Int. J. Adv. Eng. Pure Sci. 2021, 33(1): 53-57
DOI: 10.7240/jeps.705202

RESEARCH ARTICLE / ARAŞTIRMA MAKALESİ

Numerical Examination of Heat Exchanger Shape Effects on Thermal Efficiency In Air to Air Plate Type Heat Recovery Unit

Havadan Havaya Plakalı Tip Isı Geri Kazanım Cihazlarında Isı Değiştiricisi Geometrisinin Isıl Verimlilik Üzerindeki Etkilerinin Sayısal Olarak İncelenmesi

Hasan SAVAT[1], Mustafa ATMACA[2]

[1] Marmara Üniversitesi Fen Bilimleri Enstitüsü, 34722, İstanbul, Türkiye
[2] Marmara Üniversitesi, Makine Mühendisliği Bölümü, 34722, İstanbul, Türkiye

Abstract

Energy needs and consumption in the world is increasing steadily. For this reason, the importance of reducing energy consumption and using energy more efficiently increases day by day. Hence, energy efficiency has great importance both economically and for the protection of nature.

Heat recovery device is a system that needs ventilation and can be used to save energy (reduce energy consumption) in all areas of different temperatures (home, office, cinema, shopping mall, conference room, etc.) with the outdoor environment. This device can save a significant amount of energy from heating and cooling processes by transferring heat between the exhaust air and the fresh air.

The most widely used type of this device is a plate heat exchanger. In this system, heat transfer is carried out without allowing to blend the egzost air and the fresh air. In this study, the aim is to increase the thermal efficiency of the device. The geometry of the heat exchanger was changed to increase the amount of heat transfer.

For this purpose, the current model was created in computer environment and computer simulations were made through the ANSYS-Fluent program. Then the new models with different geometries were created in the computer environment and simulations were made on these models using the same method. The current model and the new models have been compared in terms of thermal efficiency.
Key Words: Heat recovery, energy efficiency, heat transfer, heat exchanger, ventilation system, CFD

Öz

Dünyada enerji ihtiyacı ve tüketimi giderek artmaktadır. Bu sebep ile enerji tüketimini azaltmak ve enerjiyi daha verimli kullanılmak her geçen gün daha önem kazanmaktadır. Bu sebeple enerji verimliliği gerek ekonomik gerekse doğanın korunması açısından büyük önem taşımaktadır.

Isı geri kazanım cihazı, havalandırılma ihtiyacı olan, dış ortamla farklı sıcaklıktaki tüm mahallerde (ev, ofis, sinema, AVM, Konferans salonu vb.) enerji tasarrufu sağlamak (enerji tüketimini azaltmak) için kullanılabilen bir sistemdir. Bu cihaz dışarı atılacak hava ile iç ortama alınacak hava arasında ısı transferi yaparak, ısıtma ve soğutma işlemlerinden ciddi oranda tasarruf ve enerji kazancı sağlayabilmektedir.

Bu cihazın en yaygın kullanılan türü plakalı ısı değiştiricili olanıdır. Bu sistemde ısı transferi dışarı atılan kirli hava ile içeri alınan temiz hava arasında 2 havanın karışmasına izin verilmeden yapılmaktadır. Bu çalışmada cihazın, ısıl verimliliğinin artırılması amaçlanmıştır. Bunun için kullanılan ısı değiştiricisinin geometrisi değiştirilerek ısı transferi artırılmak istenmiştir.

Bunun için mevcut olan model sanal ortamda (bilgisayar ortamında) oluşturularak, ANSYS-Fluent programı vasıtasıyla bilgisayar benzetimleri (simülasyonları) yapılmıştır. Sonrasında bilgisayar ortamında farklı geometrilere sahip yeni modeller oluşturulmuş ve aynı metot kullanılarak bu modeller üzerinde benzetimler (simülasyonlar) yapılmıştır. Yeni modellerin mevcut model ile ısıl verimleri karşılaştırılmıştır.
Anahtar Kelimeler: Isı geri kazanımı, enerji verimliliği, ısı transferi, ısı değiştiricisi, havalandırma sistemi, HAD

I. INTRODUCTION

Energy efficiency has a great importance today because of its economic importance and it can slow down or even prevent global warming. In order to prevent the effects of global warming, a study found that; one of the effective methods that can be used is to increase energy efficiency [1]. Energy efficiency is possible with minimizing energy losses without reducing quantity and quality.

Corresponding Author: Mustafa ATMACA, Tel: (0216) 7773869, e-posta: matmaca@marmara.edu.tr
Submitted: 17.03.2020, **Revised:** 02.09.2020, **Accepted:** 14.09.2020

Heat recovery device is a system that can be used to save energy by utilizing waste heat in all locations (home, office, cinema, shopping mall, conference room, etc.) that need ventilation at different temperatures with external environment. This device can save a significant amount of heat and energy from heating and cooling processes by transferring heat between the air to be ejected and the air to be taken into the interior. Pamir [2] talked about the importance of saving energy and reducing energy consumption. Another study mentioned the importance of energy efficiency and what can be done about it [3]. In his study, Satman [4] touched on the effects of energy efficiency on the economy and suggest the importance of the steps to be taken and the vision in this regard.

Şahan examine, the usage of different shapes air to air heat recovery exchanger using, in his study. He said these systems are aimed at providing a business economy and that the right system will provide economic profitability in the long term [5].

Şentürk and his colleagues [6] have done experimental and numerical work of a heat recovery ventilation device. They have confirmed the numerical results with the experimental results. Simulations; they were carried out with ANSYS Fluent software.

Heat recovery in air conditioning systems has three important advantages besides increasing energy efficiency:
1. Thanks to the restored heat, the external air load required to bring the interior environment to comfort conditions is significantly reduced. This reduces the amount of energy expended for air conditioning, saves money.
2. The initial investment cost can be reduced if both the capacities of the components used in the air conditioning system are reduced and there is no need for humidification and dehumidification components.
3.Decreasing external air load makes the use of fresh air devices are more reasonable. Thus, the superiority is achieved in ensuring the quality of internal air [7].

This operation was done for heat recovery device with plate heat exchanger. The fig.1. includes installation view and internal structure view of the device in its operating position.

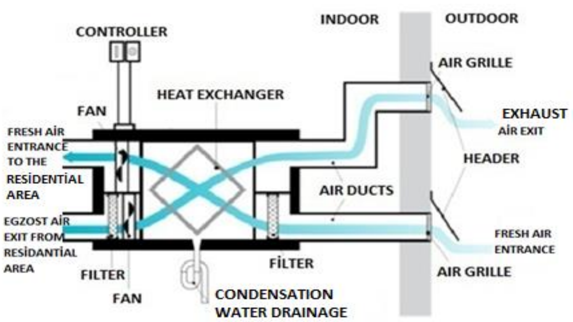

Fig.1. Installation and internal structure of the device in operation position [8]

In this study, the heat exchanger of the device' thermal efficiency; numerically was examined. New heat exchanger models were created, numerically examined with the same method and compared with the current model.

II. MATERIAL AND METHOD

The heat exchanger used in the device was created in a computer through Solidworks program.

This heat exchanger includes of 108 lamellas overlapping. One of these lamellas is seen in fig.2.

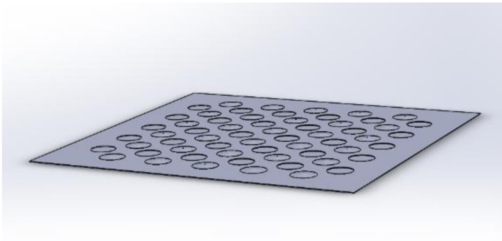

Fig.2. Lamella

The lamellas were rotated and lined up to overlap and the lower-upper covers and slats were added to the model. Computer generated of the model, is seen in Fig.3.

Fig.3. Current Model

The heat exchanger, which will be simulated by the computer, has a geometry that repeats itself periodically. For this reason, the simulation of only one of the repeated shapes and the whole will give the same result.

Because geometry is made up of complex and small units, the mesh network to be created on the whole model will be extremely complex and the processing power required for computer simulation will be at a very high level. For this reason, apart from not being able with existing devices, it will cause an unnecessary waste of process.

One of the pieces which geometry is divided periodically is examined. Periodic region is seen in fig.4.

Fig.4. One of The Periodic Piece

The relevant flow area, where hot and cold air pass through the periodic zone, has been added and mesh networking has been started in order to make computer simulation. Images can be seen from the established mesh network in Fig.5. The established mesh network was prepared through the mesh module owned by the Ansys program.

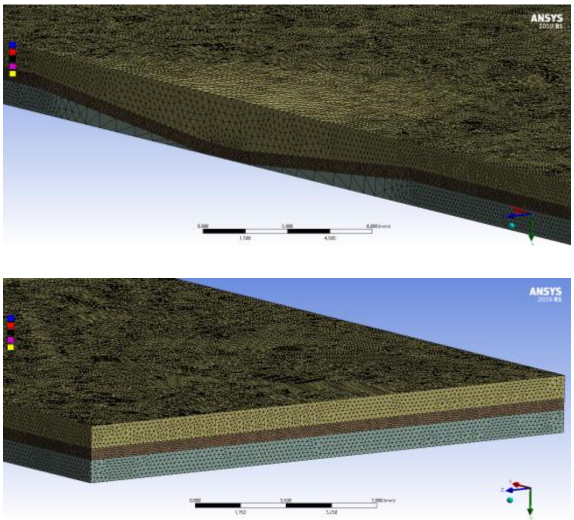

Fig. 5. View of Mesh Network

The details and features of the established mesh network are seen in fig.6.

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	10, mm
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1,2)
Max Size	Default (20, mm)
Mesh Defeaturing	Yes
Defeature Size	Default (5,e-002 mm)
Capture Curvature	Yes
Curvature Min Size	Default (0,1 mm)
Curvature Normal Angle	Default (18,°)
Capture Proximity	Yes
Proximity Min Size	Default (0,1 mm)
Num Cells Across Gap	6
Proximity Size Function Sou...	Faces and Edges
Bounding Box Diagonal	300,85 mm
Average Surface Area	397,35 mm²
Minimum Edge Length	0,5 mm
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	0,8
Smoothing	Medium
Mesh Metric	Element Quality
Min	4,9516e-002
Max	1,
Average	0,83672
Standard Deviation	8,2991e-002
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Assembly Meshing	
Method	None
Advanced	
Number of CPUs for Parallel...	4
Straight Sided Elements	
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Default (9,e-002 mm)
Generate Pinch on Refresh	No
Statistics	
Nodes	6500042
Elements	33184738

Fig.6. Mesh Details

The mesh network has been transferred to the Fluent Module of the ANSYS program and the energy option has been switched from off to on for the Fluent module is requested to include thermal data in the simulation. The mathematic model was selected as k-epsilon model realizable, Near Wall treatment: Standard Wall Function [9] [10]. For material information, fluid as air and solid as aluminum were defined. Inlet, outlet (pressure outlet), wall and symmetry are defined, and air speed is calculated by dividing the total flow rate into the inlet surface area of the heat exchanger; 6.9m/s. Turbulent intensity: 5%, turbulent viscosity ratio: 10%

[11]. By entering the input temperatures of the air, the computer was simulated through the module. The temperature distribution obtained as a result of this process is located at Fig.7. The results are consistent with the values provided by the manufacturer.

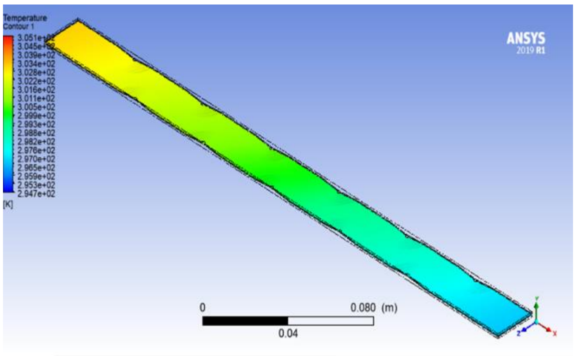

Fig.7. Temperature Distribution

While creating new models, the exterior dimensions, exterior geometric forms and lamella ranges of the models remained the same. These changes are to change the protrusions on the conical lamellae in the current model to be cylinder, rectangular prism and hexagonal prism. View of periodic regions of the generated models is seen in Fig.8.

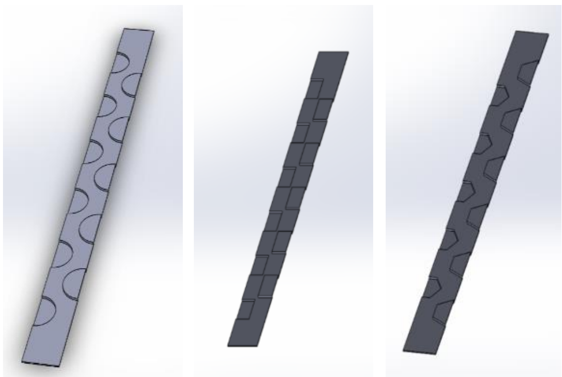

Fig.8. Periodic Region Views of Models

Computer simulation operations of all models were performed using the same method as the current model. Obtained temperature distribution is seen in Fig.9.

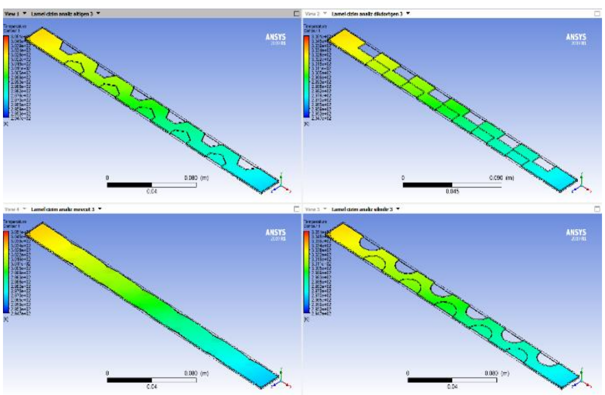

Fig.9. Temperature Distribution of Models

III. RESULTS

The temperature values obtained as a result of computer simulation are in the table 1.

Table 1. Temperature Values Obtained as a Result of Simulation

	Current Model	Cylinder Model	Rectangle Model	Hexagon Model
Air Flow (m^3/h)	800	800	800	800
Tei (°C)	32	32	32	32
Tfi (°C)	21,6	21,6	21,6	21,6
Tfo (°C)	26,9	27,3	27,3	27,2

$$\textbf{\textit{Thermal}}\ Efficiency = (T_{fo} - T_{fi})/(T_{ei} - T_{fi})[12] \quad (1)$$

Tfo, Tfi and Tei are the inlet and output temperatures for the relevant locations of the air. Temperature values in "°C".

Tfo: Fresh air outlet temperature

Tfi: Fresh air inlet temperature

Tei: Exhaust air inlet temperature [12]

Thermal efficiency values calculated by equation 1 are shown in table 2.

Table 2. Comparison of Thermal Efficiency Values

	Current Model	Cylinder Model	Rectangle Model	Hexagon Model
Air Flow (m3/h)	800	800	800	800
Tei (°C)	32	32	32	32
Tfi (°C)	21,6	21,6	21,6	21,6
Tfo (°C)	26,9	27,3	27,3	27,2
Thermal Efficiency (%)	51	55	55	54

IV. CONCLUSION

As energy consumption is increasing in line with increasing population and technological developments, the importance of energy efficiency, which has great importance, will increase day by day. Depending on population growth, technological developments and global warming, the increasing use of air conditioning processes will increase energy consumption in this regard day by day.

This study has provided us to obtain important information about energy efficiency and computer simulations in the field of air conditioning systems.

Looking at the numerical working results, the new models that have been created have increased the thermal efficiency compared to the current model. The cylinder and rectangular model increased by 4%, while the hexagonal model increased by 3%. Increasing thermal efficiency means that increasing heat being gained so this is desired situation. However, the new models have structures that make air passage difficult, they need to be checked before they can be implemented.

The computed results of the numerical work for the current model match the data provided by the manufacturer. This situation confirms the numerical working method. For this reason, the numerical working method can be use in determining the new models to be created in the future.

REFERENCES

[1] Bayraç, H. N. (2010). Enerji Kullanımının Küresel Isınmaya Etkisi ve Önleyici Politikalar. *Eskişehir Osmangazi Üniversitesi Sosyal Bilimler Dergisi*, *11*(2), 229-259.

[2] Pamir, N. (2005). Enerji Politikalar ve Küresel Gelişmeler. *Stratejik Analiz*, *6*(68).

[3] Narin, M., & Akdemir, S. (2006). Enerji verimliliği ve Türkiye. *Türkiye Ekonomik Kurumu UEK-TEK*.

[4] Satman, A. (2007). Türkiye'nin enerji vizyonu. *VIII. Ulusal Tesisat Mühendisliği Kongresi*, (s 3), 18.

[5] Şahan, A. M. Isı Geri Kazanım Eşanjörlerinin Kullanım Opsiyonları.

[6] Şentürk, U., Altın, M., Ağar, G., Aktakka, S., & Toksoy, M. (2015). Bir Isı Geri Kazanım Cihazının Performans Karakteristiklerinin Deneysel ve Sayısal Olarak Belirlenmesi. *Engineer & the Machinery Magazine*, *56*(667).

[7] Altın, M., Aktakka, S., Günerhan, H. & Karakoç, T.H. (2013). Farklı İklim Bölgelerine Ait Isıtma-Soğutma Tasarım Sıcaklıklarının Dikkate Alınarak Bir Isı Geri Kazanım Sisteminin Tasarımı. *11. Ulusal Tesisat Mühendisliği Kongresi*, 1055-70.

[8] Şentürk, U., Aktakka, S., & Toksoy, M. (2015). Havalandırmada Enerji Geri Kazanımı: Araştırma ve Geliştirme. 12. *Ulusal Tesisat Mühendisliği Kongresi*, 301-327.

[9] Atmaca, M., & Ezgi, C. (2019). Three-dimensional CFD modeling of a steam ejector. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 1-12.

[10] Atmaca, M., Girgin, I., & Ezgi, C. (2016). CFD modeling of a diesel evaporator used in fuel cell systems. *International Journal of Hydrogen Energy*, *41*(14), 6004-6012.

[11] Atmaca, M., Çetin, B., & Yılmaz, E. (2019). CFD analysis of unmanned aerial vehicles (UAV) moving in flocks. *Acta Physica Polonica A*, *135*(4), 694-696.

[12] Kaya, O. (2013). *Havadan havaya ısı geri kazanım cihazları için ts en 308 standardına göre test düzeneği tasarımı* (Doctoral dissertation, DEÜ Fen Bilimleri Enstitüsü).