Simulation and Verification of Bionic Heat Exchangers with COMSOL Multiphysics

A. Kremers*[1] and M. Pieper[1]
[1]Faculty Energy Technology, FH Aachen
*Corresponding author: Heinrich-Mußmann-Str. 1, 52428 Jülich, Deutschland, Contact: pieper@fh-aachen.de

Abstract: This paper describes the simulation and verification of heat exchangers with bionic channel structure, using the simulation software COMSOL Multiphysics. There exist different designs of heat exchangers, e.g. parallel or serial channel structure. But they all entail disadvantages like high pressure losses and hot spot formation. Bionic heat exchangers revert to the human blood vessel system. They are optimized for a steady and energy efficient fluid transport with low pressure losses, which are design requirements for heat exchangers as well. To analyze and verify the behavior of a bionic heat exchanger, a comparison between experimental test data and simulation results of COMSOL Multiphysics was performed. The simulation results differ, depending on the considered physical parameters, only in an acceptable extend from the experimental results. On the basis of this knowledge, Comsol Multiphysics can be verified as an extremely useful tool to design and test physical and technical processes e.g. in future work.

Keywords: COMSOL Multiphysics, bionic, heat exchanger

1. Introduction

Bionics is the scientific discipline between biology and technology. “The aim of bionics is to recognize natural design principles and derive technical solutions.” [1] Natural structures have been optimized over generations by evolution. In most cases the minimization of the energy demand is a major intend in nature, e.g. the human blood vessel system is optimized for efficient fluid transport. The main characteristics of this system can be transferred to heat exchangers, because in both cases the energy efficient fluid transport is one of the most important properties.

2. Design of the channel structure

First a 3D CAD model of a bionic channel structure was designed according to an existing, 2D model, taken from [2], which is based on the Michael Hermann’s FracTherm® algorithm. [3] Coordinate points were used as a basis for the 3D model, so that subsequent changes of the design could be performed easily. For an optimum comparability between experiment and simulation, it is necessary to use a 3D model of a bionic heat exchanger in the simulations.

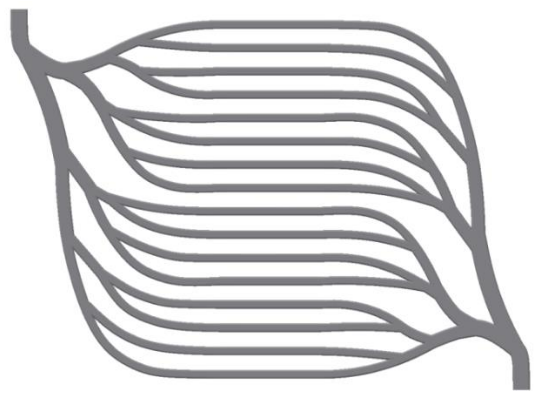

Figure 1: Bionic heat exchanger channel structure

Figure 1 shows the 3D channel structure of a bionic heat exchanger. The actual heat exchanger was extruded form the channel structure afterwards.

2.1 Solving CAD Problems

Figure 2 shows in both pictures the same branching of the bionic channel structure.

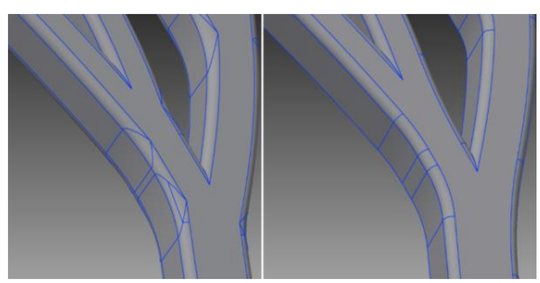

Figure 2: CAD problems

The geometry of both pictures is exactly the same, but generated in Inventor with two

different methods. The blue lines are construction lines, which Inventor needs to create the geometry. These lines can be regular (right picture) or very irregular (left picture), depending on the construction method. It can be very challenging to utilize the CAD model with “irregular lines” in a simulation, because of mesh-building and other problems. On the other hand, if the model with “regular lines” is taken, the simulation is fast and no problems will occur. This shows clearly, that choosing the right CAD model can simplify and speed up a simulation.
For simplicity reasons and because of the experiments, a square channel geometry was used.

3. Experimental Data

For a meaningful comparison, the same CAD model was used for both, the simulations, and to produce two real bionic heat exchangers out of aluminum.
To get a wide range of significant data, the two real heat exchangers got different cover plates. One out of aluminum, the other one out of glass. Figure 3 shows the real bionic heat exchanger with a glass cover plate.

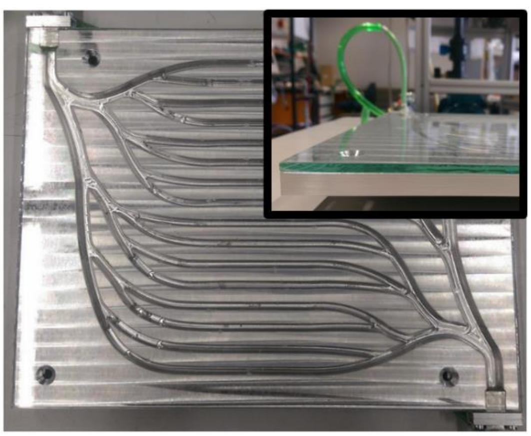

Figure 3: Real bionic heat exchanger

Several sensoric was used to record data of physical parameters, like temperature, pressure and flow velocity during the experiments.
Three different series of experiments were done:

3.1 Differential pressure

As the inlet fluid velocity was constant during the experiments, the differential pressure between heat exchanger in- and outlet was constant as well. A calibrated differential pressure measurement device was used to record the pressure difference between heat exchanger in- and outlet. As the pressure loss depends mainly on the fluid velocity, several experiments with different inlet fluid velocities were done. Usual tab water was used as fluid.

3.2 Temperature behavior

The bionic heat exchanger with aluminum cover plate was used in this experiment series. Water with a temperature of 50°C was selected as fluid. Temperature sensors recorded the in- and outlet temperature of the bionic heat exchanger once a second during the experiments. A thermographic camera was taken to record the temperature behavior on top of the heat exchanger cover plate. The fluid velocity stayed constant during the experiments.

3.3 General flow behavior

In this experiment series, the bionic heat exchanger with glass cover plate was selected. A special mixture of different fluids and micro particles was taken. In combination with the glass cover plate we were able to visualize the general flow behavior, like laminar/turbulent flow behavior, fluid velocities and swirls. This data was recorded by a high definition camera.

4. Use of COMSOL Multiphysics

The data, recorded during the experiments, was considered as input data for the comparability-simulations. The Reynolds number was calculated by the following equation:

$$Re = \frac{\rho v L}{\mu} \quad (1)$$

Here ρ is the density of the fluid, v is the maximum velocity of the fluid, L is the characteristic linear dimension and μ the dynamic viscosity of the fluid. According to (1) the Reynoldsnumber is, even with maximum fluid velocity, located in the border zone of laminar flow behavior. Therefore turbulent flow only occurs at the inlet of the bionic heat exchanger. Away from the inlet, the channel splits in several smaller ones, so that the fluid

velocity is getting slower, involving laminar flow behavior.
The same CAD model, which was taken to produce the real heat exchangers, was considered in the simulations. Depending on the simulation series, aluminum or glass was used as material for the cover plates. The material parameters were selected from COMSOL's material library.
All simulations were performed with several different simulation set-ups to get a good comparability between experimental data and simulation results.
These are the following two simulations:

4.1 Differential pressure and flow behavior:

The flow behavior inside the heat exchanger channels shows predominantly laminar behavior, so the "laminar flow" physic was chosen for this simulation series. The fluid velocity was constant, so the pressure loss and general flow behavior was constant as well. A stationary study was selected. As the characteristics of the special fluid mixture were not known, water was used approximately in the simulations.
The results will be discussed in chapter 5.

4.2 Temperature behavior

The fluid velocity in this simulation series was constant as well. But as the temperature behavior was from interest, a time dependent study was conducted. The non-isothermal flow physic interface of COMSOL 4.4 was applied to generate all important results.
The heat exchanger inlet temperature, recorded during the experiments, was not constant. At the beginning of the experiment, the set-up was in "cold" condition, so a rising inlet temperature was recorded. These recorded data was used to create an interpolation function as inlet temperature in COMSOL. Water was selected as fluid in these simulation series.
Again, the results will be discussed in chapter 5.

5. Results

The results of the comparison between experimental data and simulation results will be compared directly in this section.

5.1 Differential pressure

Table 1 shows the comparison of the differential pressure between inlet and outlet of the experimental data and simulation results at different inlet velocities.

Table 1: Differential pressure experiment/simulation

Inlet velocity [m/s]	Experiment Δp [mbar]	Simulation Δp [mbar]	Error [%]
0.55	3.95	3.63	8,1
0.91	9,95	8.82	11,3
1.27	19.22	16.65	13,7

The average error of 11% between simulation and experiment can be lead back to different sources of error:
On the one hand, this error can be justified with an imprecise measuring method during the experiments. On the other hand a laminar flow model was applied for simulation, but in reality there are small areas with turbulent flow as well. This thesis is supported by the comparative data shown in table 1. The COMSOL "laminar flow" physic was used in the simulations, because the calculated Reynolds number was in laminar border zone area. Since the real flow characteristic is shifted to turbulent flow behavior with increasing fluid velocity, the error increases proportionally to the flow velocity.

5.2 Temperature behavior

The comparison of the temperature behavior results was done in two steps. First the temperature difference between in- and outlet of the bionic heat exchanger was compared between experiment and simulation. Second, the video, recorded during the experiments, was compared to a "slice" evaluation on top of the simulated bionic heat exchanger.
Figure 4 shows a direct comparison between the temperature differences of simulation and experiment results.

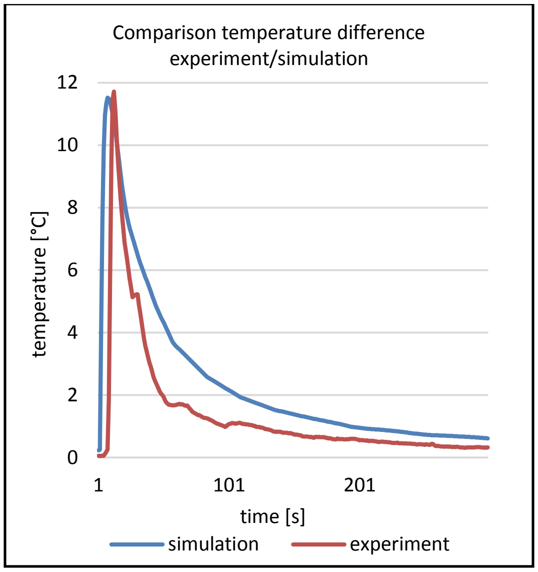

Figure 4: Comparison temperature difference between experiment/simulation

This comparison was done with several different inlet velocities and shows always the same behavior. The graphs do always have a similar trend, but are not congruent to each other. This can be justified by different reasons.

One the one hand the use of a laminar flow model, though in reality turbulent flow behavior dominates in some spots, has a significant influence on the results. On the other hand all material data in the simulations was used from COMSOL material library. Especially the characteristics of the aluminum, used to produce the real heat exchangers, are not known. The simulation results differ, depending on the used aluminum and its characteristics, significantly from each other. To get more precise results, an analysis of the real bionic heat exchanger aluminum would be necessary.

Figure 5 shows the comparison between simulation and experiment results of the temperature behavior on top of the bionic heat exchanger cover plate at a specific time.

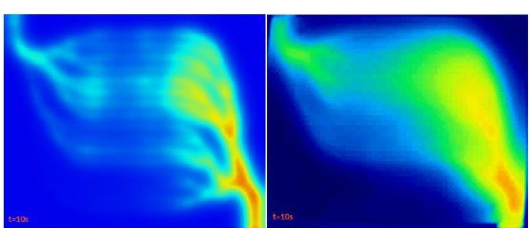

Figure 5: Comparison temperature behavior

The left hand picture shows the result of a slice evaluation of a simulation. The generated pictures do have a high contrast and are very detailed. The right hand picture is a snap shot of a video tape recorded with a thermographic camera during the experiments. It was shot exactly at the same time, compared to the simulation picture. The pictures of the thermographic camera do unfortunately not have a high resolution and quality.

But a direct comparison between simulation and experiment results shows an identical temperature trend. Primarily the heat exchanger channels with the highest fluid velocities get hot, because there is the highest amount of thermic energy applied. These channels are illustrated very detailed and with high contrast. With time proceeding, the details become blurred because the surrounding aluminum of the channels get hot was well.

In addition, the experimental results as well as the simulation results, show an unequal flow allocation inside the bionic heat exchanger channel structure. This can be recognized in figure 5. The lower half of the heat exchanger channels doesn't get hot as fast as the upper half. This can be reasoned with an awkward inlet channel design. The fluid velocities are higher in the upper half of the bionic heat exchanger, because the first bending of the inlet channel is too tight.

But both, the experimental results as well as the simulations results show this behavior, which proofs that the simulation results are close to the experimental results.

5.3 General flow behavior

The general flow behavior was made observable with a special fluid in the experiments. This way swirls, fluid velocities and laminar/turbulent flow behavior could be visualized. To compare this results with the simulation results, several evaluations where done.

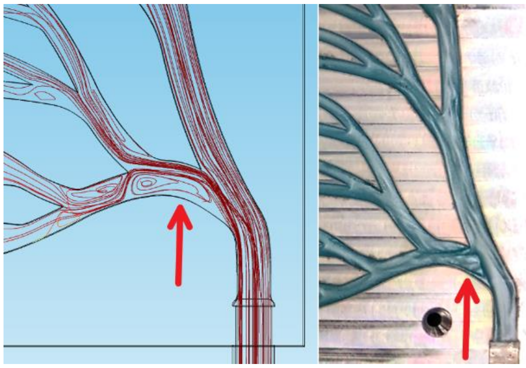

Figure 6: Comparison general flow behavior

Figure 6 shows the same bending of the bionic heat exchanger during the experiment/ simulation. The already discussed tight bending at the inlet channel (Ch. 5.2) causes a swirl. COMSOL predicts this swirl with same size and same position, compared to the measurements.

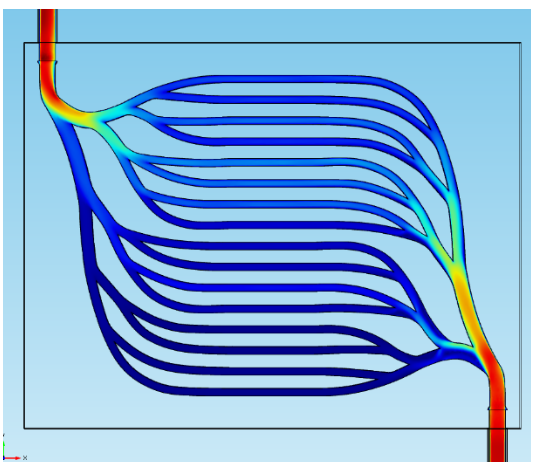

Figure 7: Fluid velocity profile

Figure 7 shows the fluid velocity profile of the bionic heat exchanger and the, already mentioned, unequally distributed flow allocation. In the lower half of the heat exchanger, significant lower fluid velocities can be recognized, than in the upper half. This behavior could be shown in the experiments as well.
In addition, the experiments with the special fluid show clearly, that in some spots of the bionic heat exchanger, turbulent flow is dominant. Especially at the inlet channel, where the highest fluid velocities are present. But as assumed, most of the bionic heat exchanger channels show a laminar flow behavior, which finally confines the choosen laminar module in COMSOL.

6. Improvement of the design

To get a higher efficiency and better results, the design of the bionic heat exchanger had to be improved. The too tight bending at the inlet channel had to be straightened.

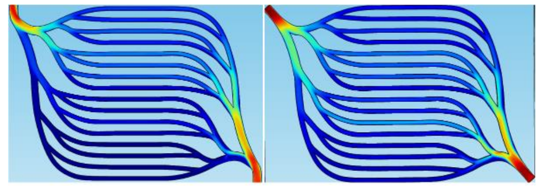

Figure 8: Improved bionic heat exchanger design

Figure 8 shows a comparison between original (left hand side) and the improved (right hand side) design. It can clearly be seen, that the flow allocation is much more homogeneous and that the fluid velocities are nearly equal all over the improved bionic heat exchanger.

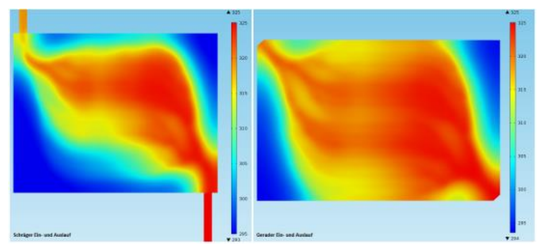

Figure 9: Comparison temperature behavior original/improved design

Figure 9 shows a comparison of the temperature behavior on top of the heat exchanger cover plate, between original (left hand side) and improved (right hand side) bionic heat exchanger design. The “straightened” bionic heat exchanger shows a homogeneous temperature distribution in all channels. This supports the conclusion, that a “straight” inlet design increases the efficiency of the bionic heat exchanger significantly.

7. Conclusion

The simulation results differ, depending on the considered physical parameter, only in an acceptable extend from the experimental results. These deviations are largely attributable to inaccurate measuring methods and flow velocities in the border zone of laminar flow behaviour. On the basis of this knowledge, Comsol Multiphysics® can be verified as an extremely useful tool to design and test physical and technical processes e.g. in future work. Now with the validated COMSOL simulation, further improvements and optimizations, similar to the discussed inlet position, can be virtually conducted with COMSOL.

6. References

1. M. Hermann, „Bionische Ansätze zur Entwickung energieeffizienter Fluidsysteme für den Wärmetransport,“ Dissertation (Fakultät für Maschinenbau Universität Karlsruhe (TH)), Karlsruhe, 2005.

2. P. Klein M. Pieper, „A simple and acurate numerical network flow model for bionic micro heat exchangers,“ Frauenhofer Institute (ITWM), Kaiserslautern, 2010.

3. M. Hermann, “Bionische Ansätze zur Entwicklung energieeffizienter Fluidsysteme für den Wärmetransport”, PhD-Thesis, Karlsruhe, Germany, 2015