	117,1	100	110	125	105,5	36,2
	125	150	310	118,0	103	110	125	107,8	31,9
	125	150	308	119,0	106	110	125	110,2	27,5
Fourth experience	125	150	315	120,8	103	20	5	119,0	19,2
	125	150	313	119,5	103	50	27	111,4	25,2
	125	150	312	118,6	103	80	68	109,1	29,3
	125	150	311	118,4	103	90	86	108,6	30,1
	125	150	311	118,2	103	100	105	108,2	31,1
	125	150	311	118,0	103	110	125	107,8	31,9
	125	150	310	117,8	103	120	149	107,6	32,9
	125	150	310	117,5	103	140	202	107,1	34,4
	125	150	309	117,0	103	170	295	106,6	36,5
	125	150	308	116,6	103	200	405	106,2	38,3

Appendix C Method

C.1 Modelling process

The first difficulty when modelling arose while doing the conventional HEX. As COMSOL cannot mesh on voids the void in the pipes and shell side (where there is fluid) had to be modelled in a way so the mesh could be created. Firstly, the whole model was imported but an unfixable amount of errors came up when simulations were attempted, hence a more careful approach had to be applied. It was not clear if the problem was with the CAD model or with COMSOL and this needed to be explored.

In COMSOL it is possible to choose between 9 different mesh quality sizes, ranging from extremely coarse through normal to extremely fine. When the mesh was chosen *coarse,* to reduce computational requirement, the problem occurred that the mesh was too coarse for some of the surfaces to be meshed properly. However, when the mesh was made increasingly fine, the problem with big and time-consuming simulations occurs. It was unclear at this stage where errors were, if it was with the mesh or with boundary conditions being set incorrectly, or the model being corrupt. The result was that for several iterations the simulations would run a while before noticing something was amiss and this wasted a lot of time.

The process had to be moved back a few steps and attempts were made to import a just one pipe and run simulations for laminar flow. When reasonable results were received the study could be expanded and made more complex. Turbulent flow and heat transfer in fluids could be added. The pipe attempts ran relatively well without errors but when attempts were made to simulate flow on the shell some issues came up, halting progress again.

After some evaluation of the progress it was decided that a much simpler model was needed to use to find faulty design easier. The first simplified geometry can be seen in figure C.1.

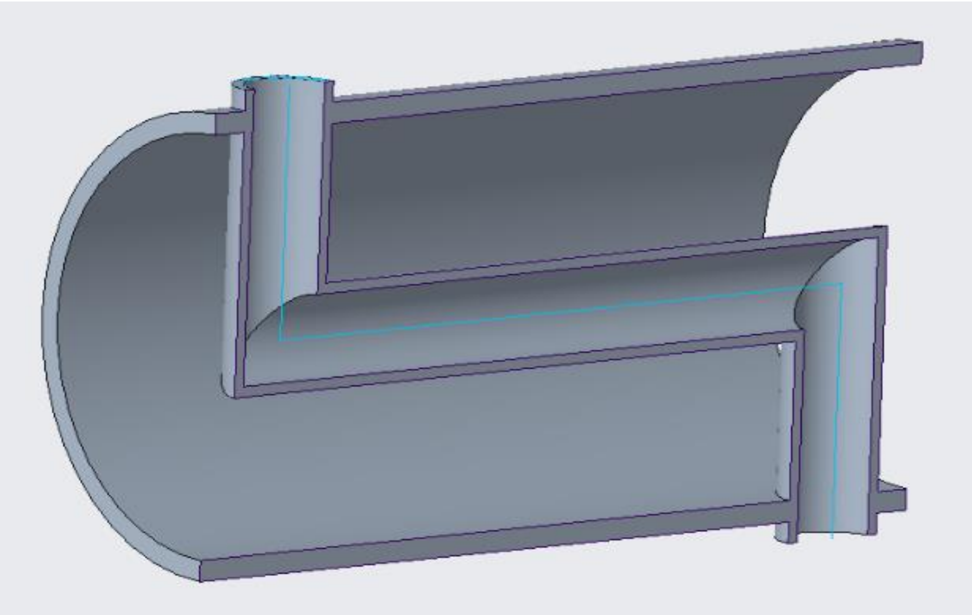

Figure C.1 Simplified shell with tube made for faster simulations and to find errors in design.

This test was a success, so it was concluded there was nothing wrong with the CAD model and the problem lied with COMSOL and boundaries and properties being applied wrong. The process now was to step-by-step continue with the modelling of the conventional HEX and make the simulation more complex.

After managing to do simulations for all domains (coolant, oil, and solid) for the conventional HEX separately it was time to do the couplings between the fluids and solids to get heat transfer. It was made clear the design was made too big to run smooth simulations on. Many attempts were made and sometimes it seemed something was wrong with the mesh while other times something was wrong with the boundary condition selections. It was decided to go back and redesign again to make a simpler heat exchanger to find cause for the errors. This time the design was made to imitate the larger designs better.

This new simple model showed the importance of really scaling down simulations when dealing with CFD. Had the original plan to do a very simple model first been followed better this might have saved a lot of time. This new, much quicker model was far easier to find errors in since the computational cost was cut to less than a tenth. The fluid flows were set to laminar, being the quicker of the fluid flows to calculate, and the rest of the model was done the same way as the complex larger model. It turned out that some of the issues could be solved with boundary conditions being properly set up which was extremely time consuming previously.

Learning from example, the model was slowly built. Since a lot of the problems previously seemed to be due to the heat transfer in the solid, that is where the process started. Once the solid could be modeled, flow and heat transfer in fluids were added gradually.

C.2 Modell for 3D printing

When time came to do simulations there were unfortunately internal errors on CAD kernel in COMSOL. When attempts were made to import as separate parts the error could be seen on the twisted pipes. The surfaces where the pipe and baffle meet are curved, and these were not imported properly. The model was possible to import as .stp file but problems still came up when defeaturing and meshing. To get around this problem another, much smaller, complex HEX was designed. The hope was to see if something was wrong with the old CAD file and if the problem still came up with the curved surfaces the hope was to try and simplify even further and make the surfaces flat at the pipe entrance since this seemed to be the problem. This would make parts of the pipes not be in contact with the coolant, but something needed to be done to get around the problem.

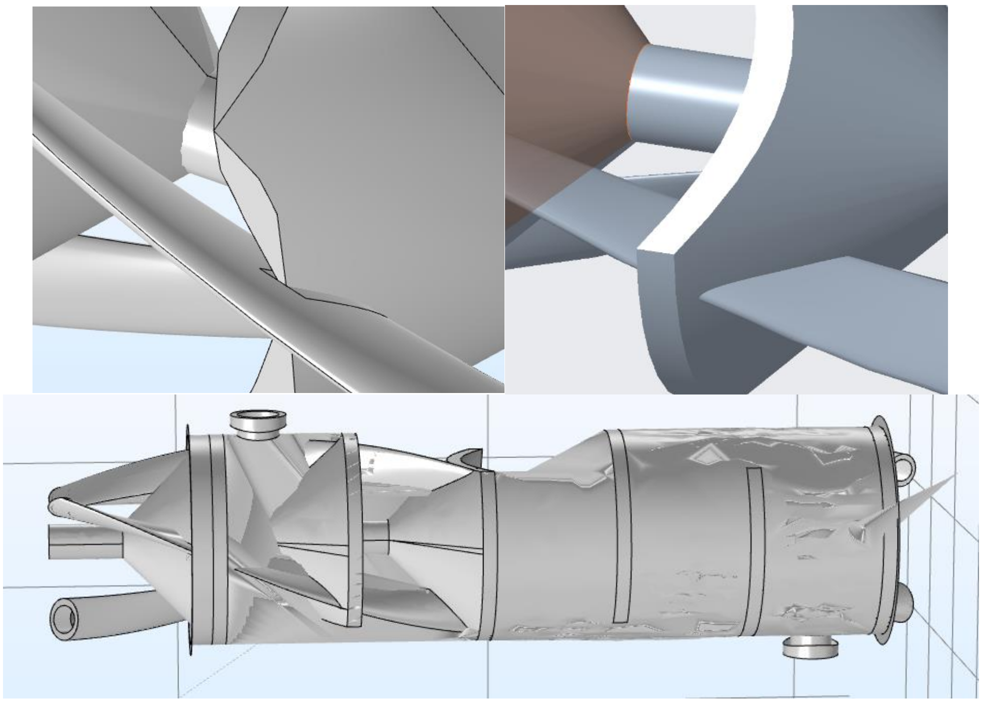

Figure C.2 Example of error with import. Top right is the error that was the result of import, top left is how it should be. Bottom figure is the result of .stl import.

More changes that had to be done to import the geometry was to make a boolean feature of the model. The holes in the baffle were made slighty (.1 mm) smaller to get the pipes and the baffles to intersect and all the model parts were then merged. The problem persisted and it was still because of the curved surfaces where the pipes inlet is assembled to the baffles. A lot of different methods were attempted to get around this persistent issue. The model was made Parasolid (.xt) and this made the model possible to import but still had many errors which made Boolean operations in COMSOL impossible and meshing thus also impossible.

After trying multiple different ways to get around this problem, including converting to stl, stp, Parasolid, iges, etc. the geometry was also changed again to try finding a solution. The pipes were made longer, slimmer, lesser diameter and so on. One solution that worked was to redo the whole design as a solid part, and not an assembly of solids. This fixed the import but made the boundary conditions difficult to set as the baffles could not be modelled separately from the shell. In the end the model was reworked so that the helical pipes in the inner ring (A) had a larger diameter as it seemed the geometry core in COMSOL was interpreting the slim pipe walls as intersecting. The model was then exported as STEP. When the model was successfully imported to COMSOL defeaturing and virtual operations had to be done on surfaces and intersecting edges.

Some positive aspect of this new design is that, like with previous simplifications, was that simulation was quicker. The simulation for the HEX with straight pipes and relatively coarse mesh took 18 hours. This also made the model possible to simulate for much smaller tolerances and thus receiving a smaller error for the results. The HEX for 3D-printing was not possible to section of as there is no symmetry plane, and this is also a good reason to simplify as the whole HEX needs to be modeled. Also, this new design was the design chosen to print as printing the whole 200 mm HEX was not necessary for the study. The interesting results for the 3D printed model was to see if powder could be removed and to see surface roughness.

Appendix D PHE calculations

When designing more in line of what is used in the automotive industry for radiators some calculations had to be done on plate heat exchangers (PHE). A first calculation was done setting the plate thickness to 2 mm and the distance between the plates to be 2 mm. The plates were first considered to be flat, with no pattern, just to see the results of this. This resulted in a required 48 plates to meet the required heat transfer area. Considering the height of the HEX this landed on goal/required ratio of 380% more than what was considered compact. Moving forward, design changes were made to make the plates in a jigsaw pattern, this resulted in the HEX being 280% off target. It should be mentioned that this excludes some geometries that were not drawn up at this stage. If the whole plate would have been drawn up the result would be closer to the target. These calculations were merely done to give a direction of where to move forward and to give an idea of the difficulty in designing a HEX.

As for the dimensions used in these calculations it can be discussed what a good estimate would have been. Much better values of compactness would be received if closer to smallest dimensions printable would have been used for the PHE but when this information came up it was to late in the project to circle back.

It would have been interesting to go further with the PHE design but due to time restrictions and for the sake of argument it was decided to go further with only the Shell-and-Tube heat exchanger as the design was easier to realize.