Article

Design and Simulation Based Optimization of a 3D-printable Induction Heat Exchanger

Janick Beck and Shukang Zhang

Version September 30, 2020 submitted to Processes

Keywords: 3D-Printing, FLM, AMIR, Mixing Reactor, Induction

1. Introduction

Additive Manufacturing, AM, is in fact not a new concept. It can track back to 150 years ago, when people used two-dimensional layer overlays to form three-dimensional topographic maps. During the 1960s and 1970s came the first AM-Technology, include photopolymerization technology, Powder fusion in 1972 and sheet lamination in 1979. But at that time, it has no commercial market at all and very few investment in research and development. [1] The first 3D printer, which used the stereolithography technique, was created by Charles W. Hull in the mid-1980s. [2]

After 30 years development, 3D Printing has come into personal home. The price is nowadays down to 300 dollars.3D Printing, as a bottom-up-process, has many advantages. With 3D printing, designers have the ability to quickly turn concepts into 3D models or prototypes, and implement rapid design changes. It makes development so much easier, quicker and cheaper.

Generally, development steps look like this:

1. Identification of project requirements
2. Computer-aided 3D Model design
3. Simulation of the 3D Model in corresponding physical field
4. Optimize according to the results from Simulation
5. Print real part via 3D Printing and conduct experiments
6. Optimize according to the results from experiment

In this project steps 2. to 4. are used to create CAD models, create a simulation to test those and draw conclusion from the results to improve the CAD models further in preparation of future real life experiments.

2. Problem

In our project we have a pipe (Material: Polymer) up to 300mm long, with a internal diameter of 94 mm. A static heat exchanger needs to be put within the pipe, so that 10 °C hot water flow from the input flows to the outlet and is heated to 80 °C± 5 °C. The temperature distribution should be evenly along the radial direction. Fluid volume is given with 0.5 m^3/h. The heat source will be electromagnetic induction on the heat exchanger. Further less important restrictions can be found in appendix A.

3. Solution

3.1. CAD-Model

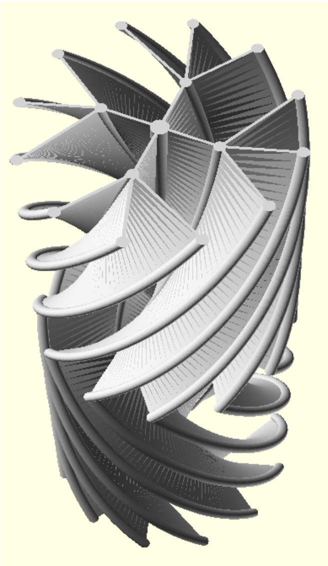


Figure 1. Example CAD model

The idea is to create a helical structure with high parameterizability. The helical structure is chosen for it's ability to create a vortex at the outlet and thereby creating a higher mixing of the liquid in the pipe.

The structure is first created as a two-dimensional drawing which is linearly extruded with a twist. This drawing consists of a number of rings around the center, each containing a certain number of circles. These circles are connected by the shortest path with a circle in the next inner ring. The circles in the innermost ring are connected with the closest other circle from this ring.

This construction allows for the following parameters to be easily changed: Number and distance from center of the rings, number of circles for each ring, twist and the thickness of the edges between the circles. All of those parameters together allow for the structure to have a different density dependent on the distance from the center, thereby being adaptable to the requirements of electrical induction.

After the first simulations it was clear that the structure needs to be longer to have more mass and create more area to transfer the heat. So the structure was mirrored at the top to double it's length and mix the liquid further within the structure with the change in the twist direction.

The CAD modelling script language OpenSCAD [6] was used to create STL files of the model together with Python [7] scripts for batch file generation in the design study. Code and further explanation can be found in the corresponding Github repository [8].

236 different CAD models were created with different bridge thickness, twist and number of circles per ring. To test this amount of structures for their efficiency a simulation was created with Siemens Star-CCM+ and automatically tested.

3.2. Computational Fluid Dynamics(CFD) simulation with Siemens Star-CCM+

Simulation is a powerful tool to check the quality of the designed system and help to optimize the model and the process.

Here in our project it is about Computational fluid dynamics simulation that combines fluid with solid.

When you run a CFD-simulation, the first thing to think about is, that the fluid behaviors laminar or turbulent. First we take a look at a much easier system, water flows direct through the pipe without mixer. Then we can easily calculate the Reynolds number.

$$\mathrm{Re} = \frac{\rho u d}{\mu}$$

ρ represents the density of the water, u represents the velocity of the water normal to the inlet-surface(see figure 3-1), μ represents dynamic viscosity of the water, and d the characteristic length, here i.e. the inner diameter $d = 0.09$[m]. In this case we have fluid volume of $V = 0.5$ $[\frac{m^3}{s}] = 1.389 * 10^{-4}$ $[\frac{m^3}{s}]$, $\rho_{283K} = 999[\frac{kg}{m^3}]$, $\mu_{283K} = 1.3077 * 10^{-3}$[Pa*s]. The value of the dynamic viscosity of water comes from literature [3]. So we get the result $Re_{283K} = 1437$.The critical Re value for a laminar fluid in a pipe is 2320 [4]. Then in order to obtain the press drop when water flows through the empty pipe, we run the simulate under laminar model. Under is the software settings as show in Fig. 3.

Simulation shows a result of 0.16619[Pa]. This simulation was also run again under same settings except that laminar model changed into turbulent model. And we got a result of 0.16722[Pa]. Almost the same result. Later we will compare these results with the results from experiments.

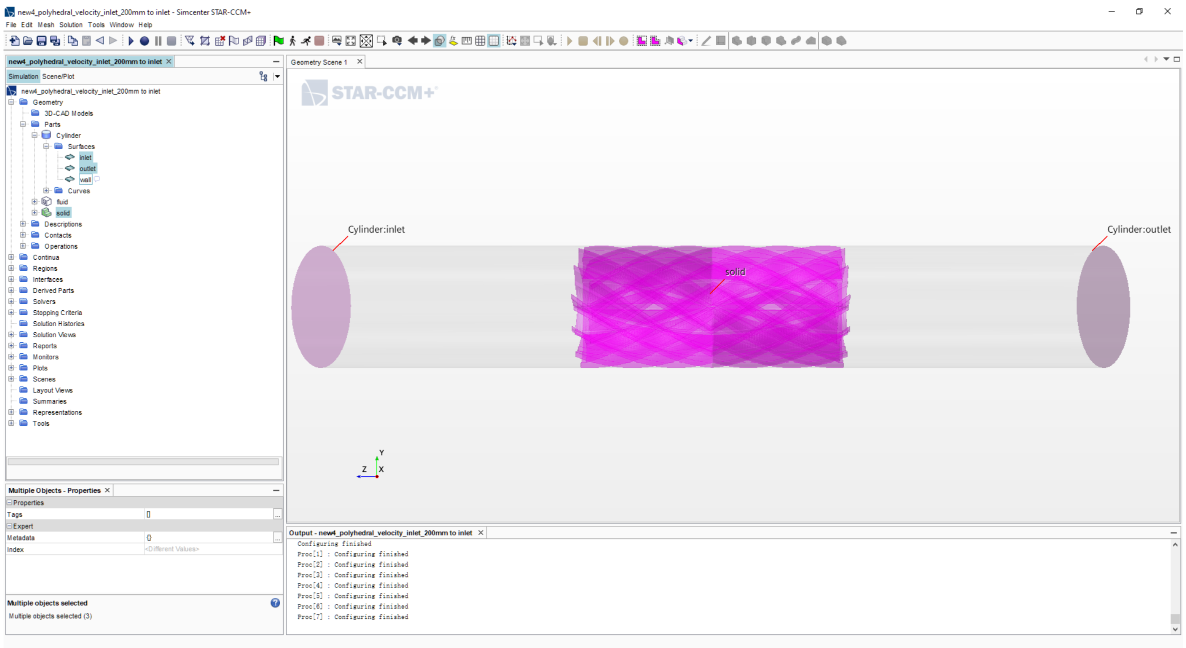


Figure 2. Geometric view of the simulation. Static heat-exchanger in the middle.

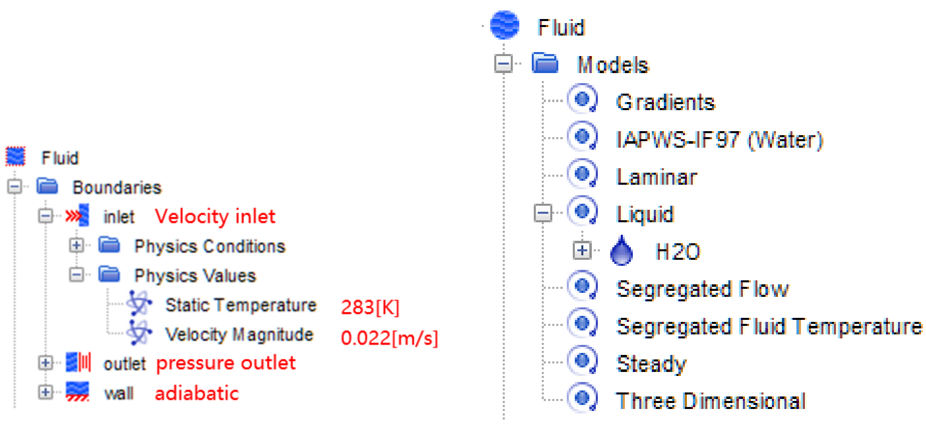


Figure 3. Simulation fluid settings

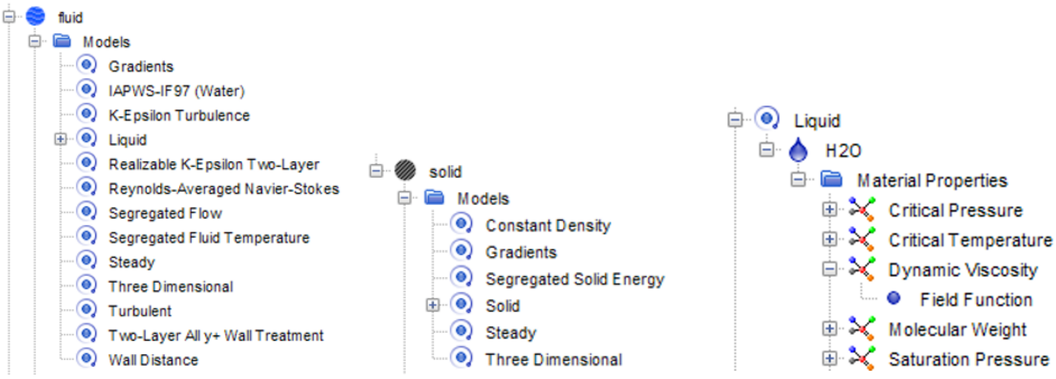


Figure 4. Simulation software Settings

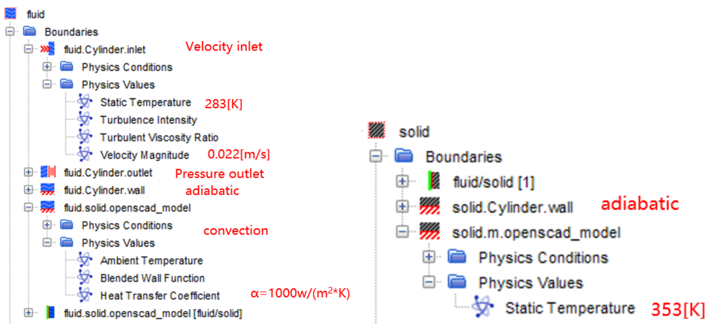


Figure 5. Simulation solid settings

Now we can add the mixer into the pipe. This has influence to the characteristic length. Since the definition for the d is:

$$d \equiv 4 \cdot \frac{\text{fluid passed surface}}{\text{wetted circumference}}$$

So the adding of the mixer will reduce the characteristic length and lead to the reducing of Re. But on the other hand, we want to heat the water from 10[°C] to 80[°C], increasing temperature will reduce the dynamic viscosity very obviously. Under 80[°C] is $\mu_{353K} = 0.3565 * 10^{-3}$[Pa $*$ s], and then $Re_{353K} = 5272$ when it flows through empty pipe. And $\kappa - \epsilon$-turbulence model, which uses Reynolds-Averaged Navier-Stokes-Equation (RANS), include extra terms to describe the disturbance from environment, that cannot be found in normal Navier-Stokes-Equation. So it's obviously better to run our simulation under turbulent model when we add mixer and thermal source. The software settings are shown in Fig. 4.

We use the equation from literature[3] to describe the change of the dynamic viscosity of the water:

$$\mu = A * 10^{\frac{B}{T-C}}$$

Where $A = 2.414 * 10^{-5}[Pa * s]$; $B = 247.8[K]$; $C = 140[K]$.

We plan to use the electromagnetic induction method to heat the water, but in order to simplify the simulation. We give here the mixer a steady temperature of 90°C. as shown in Fig. 5.The heat transfer coefficient between mixer and water is given by $1000[\frac{W}{m^2 * K}]$. It's a typical value from literature [5]. With this base simulation a design study with 216 different CAD-models was conducted.

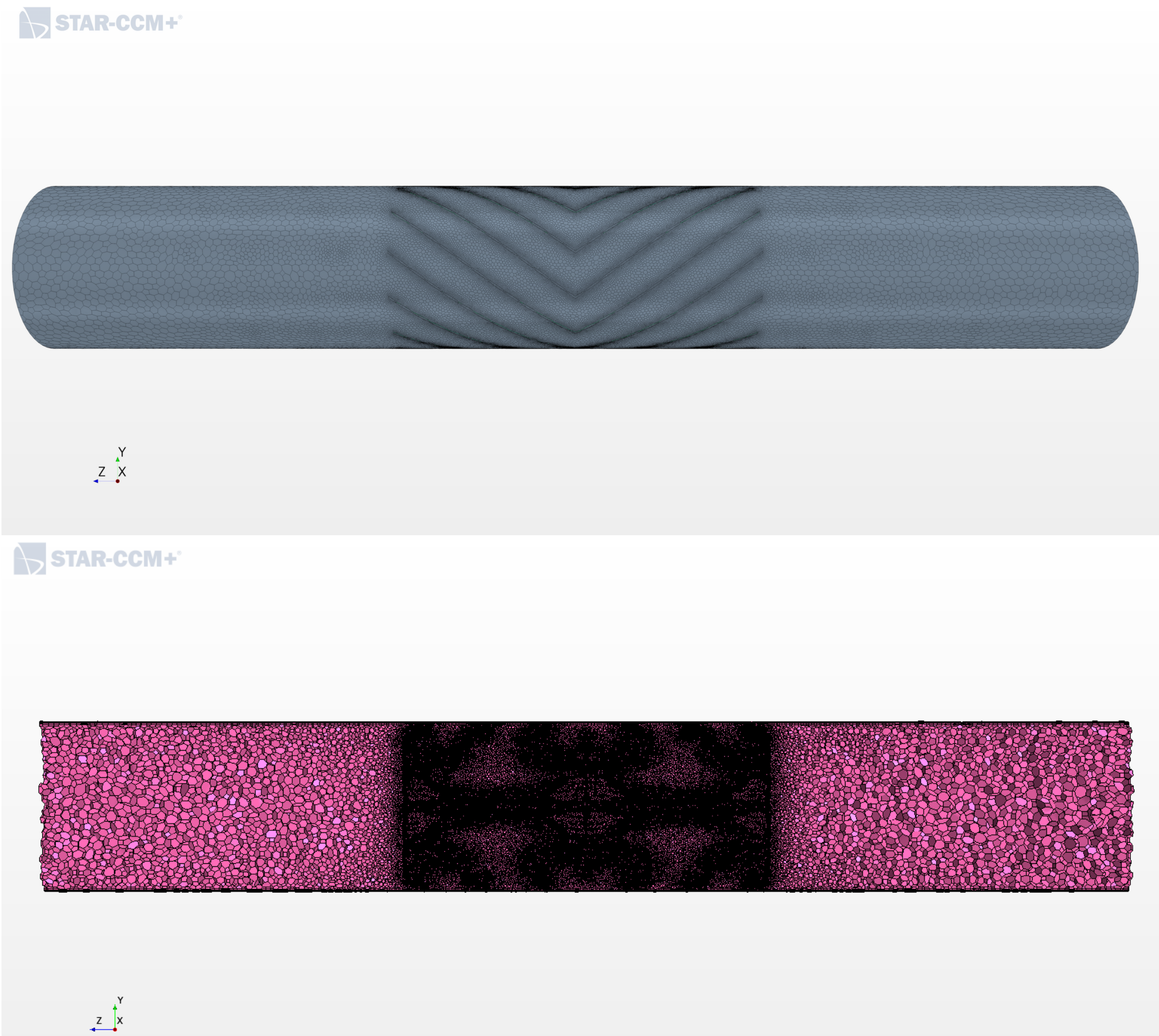


Figure 6. Mesh view of the model

After generating mesh using polyhedral cell we obtain for fluid 6784729 cells and for solid 3429631 cells, which should be far enough for accurate results. Pictures of the mesh are shown in Fig. 6 and the results in Fig. 7, Fig. 8.

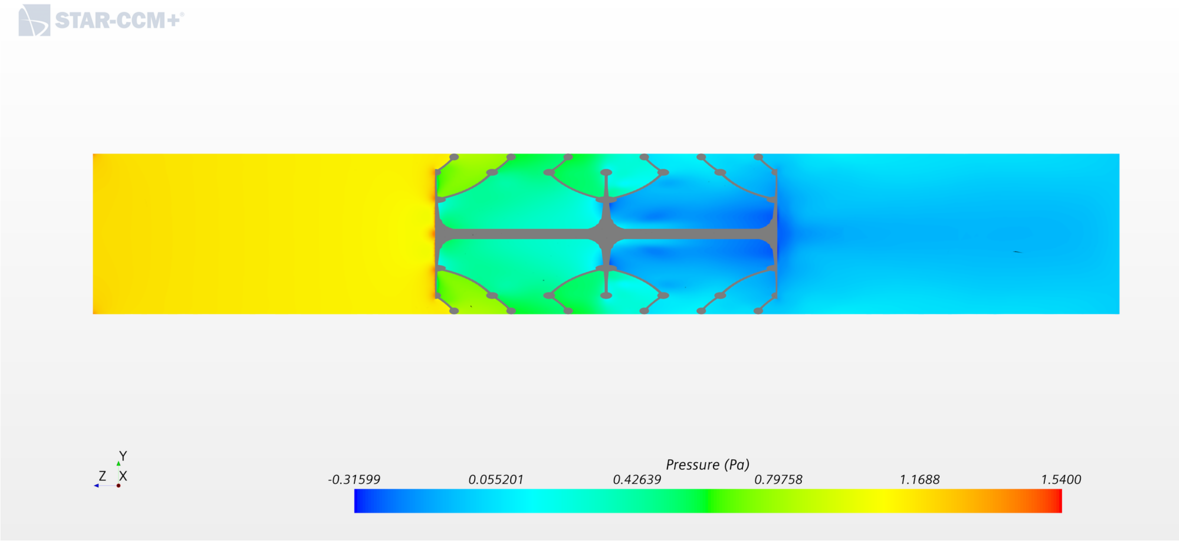
4. Conclusions

The temperature increases only by 7 K after passing the mixer, which is far away from what was required. But in the future this results can be improved. For this the geometry must certainly be changed. The heat exchange is based on contact between fluid element and is almost steady and slow, but we can enhance the convection to increase the heat exchange based on material transport and diffusion. So we need to improve the degree of turbulence.

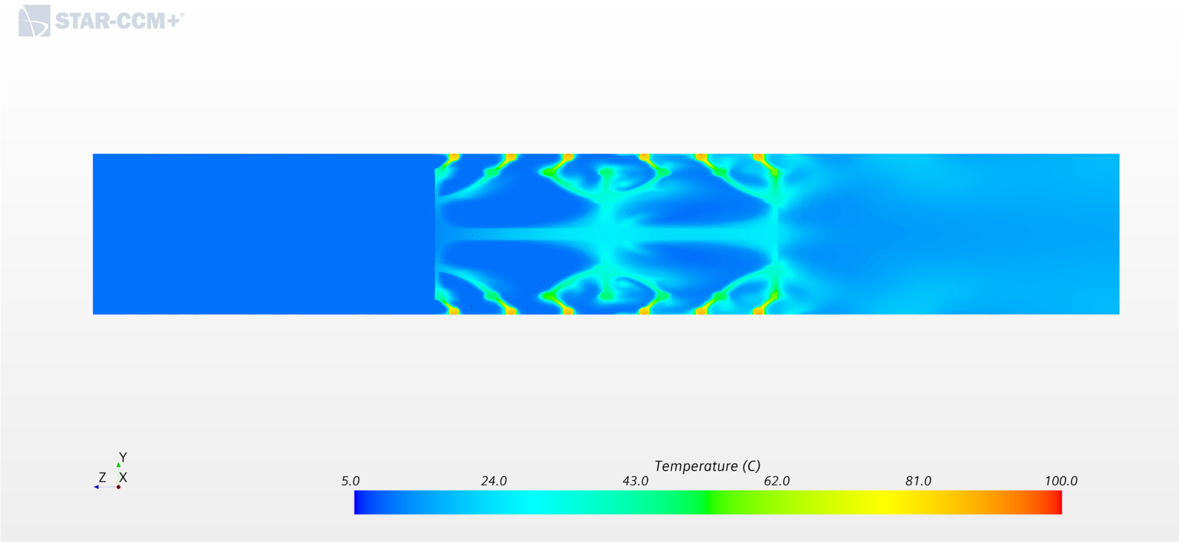
On the other hand, we can slow down the fluid velocity, so that the fluid element have more time to exchange heat.

At last, we can also increase the solid temperature, so that water get more heat under the same contact time. But this could lead to higher pressure when the liquid starts evaporating along the mixer.

Abbreviations

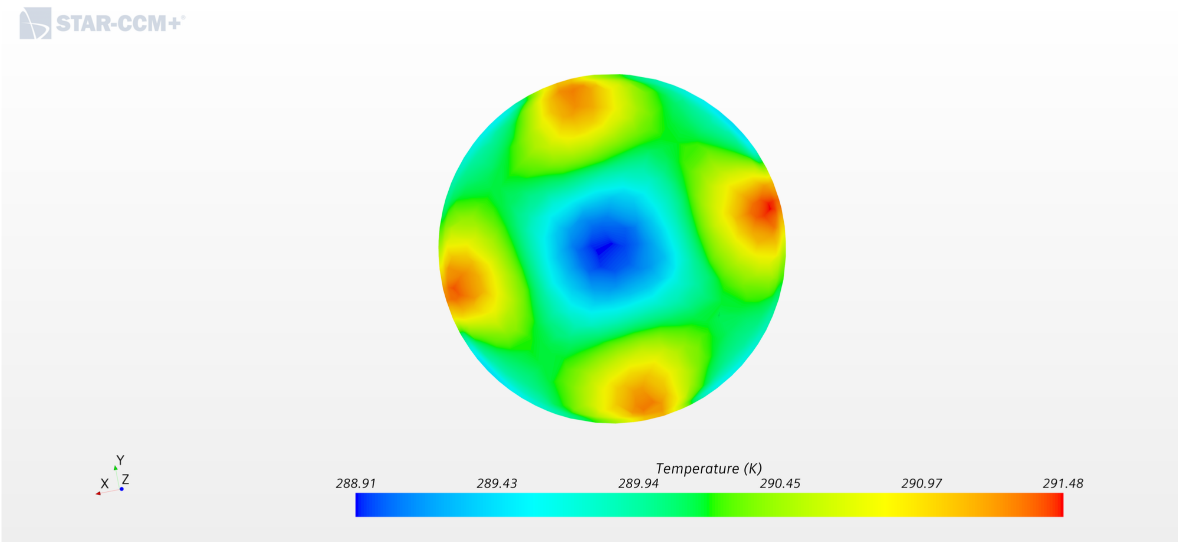


The press drop between inlet and outlet is 1.19[Pa].



The picture above shows temperature distribution along the pipe. It's clear, that water obtain heat from the mixer, because the color change near the interface between the water and the mixer shows obvious temperature gradient.

Figure 7. Side view of the results



Average temperature of the outlet is 17.045[°C], i.e. 290.045[K]. Standard temperature deviation at the outlet is 0.575[°C], which smaller than 1[°C]. Almost a evenly distribution along radial direction.

Figure 8. Outlet view of the results

The following abbreviations are used in this manuscript:

MDPI	Multidisciplinary Digital Publishing Institute
DOAJ	Directory of open access journals
TLA	Three letter acronym
LD	linear dichroism

Appendix A. Requirement Specification

Table A1. Requirement specification list for the heat exchanger

Obligatory or Desirable	Category	Description	Value
Obligatory	Performance	volume flow rate	greater 0.5 m^3/h
Obligatory	Performance	heat distribution	radial and evenly
Obligatory	Performance	heating water flowing trough	from 10 °C to 80 °C
Obligatory	Performance	low pressure drop	
Obligatory	Material	material temperature resistance	between 0 °C and 90 °C
Obligatory	Material	material water solvability	unsolvable in water
Obligatory	Material	electrical conductivity	greater 10^6 S/m
Obligatory	Manufacturing	manufacturing process	additive manufacturing
Desirable	Geometry	customizability of model	model is parameterized
Obligatory	Geometry	outer Shape	cylindrical
Obligatory	Geometry	outer diameter	94 mm
Desirable	Geometry	length	up to 300 mm
Obligatory	Geometry	outer wall	closed
Desirable	Geometry	outer wall thickness	smaller 5 mm
Obligatory	Geometry	water flow direction	fixed direction

Appendix B. Simulation Model Requirement Specification

Table A2. Requirement specification list for the heat exchanger simulation model

Obligatory or Desirable	Category	Description	Value
Desirable	Input	file type	STL
Obligatory	Input	fit within L=300 mm, D=94 mm cylinder	
Obligatory	Input	material	?
Obligatory	Simulation Geometry	tube	L=500 mm, D=94 mm cylinder
Obligatory	Simulation Geometry	inlet Velocity	See Table A.1
Obligatory	Simulation Geometry	wall type	adiabatic
Obligatory	Output Values	pressure value at outlet	outlet pressure
Obligatory	Output Pictures	cut along the length and center of the cylinder showing velocity	
Obligatory	Output Pictures	cut along the length and center of the cylinder showing temperature	
Obligatory	Output Pictures	at outlet showing temperature	

References

1. https://zhuanlan.zhihu.com/p/89651311 (visited on 14.09.2020)
2. https://uk.pcmag.com/3d-printers/74222/3d-printing-what-you-need-to-know (visited on 14.09.2020)
3. https://www.engineersedge.com/physics/water__density_viscosity_specific_weight_13146.htm (visited on 21.09.2020)
4. Skript zur Vorlesung chemische Reaktionstechnik 2, Page 42, Prof. Dr. Ing. Thomas Turek, Technische Universität Clausthal
5. https://www.schweizer-fn.de/stoff/wuebergang_fluessigkeit/wuebergang_fluessigkeit.php (visited on 21.09.2020)
6. OpenSCAD http://www.openscad.org/
7. Python https://www.python.org/
8. Github repository https://github.com/shukangzhang/3D-Druck/tree/master/openscad_spiral_model