Experimental and Numerical Investigation of the Heat Transfer Performance of a New Y-Shaped Fin Model

Kilic O.*, Bas F.**, Onbasioglu H.***
Research and Development Department,
Friterm Thermal Device Co.,
Tuzla, Istanbul
Turkey
E-mail: oguzkilic@friterm.com

ABSTRACT

In this study, convection heat transfer for a newly developed fin model of an air-cooled heat exchanger has been numerically and experimentally investigated at various airflow velocity and heat flux boundary conditions. One-dimensional parametric analyses were carried out with the fin design software internally developed by our team to obtain the optimum fin model geometry for the experimental study. A comparison has been made between the results of one-dimensional analysis and computational fluid dynamic analysis. Optimum fin geometry was obtained by using fin design software. A 10:1 scale Fin model was produced by using the additive manufacturing technology method for experimental study. The open-loop wind (EIFFEL) tunnel located at the R&D Center of FRITERM Inc. was used to perform the experimental study. During the experimental study, the surface temperature of the copper plates which were in contact with the plate resistant heaters was measured for constant heat flux. In order to maintain the constant heat flux during the experiment, the temperature difference between the inlet and outlet of the test chamber was also observed. For the numerical model, the scaled fin model within the test chamber was analyzed in 3D using a commercial computational fluid dynamics program. The computational numerical model has been initially checked for mesh independency. The results which are obtained as a result of the experimental studies and the outputs of the commercial CFD software were compared.

As a result of the comparisons, it has been observed that the new fin model simulated independently of production tolerances in the computer environment and tested in the wind tunnel and which is produced by additive manufacturing show similarities. As a result of these observations, it is aimed to develop a mathematical model in which the heat transfer coefficient of the new fin model can be expressed mathematically.

INTRODUCTION

Heat exchangers are used to transfer thermal energy between two or more environments and there are various types for different industrial applications. Compact heat exchangers and finned tube heat exchangers are some of these important types. The advantage of compact exchangers is that they have a high volumetric heat transfer area. Therefore, volume and weight are reduced and efficiency is increased. [7] Heat exchangers are widely used in gas-gas and liquid-gas applications in the cryogen, microturbine, automotive, chemical processes, marine, aerospace, heating, cooling, and air conditioning industries. Geometrically, these heat exchangers are in the form of finned tube and finned plate. The materials used are copper or aluminum tubes and plates on the liquid and fins on gas sides. Since the heat transfer coefficient on the airside is significantly smaller than the liquid side, the airside is the dominant side in the heat transfer coefficient. [9]

There are two ways of improving the heat transfer on the airside. The first is to increase the surface area outside the tube. In other words, the surface is expanded by adding circular or plate-type fins to the outer surface of the tube. The second is to increase the air velocity. In other words, it improves the heat transfer by increasing the heat transfer coefficient. [4] In this case, it is desired that the air velocity should not exceed a certain value, since the water entrainment is increased in case of pressure loss and condensation. In addition, since the increase in air velocity will increase the fin efficiency, then the heat transfer, as a consequence will also be affected positively. [3]

Optimization of thermal systems has an important role in improving heat transfer. In finned-tube heat exchangers, the thermal performance, besides fin type and fluid properties also depends on lots of parameters such as geometric parameters, tube diameter, tube arrangement, fin thickness, the distance between fins, number of tube column, number of tube rows.

NOMENCLATURE

Q	[W]	Heat Capacity
V	[W]	Heat Capacity
A_c	[m^2]	Cross-Section Area
m	[kg/s]	Mass Flow Rate
Cp	[J/kgK]	Specific Heat Capacities
ΔT	[K]	Temperature Difference
h	[W/m^{2}K]	Heat Transfer Coefficient
ΔT_{lm}	[K]	Logarithmic Temperature Difference
A_s	[m^2]	Heat Transfer Surface Area

The purpose of the study is to examine a newly developed fin model for finned tube heat exchangers. Newly developed fin model is being inspired from metal foam structure. Use of metal foam is aiming to increase the performance of the heat exchanger due to dispersion and low density. Therefore, the use of metal foams in the manufacture of compact heat exchangers for refrigeration and air conditioning systems has a high potential. [7] Metal foam also has a porosity range is between 0.4 - 0.6. [6] A Model with arranged structure with high porosity ($\varepsilon > 0.9$) was used in this project. The work carried out in this project has aimed to examine the heat transfer coefficient of the newly developed fin model under various boundary conditions (velocity, heat flux) experimentally, analytically, and numerically.

For analytical work Gnielinski correlation [1] has been used in 1-Dimensional thermal design. Dittus-Bolter [10] and similar correlations are also available in the literature. However, when the Reynolds number increases in complex geometries, the deviation in the Dittus-Bolter correlation increases. Therefore, the Gnielinski correlation, which is more suitable for complex geometries and can also be used at high Reynolds numbers, has been used. The Gnielinski Correlation which is referred to as legend in the result graphs, represents 1-Dimensional thermal design. Gnielinsnki correlation can be written as following (See. Equation 1).

$$Nu_{Dh} = \frac{(f/8)(Re_{Dh}-1000)Pr}{1+12.7(\frac{f}{8})^{1/2}Pr^{2/3}-1} \quad \text{(Eq.1)}$$

The newly developed Y-shape fin model used in experimental work could be seen in Fig.1.

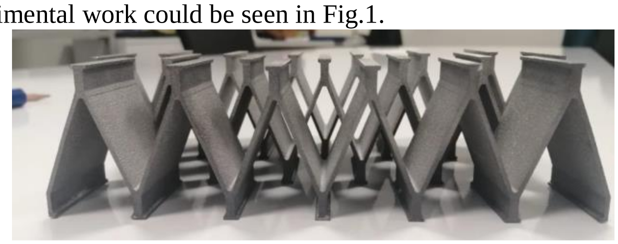


Figure 1. New Arranged Porous Media Y Fin Model

EXPERIMENTAL METHOD

Experimental studies were carried out on the fin model produced by the additive manufacturing method at different velocities and heat fluxes, and to compare with other different fin models. Experimental studies were carried out on the wind tunnel located at FRITERM R&D Center.

The Wind Tunnel at FRITERM R&D Center can be used as open-loop or close-loop. An open-loop wind tunnel was used for the tests to be carried out in this study.

Figure 2. FRITERM Co. Wind Tunnel

Siemens® QVB62.1 Velocity sensor was used to determine the velocity in the Wind Tunnel. The Siemens Velocity sensor was verified with the Testo 405i calibrated velocity probe. The comparison curve of these two sensors is given in figure 3.

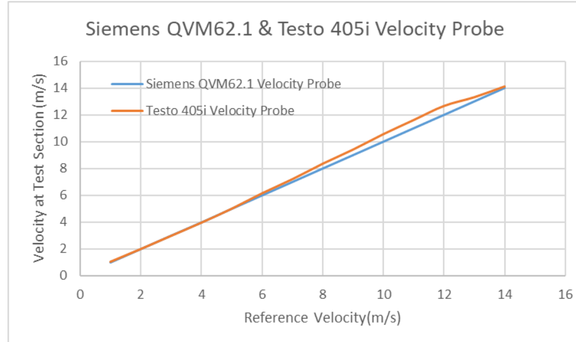


Figure 3. Siemens QVB6.2 and Test 405i Velocity Sensor

19 T-type thermocouples were used to measure the temperature at various points to determine the temperature distribution on the fins. The uncertainties of the thermocouples and velocity probe were calculated and shown in Table 1. Values taken from the thermocouple were recorded by the Agilent Datalogger 34970A data collection device. Heating plates are placed in the system layout to provide a constant heat flux. The heaters were manufactured to provide a power of 500 Watt as lower and upper heaters on both top and bottom surfaces. To control the power of the heaters, 0-10 V ENDA Erva1(SSR) heater control was used.

Table 1. Uncertainty Value Table

Measurement	Measurement Device	Measurement Range	Uncertainty (%)
Temperature	Type T Thermocouple	-200°C to 200°C	0.75
Velocity	Siemens Air Velocity Sensor	0-20 m/s	3
Data	Datalogger	0.05 C	0.008
Heat Capacity			3.092
Heat Transfer Coefficient			3.181

The prototype of newly Y-shaped fin model was placed in the test room and the tests were carried out. The heat transfer coefficient obtained from the software was compared with the heat transfer coefficient of the model as a result of experimental study under the same boundary conditions. Two plate heaters of 500 Watt each were used at top and bottom of the fin model The heat transfer coefficient was calculated from the measured data from the surface temperature of the copper plates at various heat fluxes and airflow velocity.

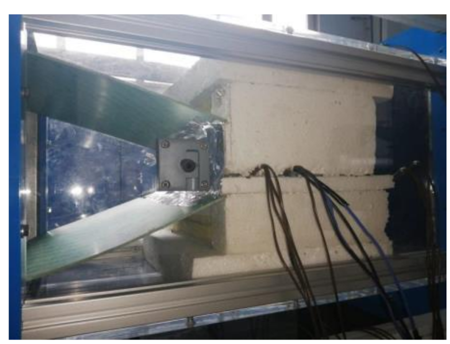


Figure 4. Test Section

A nozzle consisting of two contraction plates was placed in between the front of the fin model and the entrance of the test room. The nozzle is used in order to provide uniform airflow over the fin model and prevent secondary flows and thus fluctuations. Air velocity is measured at the Test Section entrance. The velocity values given in the test results are the test section entry velocity. If we simplify the continuity equation assuming that the density does not change for the same flow, the equation 2 can be written. [10]

$$v_1 A_1 = v_2 A_2 \tag{2}$$

Thus, the air velocity entering the fin model could be calculated. Verification was made with two different velocity probes.

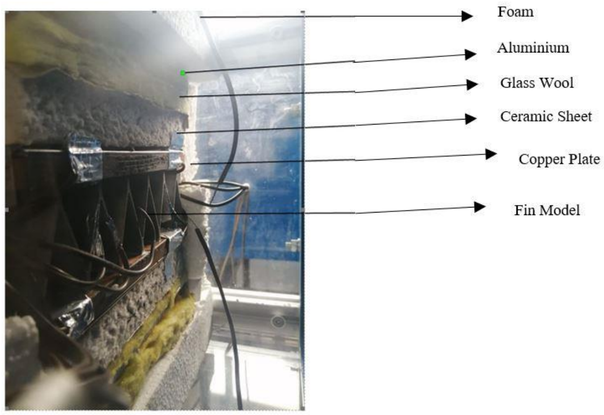


Figure 5. Test Section Materials of layer Structure

The experiments were carried out for the constant heat values of 400 Watt, 600 Watt, and 800 Watt. A Wind tunnel has a limitation of velocity. Pressure loss is high in the test section because of geometry and insulation layers. Due to this situation, velocity limitation is less than the default value. Therefore, no experiment was not carried out at high velocity. The air temperature in the test section should not exceed 50 °C for safety reasons. Therefore, heat values are limited to at most 800 Watt.

Experimental results were evaluated by comparing with MathCAD® software results. The comparison values as the results of MathCAD® software are air outlet temperature and surface temperature. The surface temperature is represented in the software as the temperature of the copper plate. There are 15 different points in the software for the change of the surface temperature.

Heat flux equal to the desired Watt value was provided on the heating plates located at the top and bottom of the fin model. Since the voltage value in the system is not constant and changes over time, the heat flux to the heaters was not constant either due to the work of other machines in the facility. The basis for calculating the heat flux is given in the system is equation 3[2].

$$Q = mCp\Delta T \tag{3}$$

Since the dimensions of the test chamber and the velocity of the air entering the test chamber are known, the mass flow rate can be calculated. It was observed that the Specific heat capacity is not significantly changing with temperature. Therefore, it can be taken as a constant of 1006 J/kgK. The ΔT value is taken from the test data as the inlet and outlet temperature differences. Using these values, the capacity calculation in the system is made.

The surface temperature values of different heat fluxes at different velocities are shown in the graph. The velocity values data obtained during the test are used to find the mass flow rate. Accordingly, the calculated heat transfer coefficient value was compared with the software and test results. In the results, the average surface temperature difference and the heat transfer coefficient difference between the software and the test can be examined. The heat transfer coefficient can be calculated that shown equation 4. [2]

$$Q = hA_s\Delta T_{lm} \tag{4}$$

Test results can be examined given below in tables and graphs.

Table 2. 400 Watt Test Result

Velocity (m/s)	Reynolds Num.	HTC Value(W/m2K) (Software)	HTC Value (W/m2K) (Test)	Calculated Heat Capacity (Watt)
1	6000	85.32	78.88	398.59
1.16	6960	96.91	93.78	385.042
1.33	7980	106.82	99.84	392.47

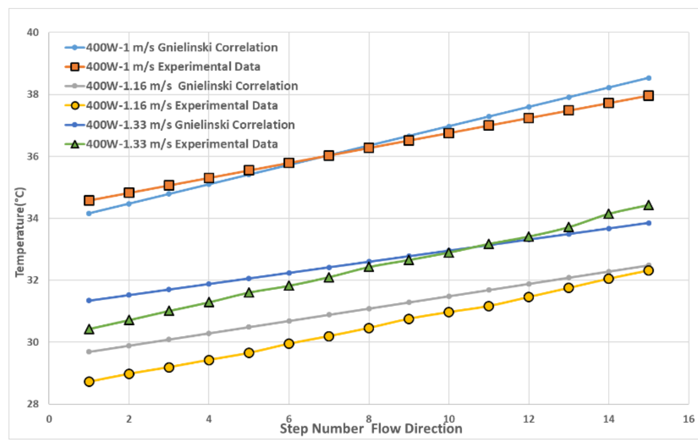


Figure 6. 400 Watt Various Heat Flux -Various Air Flow Velocity Test Results

Table 3. 600Watt Test Result

Velocity (m/s)	Reynolds Num.	HTC Value(W/m2K) (Software)	HTC Value (W/m2K) (Test)	Calculated Heat Capacity(Watt)
1m/s	6000	85.30	72.85	562.63
1.16 m/s	6960	96.80	96.44	594.06
1.33 m/s	7980	106.65	103.35	587.23

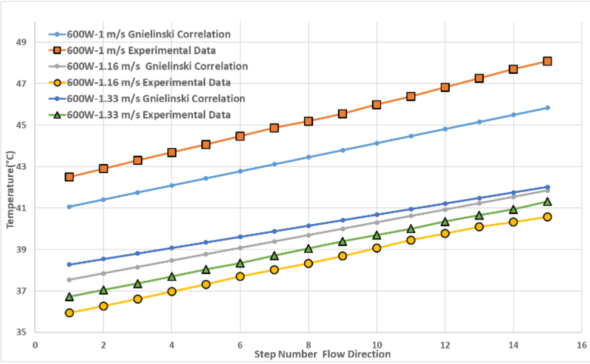


Figure7. 600 Watt Various Heat Flux -Various Air Flow Velocity Test Results

Table 3. 800 Watt Test Result

Velocity (m/s)	Reynolds Num.	HTC Value(W/m2K) (Software)	HTC Value (W/m2K) (Test)	Calculated Heat Capacity(Watt)
1m/s	6000	85.53	80.05	798.69
1.16 m/s	6960	96.23	95.50	777.42
1.33 m/s	7980	106.97	103.20	775.66

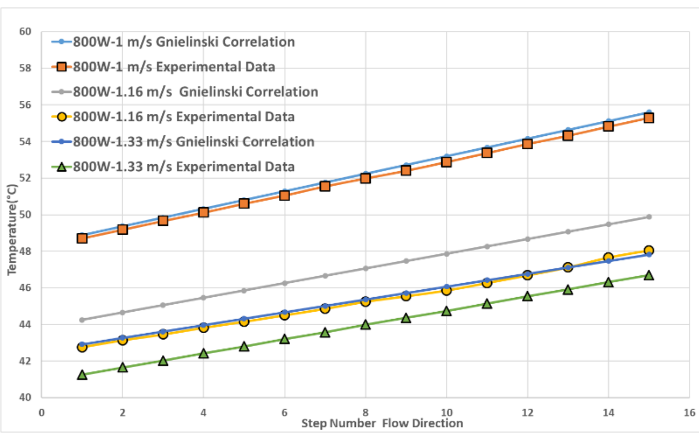


Figure 8. 800Watt Various Heat Flux -Various Air Flow Velocity Test Results

Experimental Result and Discussion

The analytical results and experimental results were compared. MathCAD® software was used for analytical calculations. It has been observed that experimental results are compatible with the analytical results.

The behaviour of the model at 400Watt, 600Watt and 800Watt heat fluxes for various air velocities were investigated. In previous analytical studies, since the surface area of the Y fin model is higher than the other fin models, the heat transfer coefficient value is then expected to be higher. Surface temperatures, on the other hand, depends on the air inlet temperature. Since the air inlet temperature is ambient air, surface temperatures may not give different results under the same test conditions. In the test with a lower mass flow rate, the surface temperature was higher. Experimental results and theoretical calculations seems to be compatible with each other. As the velocity value of the air entering the test chamber increases, the value of the Nusselt number, depends on the Reynolds number, increases due to the increase in the Reynolds value of the fluid in the duct. As the Nusselt number increases, the heat transfer coefficient is expected to increase. In the tests performed at different velocity, it was observed that the heat transfer coefficient increases as the velocity value in the channel increases.

When the experiments are performed at different heat fluxes for the same velocity are compared, it has been observed that the given constant heat flux changes the heat convection coefficient at a negligible level (0.32%). In this case, it was concluded that the thermal convection coefficient is independent of the heat flux given to the system.

NUMERICAL METHOD

CFD studies have been carried out in 3 steps. In the first step, the correct analysis solution model setup and proving mesh independence. The second step is, the analysis of the experimental setup with the correct mesh elements. The third step is evaluation of all analyses and compare to test results.

Mesh Independence Analysis and Correction of Analysis Model

In this stage, the geometry in the test tunnel was modeled and reduced to the extent allowed by the computer physical properties. As a result of these stages, it has been advanced as in figure 9.

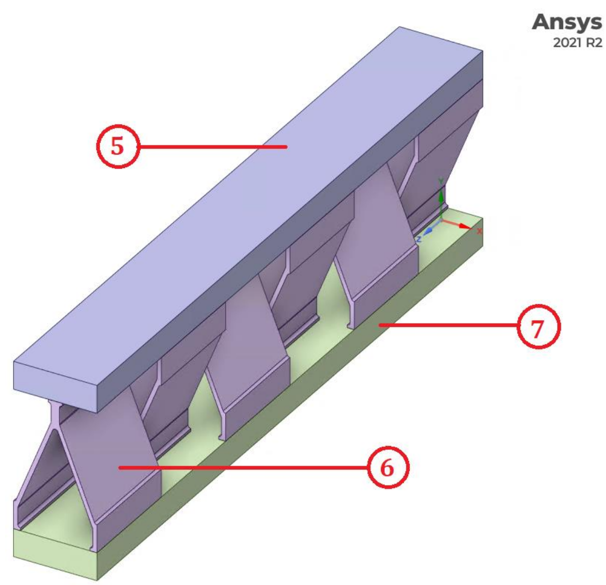


Figure 9. Fin Model

The description of the numbers is shown in Table 4.

Using the symmetry assignment feature which is a feature of ANSYS the model is designed to repeat from the right to the left.

Mesh independence analysis, it was first reduced from 65 Million mesh elements to 14 million mesh elements depending on the skewness value. In the second study case it was reduced from 14 million mesh elements to 2.6 million mesh elements depending on ΔT and HTC. The optimum mesh elements structure is also shown in figure 10.

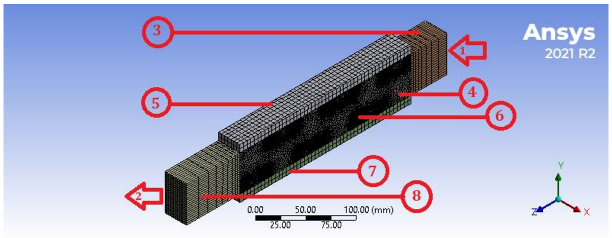


Figure 10. Optimum Mesh Element

Table 10. Description of the Numbers

1	Inlet
2	Outlet
3	Inlet side air domain
4	Inner air domain
5	Upper copper plate
6	Fins
7	Bottom copper plate
8	Outlet side air domain

Moreover, In the images below shows the mesh independence work that was done before. If we examine the previous studies, it shows that the 2.5 Million mesh element structure is the correct solution

Analysis of Experimental Sets

After the determined mesh number, the analyzes were set up to resemble the tests performed in the tunnel. At this stage, constant heat flux is defined on the copper plates. In previous studies, constant temperatures were defined for these surfaces.

Evaluation of Results

The temperature profile taken from the middle plane is shown in the figure 11 and 12 below.

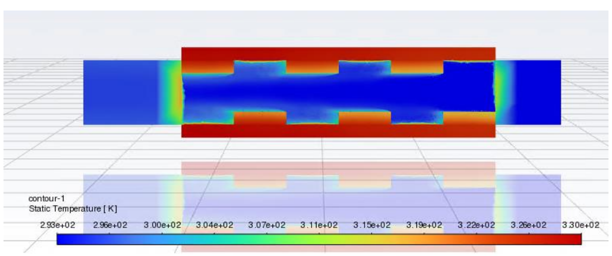


Figure 11. Temperature Distribution Left Surface

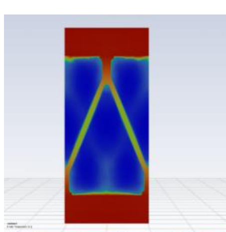


Figure 12. Temperature Distribution Front Surface

The velocity distribution is shown in the figure 13 below.

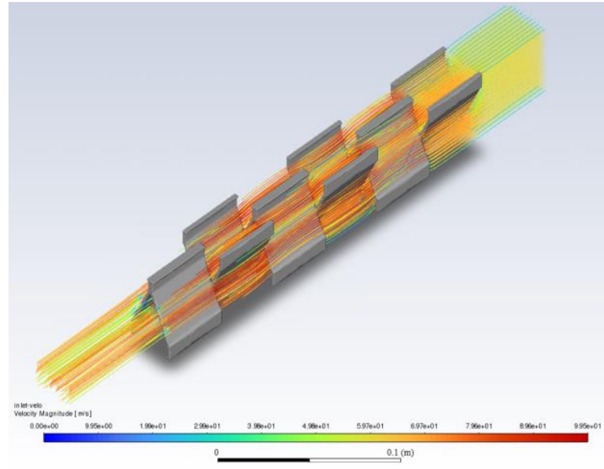


Figure 13. Velocity Distribution

The results of all these solutions are expressed in the table 5 below.

Table 5. CFD Results

400 W	0.1098	0.12528	0.146	kg/s
	77.67997	86.03348	97.74143	HTC
600 W	0.1098	0.12528	0.146	kg/s
	76.99751	86.05755	97.73703	HTC
800 W	0.1098	0.12525	0.146	kg/s
	77.66142	86.05842	97.73855	HTC

CFD Result Discussion

The results of the analysis were confirmed in two ways. The first of the verification methods are mathematical and physical evaluation. The second form of evaluation is the wind tunnel test results.

The outputs of the analysis results were designed as Temperature change and HTC values. Likewise, in the articles written about HTC calculation, how to calculate HTC is also written. Within the framework of these calculations, it was determined that the CFD studies repeated with other W values produced results close to the expected values.

In the physical control phase, velocity vectors and temperature distributions were examined. When we examine the temperature factor above, we need to see the temperature change in the parts of the copper plates that touch the fins. Likewise, the fluid passing between the fins should be heated towards the end.

When we evaluate all these, it can be thought that the solution is correctly constructed.

It also provides the fact that HTC values should not change in repeated analyzes at the same flow rates at different heat flux values, which is one of the cornerstones of heat transfer.

The results produced with the CFD are compared with the experimental results. The values obtained in the wind tunnel tests were used to validate the CFD results. It has been noticed that there are acceptable differences between the test results explained in more detail in the relevant parts of this report and the CFD results.

This acceptable difference is defined as follows. HTC values produced as a result of the experiments and HTC values produced by the software were compared with the percentage of absolute difference method of HTC Values produced as a result of CFD. It has been determined not to exceed 15% for each of these different value. In the calculation of the average error, it was determined not to exceed 10%. The graphs of the heat transfer coefficient calculated with the software, CFD, and test data for the various constant heat flux and various airflow velocities are shown below.

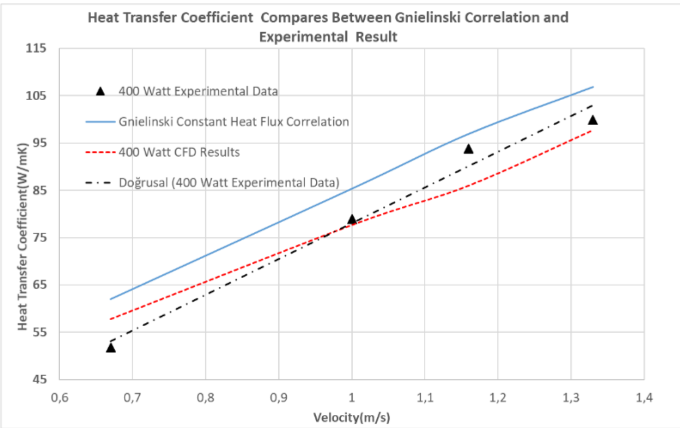


Figure 14. 400 Watt HTC Compare with All Results.

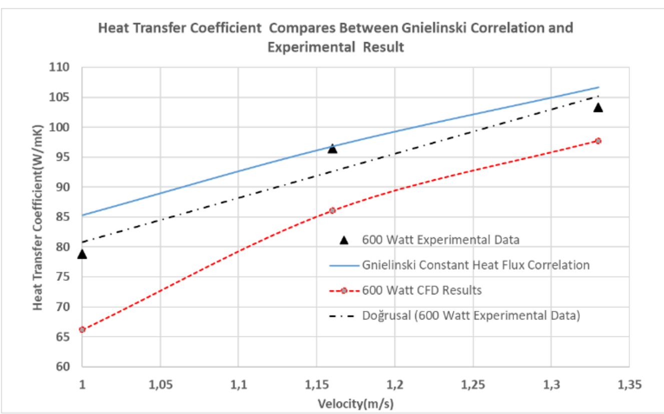
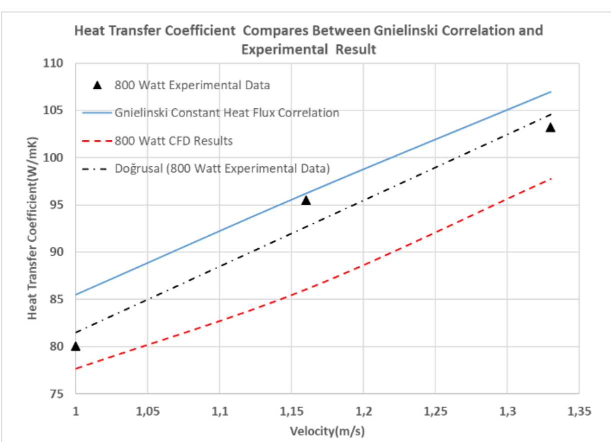


Figure 15. 600 Watt HTC Compare with All Results.



S

Figure 16. 800 Watt HTC Compare with All Results.

CONCLUSION

The graphs of the variation of the heat transfer coefficient with velocity are given above. Here, the software result calculated analytically by using MathCAD, tests with a wind tunnel, and a comparison of computational fluid dynamics modelling with a numerical method using ANSYS FLUENT® can be seen. While the highest heat transfer coefficient was obtained in the software, the lowest heat transfer coefficient was obtained in computational fluid dynamics modelling. The velocity-heat transfer coefficient graph obtained in MathCAD software shows a linear behaviour. Test and computational fluid dynamics modelling results also tend to be linear. Particularly, it gave results compatible with the software at a velocity of 1.16 m/s.

ACKNOWLEDGMENT

I would like to thank our consultant A. Feridun OZGUC for his support throughout this study. I would also like to thank FRITERM Co. General Manager Naci SAHIN and R&D Chief Feyza SAHIN. Thanks to Aydın KARABABA and Yüksel ALTUNKIRAN who always supported us during the tests process.

REFERENCES

[1] V. Gnielinski
On heat transfer in tubes
J HeatTran, 63 (2013),pp. 134140, 10.1016/j.ijheatmasstransfer.2013.04.015 in Engineering and Technology, Volume 6 Issue 2– November 2016 28 ISSN: 2319-6319

[2] Incropera, F., DeWitt, D. (2002): Introduction to Heat Transfer, 4th ed., Wiley, NewYork.

[3] Sahiti, N. Durst, F. and Dewan, A. (2005a): Heat Transfer Enhancement by Pin Elements, Int. J. Heat Mass Transfer, pp. 4738-4747.

[4] Ya-Ling He, Song-Zhen Tang, Wen-Quan Tao, Ming-Jia Li, Fei-Wang (2019): A general anad rapid method for performance evaluation of enhanced heat transfer techniques, Int. J. Heat Mass Transfer, 145 118780.

[5] Hannes Fugmann, Eric Laurenz and Lena Schnabel (2019), Multi-Dimensional Performance Evaluation of Heat Exchanger Surface Enhancements, Energies 2019, 12, 1406; doi:10.3390/en12071406

[6] Azadeh Ejlali1, Arash Ejlali, Kamel Hooman, Hal Gurgenci, Application of high porosity metal foams as air- cooled heat exchangers
to high heat load removal systems, International Communications in Heat and Mass Transfer 36 (2009) 674–679.

[7] K. Boomsma, D. Poulikakos, F. Zwick, Metal foams as compact high performance heat exchangers, Mechanics of Materials 35 (2003) 1161–1176.

[8] Francesco Nicolò Ponzio, Alessandro Tamburini, Andrea Cipollina, Giorgio Micale, Michele Ciofalo, Experimental and computational investigation of heat transfer in channels filled by woven spacers, International Journal of Heat and Mass Transfer 104 (2017) 163–177

[9] Okbaz, Abdulkerim & Onbaşıoğlu, Huseyin & Olcay, Ali & Pinarbasi, Ali. (2017). Investigation of Louvered Fin Heat Exchangers Performance via Experimental and Computational Fluid Dynamics Approach. Engineer and Machinery. 58. 41-55

[10] Cengel, Yunus A., and Afshin J. Ghajar. *Heat and Mass Transfer: Fundamentals and Applications*. 5th ed., McGraw-Hill Professional, 2014.