Ingeniería Mecánica. Vol. 22. No. 1, enero-abril, 2019, p. 07-13 ISSN 1815-5944

Original Article

Determination of the optimum louver angle of a louvered fin with elliptical tubes

Determinación del ángulo de la veneciana óptimo de una aleta veneciana con tubos elípticos

Alberto Menéndez-Pérez [I,*], Michel Textor[II]. Lenin Pita-Cantos[III], Rubén Borrajo-Pérez[I]

I. Universidad Tecnológica de la Habana José Antonio Echeverría. Centro de Estudio de Tecnologías Energéticas Renovables. CETER. La Habana, Cuba

II. Technische Hochschule Mittelhessen, THM. Giessen, Germany

III. Universidad Técnica de Manabí. Facultad de Ciencias Matemáticas, Físicas y Químicas. Portoviejo. Ecuador

*Corresponding author: amenendez@mecanica.cujae.edu.cu

Received: 31 de october de 2018 Accepted: 6 de december de 2018

Abstract

Heat exchangers are an essential component in many of the vital processes of our daily lives. The following work is directed to the study of compact heat exchangers, specifically those with elliptical tubes and louver fins. Through numerical simulation it is intended to obtain the thermo-hydraulic behavior of louver tube-fin geometries. Model inlet velocities corresponding to low Reynolds numbers (less than 1000 based on hydraulic diameter) are studied. The behavior ofthese geometries is simulated through the use of Computational Fluid Dynamics techniques. Establishing of the necessary boundary conditions to solve the equations as well as meshing of all models are described. The fin efficiency was determined according to the Schmidt method and the different models were compared regarding their thermohydraulic behavior.

Key words: louver fins, enhancement of heat transfer, numerical simulation, CFD**.**

Resumen

Los intercambiadores de calor constituyen un componente esencial en muchos de los procesos vitales de nuestra vida cotidiana. El siguiente trabajo va dirigido al estudio los intercambiadores de calor compactos, específicamente a los de tubo elíptico y aleta veneciana. El objetivo final es obtener el comportamiento termo-hidráulico de geometrías tubo-aletas venecianas, a través de simulación numérica. Se estudian velocidades de entrada al modelo correspondientes a bajos números de Reynolds (inferiores a 1000 basados en el diámetro hidráulico). El comportamiento de estas geometrías es simulado mediante el empleo de técnicas de Dinámica de Fluidos Computacional. El mallado de los modelos es descrito y se establecen las condiciones de contorno necesarias para la solución de las ecuaciones. Se determinó la eficacia de la aleta según el método de Schmidt y se realizaron comparaciones del comportamiento termo hidráulico entre los diferentes modelos estudiados.

Palabras clave: aletas venecianas, intensificación de la transferencia de calor, simulación numérica, CFD..

How to cite:

Menéndez Pérez A, Textor M, Pita-Cantos L, et al. Determination of the optimum louver angle of a louvered fin with elliptical tubes. Ingeniería Mecánica. 2019;22(1):7-13. ISSN 1815-5944.

Introducción

There are multiple types of heat exchangers, which differ, among other characteristics, by their size, flow arrangement, surface arrangement and construction. The so-called compact heat exchangers are widely used in industry, automotive transport, refrigeration industry and in multiple applications of medicine. They are devices that are characterized by a small volume and a low relative weight if one considers the amount of heat they are capable of transmitting.The geometry that presents elliptical tube is one of the most studied non-circular provisions, however, the information about these is not complete due to the large number of parameters that affect the performance of the heat exchanger that uses this geometry.

Burkova *et al* [1] did an extensive study for elliptical tubes in staged distribution, looking for the influence of the number of rows. It was carried out following the classic method of complete simulation and the Reynolds numbers used were within a turbulent flow regime.

Pérez *et al* [2] experimentally characterized models of heat exchangers with elliptical tubes and smooth fins. The Reynolds number and spacing varied within the laminar regime. The experiments were developed in an open circuit wind tunnel using naphthalene sublimation and heat and mass analogy. They were the first correlations that were applied to a wide range of spacings.

Han, H. *et al* [3]carried out a study, in which they investigate numerically by examining different elliptical and circular tubes the heat transfer characteristics of tube heat exchangers, for three different tube configurations. The results reveal that the use of the elliptical tube, not only can reduce the flow resistance but also improve the heat transfer capacity of the heat exchangers, which improved the efficiency of the fin. When comparing a fin with elliptical tube with one with a large circular tube, it is found that the heat transfer rate of the fin with elliptical tube increases by 4.9 %, while the heat loss decreases by 31,8 %.

In a literature review by Jacobi [4], that was published in 2001 and is covering more than 150 articles, the most studied geometric ranges of compact heat exchangers are presented. Generally, louvered fins are sized to be used in refrigeration or air conditioning equipment, which are under the most common applications of these heat exchangers. The louver angleinfluences the flow direction in the area where they are featured. Several authors have reported that the length required for the flow to be fully developed is a function of the Reynolds number and the passage between fins[5, 6].

In order to find better geometries, optimization techniques have been used. Jang and Chen [7] carried out a study concerning the bestlouver anglefor a heat exchanger utilizing louvered fins with the simplified conjugate gradient method. In order to decrease the area of the fin an optimum angle of 28,6º was obtained as a result. They validated their results experimentally using infrared thermometry.

Although there are many works devoted to the study of louver fins, most of these refer to fins that use circular tubes. The present study focuses on combining the configuration of elliptical tubes with louver fins, especially the so-called: x-shaped. The louver angle and the speed of air entering the channel between fins will vary. By means of meta-heuristic techniques we will obtain what is the optimum angle and at what speed it would present the best conditions considering the heat transfer and the pressure drop.

Methods and Materials

As in every numerical investigation two subject are very important, the first is the physical and computational domain, the other, are the boundary conditions applied to every surface limiting the model frontiers. In this section governing equations and operation condition are declared too. All aspects here considered are designed to complete the numerical problem formulation. Below in table 1 are shown the basic dimensions of the models to be studied. figure 1 shows the louver angle and the louver pitch.

Table 1. Dimensions of louvered fin models for one row of elliptical tubes

Description	Nomenclature	Value
Fin pitch (mm)	*Fp*	1,52
Longitudinal pitch (mm)	*L*	21,6
Transversal pitch (mm)	S_t	8,9
Louver angle (º)	*α*	Variable
Fin thickness (mm)	*ft*	0,125
Larger diameter of the elliptical tube (mm)	*R1*	11,01
Minor diameter of the elliptical tube (mm)	*R2*	4,94
Distance between the fin exit to the channel outlet (mm)	*y*	200
Distance between the entrance to the channel to the fin (mm)	*x*	10
Louver pitch (mm)	L_P	1,85
Fin Material	aluminum	

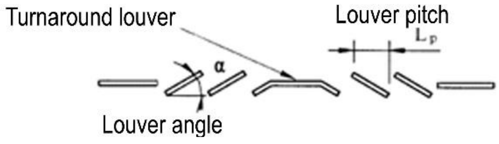

Fig. 1. Side view of the louvered fin

The phenomenon of the movement of a fluid and the heat exchange between the fluid and the surfaces that it contacts are studied. Most important for that are the equation of continuity, the equations of momentum in each of the axes and the equation of the Energy. Assumingsteady state, incompressible flow in a laminar regime with constant properties and no viscous dissipation,these equations are expressed in the same order in which they were mentioned, as follows equation 1, 2 and 3:

$$\frac{\partial u_i}{\partial x_i} = 0 \qquad (1)$$

$$\rho\left[\frac{\partial u_j}{\partial t}+\frac{\partial u_j u_i}{\partial x_j}\right]=-\frac{\partial p}{\partial x_i}-\frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

$$c_p\rho\left[\frac{\partial T}{\partial t}+\frac{\partial u_j T}{\partial x_j}\right]=\frac{\partial}{\partial x_j}\left[\lambda\frac{\partial T}{\partial x_j}\right] \quad (3)$$

The solution of equations 1, 2 and 3 are achieved for a computational domain shown in figure 2. The domainhas been extended in the input and output regions of the model. The extension length is seven times the length of the spacing between fins in the entry direction and about ten times the length of the fin in the exit direction.

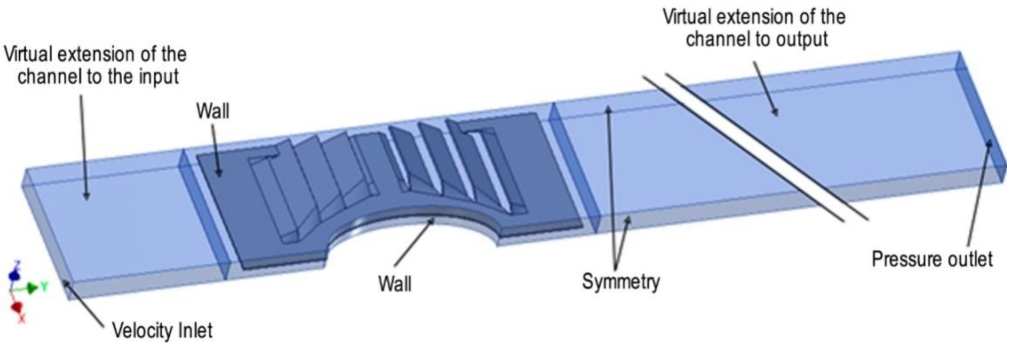

Fig. 2. Representation of channel and fin boundary conditions

The need to have a uniform and one-dimensional velocity profile at the entrance of the model, as well as to avoid the existence of reverse flow in the exit section, is the reason for these extensions [8]. The fin in the central region of the domain is taken as a solid, while the channels above and below it is considered as fluid regions. Two boundary conditions are needed, the temperature in the tube and the parameters of the fluid at the channel inlet. The velocity inletwill be varied to study a range of Reynolds number. The boundary conditions for each surface are summarized below:

At the inlet of the model, equation 4:

$$v = w = 0 \quad u = const. \quad T = const \quad (4)$$

In the upper and lower part of the domain, a periodicity conditionwas considered. On the surface of the fin, conjugated heat transfer is solved and a not slip condition is considered:

On the surface of the tube we will have, equation 5:

$$u = v = w = 0 \quad T = const. \quad (5)$$

In both sides of domain a simetry condition was applied, equation 6:

$$\frac{\partial u}{\partial y}=\frac{\partial w}{\partial y}=\frac{\partial T}{\partial y}=v=0 \quad (6)$$

At the outlet of the model, equation 7:

$$\frac{\partial u}{\partial x}=\frac{\partial v}{\partial x}=\frac{\partial w}{\partial x}=\frac{\partial T}{\partial x}=0 \quad (7)$$

The flow is considered inside the laminar regime because the air velocities in the channel are very low, to avoidkinetic noise. Calculations are made in steady state. The flow to analyze is three-dimensional, with velocity and temperature fieldsuncoupled, then the temperature field can modify the velocity field. The flow is incompressible, and with constant physical properties.The operating conditions are listed below:

1. Gage pressure: 0 kPa.
2. Air density: 1.225 kg/m^3.
3. Absolute viscosity: 1,7894 x10^5 kg/m·s.
4. Velocity inlet: ranging from 0.5 to 5 m/s.
5. Air temperature at the inlet of the model: 300 K.
6. Tube wall temperature: 333 K.
7. Thermal conductivity: 0.0242 W/m·K
8. Air specific heat at constant pressure: 1006.43 J/kg·K

The region of the geometry and the channel are meshed using hybrid tetrahedron type elements (TGrid) with Gambit 2.4.6 software. It is meshed in such a way that it produce a solution independent of the mesh, since the difference between solutions does not exceed 0.6 % between two successive meshes with different density. The mesh used does not have any skewness element, nor any element with negative or inverted volume.

Validation of the numerical model

The numerical model used in this paper need to be certified before it could be applied for research. The method used here to certificate the model was to build a computational model with similar dimensions and characteristics that a geometry which had already been researched and its results are available in the literature. A geometry studied by Han *et al* [9] was selected, which wasvalidate using experimental results by Wang *et al* [10]. Han *et al* [9] state that the average deviations of the friction factor (f) and the Colburn factor (j) between their numerical model and Wang et al' results are 9,3 % and 4,5 % respectively. The experiments of the authors were based on a similar geometry of louvered fin with elliptical tube but using louver surrounding the tube. Another difference is that the inlet air to the wind tunnel has a temperature of 308 K and the wall temperature of the tubes is 353 K. The same flow regime, interpolation procedures, boundary conditions and convergence criteria were used. The coupling pressure-velocity is implemented using the SIMPLEC algorithm.

In figure 3 it is observed that for the values of Δp there are a good match, although the numerical model underestimates slightly this parameter. The behavior of the heat transfer coefficient is also quite approximated by the numerical model. The difference between the behaviors of these curves can be producedbecausethe geometry construction shown in the work of Han et al [10] is not realistic.It doesn'treproduces all details of the model nor the correct shape. The h-curve of this work overestimates for speeds above 1.5m/s while underestimating h for speeds below 1,5 m/s when compared to those of Han et al. [9]. From this comparison it can be stated that the method, the simplifications, as well as the boundary conditions used in this work are enough to achieve the proposed objectives.

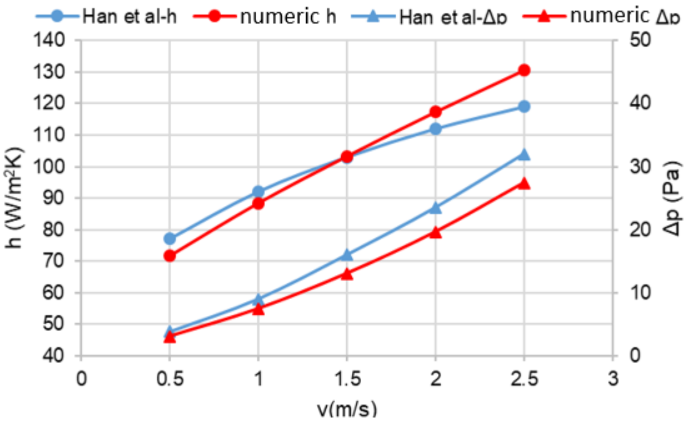

Fig. 3. Validation of the coefficient of global heat transfer and pressure drop [11]

Data reduction

The Reynolds number was determined using the velocity in the minimum flow area u_{min} while the characteristic length is the hydraulic diameter of the channel D_h equation 8

$$Re = \frac{\rho u_{min} D_h}{\mu} \tag{8}$$

The heat transferred in the surface can be calculated with the temperature difference between the input and output sections of the model (T_{out} -T_{in}), the mass flow m_a, and the specific heat of air c_{pa}, according to, equation 9:

$$Q = m_a c_{pa} (T_{out} - T_{in}) \tag{9}$$

When the fluid undergoes a phase change inside the tubes, it is common practice to consider a high value for the internal heat transfer film coefficient. By assuming this, the overall coefficient will be obtained only considering the transfer coefficient from the external part and the conduction inside the pipe wall. The temperature of the internal wall of the tubes is considered constant and with the same value as the circulating fluid. The heat transferred can also be calculated through the known equation that involves the logarithmic ΔT_{ln} (LMTD), knowing the heat transfer area A_f, and the global heat transfer coefficient h, equation 10:

$$Q_h = \eta_0 \bar{h} A_f F \Delta T_{ln} \tag{10}$$

The correction factor of the LMTD, F, was considered null because one of the fluids has a constant temperature. The global heat transfer coefficient is calculated considering the equations 10 and 11, then Q = Q_h.

The efficiency of the fined surface η_0 is a function of the heat transfer coefficient and is calculated as a function of the fin efficiency η and the total heat transfer area A_0.

$$\eta_0 = 1 - \frac{A_f}{A_0}(1 - \eta) \tag{11}$$

The efficiency of the fin was determined following the Schmidt method [12], as many authors have did [13-15].

The efficiency of the fin and the global coefficient of heat transfer have an implicit formulation therefore an iterative process is needed. The conditions of the same heat in equations 10 and 11 must be satisfied. For a fixed geometry there is only one pair of values of these magnitudes, fin efficiency and global heat transfer coefficient, which fulfill this condition. Then the Colburn factor can be obtained knowing the Prandtl number and the friction factor according to, equations 12 and 13:

$$j = \frac{\bar{h}}{\rho_m c_{pa} u_{min}} Pr^{2/3} \quad (12)$$

$$f = \frac{\Delta p}{0.5 \rho_a u_{min}^2} \left(\frac{A_{min}}{A_t}\right) \quad (13)$$

Where A_{min} is the minimum flow area and A_t is the heat transfer area.

As a method of comparison to determine how effective a geometry is, the criterion known as area goodness [16] will be used, equation 14. That is simply to determine how much the Colburn factor (j) increases in relation to the increase in the friction factor (f). Comparing the same relation but for the reference fin (see Fig. 5)

$$area\ goodness = \frac{j/j_0}{f/fo} \quad (14)$$

Results and Discussion

Next, the results obtained by means of the procedures set forth above are discussed. The results will be analyzed taking into account the behavior of the values of j and f, which are the indicators generally used to characterize the thermo-hydraulic behavior of a heat exchange surface.It is important to mention that the Colburn factor (j) is a dimensionless representation of the heat transfer coefficient, it relates to the coefficient of average heat transfer, the air thermal conductivity, the Prandtl number and velocity in the minimum flow area. Similarlythe pressure drop is represented by the Darcy (f) friction factor.

From the results of the numerical simulations in CFD, the Colburn and friction factors are calculated according to equations 12 and 13 respectively. Using the values of j and f determined above, a surrogate model is made using a multiple regression technique, where the independent variables are the inlet velocity to the model and the louver angle. The regression obtained must have a statistically significant relationship between the variables with a confidence level greater than 95.0 %. Otherwise, we proceed to increase the number of sample points that must be simulated using CFD.

Figure 4 shows the values of the Colburn factor (j) and the friction factor (f) as a function of the velocity inlet into the model and the louver angle of the louvered fin.

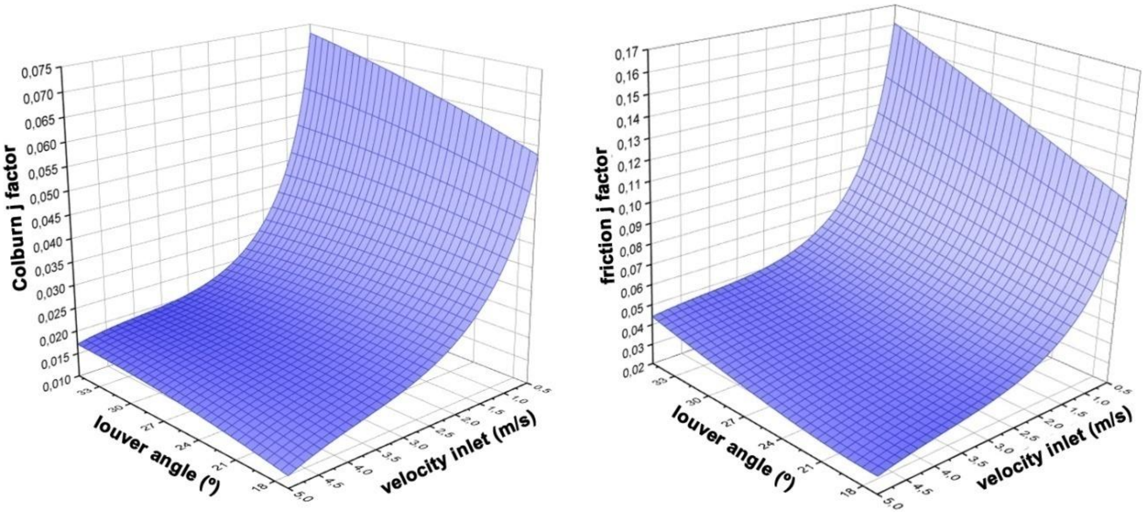

Fig. 4. Colburn factor (left) and friction factor (right) as a function of inlet velocity and louver angle

It is appreciated that for eachlouver angles j is decreasing when the velocity increases, as it corresponds to the heat exchangers.

One of the fundamental objectives of this type of research is achieving increments in the total heat exchanged in the fins of a compact heat exchanger, without paying a high price in head losses. The iterative process of multi-objective genetic algorithms achieves a better approach to this end. They obtain the set of parameters that result in one or more models with a value of j higher than the reference, without significantly increasing f. Based on the Pareto set, the genetic algorithms generate a random initial population, which is subjected to crosses and mutations obeying the objective functions (j and f), previously determined. That is, they maximize j while minimizing f. In this study, multiple regressions that respond to j and f are optimized simultaneously.

In table 2, the values found by the Pareto set were shown for which the values of j were maximized while the results of f were minimized. These were compared with a surface having elliptical tube but plate fin, as shown in figure 5.

Table 2: Result set product of the optimization by genetic algorithms

Model #	Ang(°)	Vel(m/s)	*j*	*f*
1	33,13	1,52	0,0318	0,0703
2	33,69	0,50	0,0720	0,1570
3	28,93	1,09	0,0405	0,0835
4	21,20	4,99	0,0143	0,0295
5	25,67	3,52	0,0197	0,0400
6	17,00	0,50	0,0581	0,1104
7	18,36	0,67	0,0506	0,0955
8	17,30	0,61	0,0524	0,0985
9	34,87	4,72	0,0178	0,0449
10	18,33	1,05	0,0392	0,0728
11	21,01	4,37	0,0166	0,0330
12	17,46	4,37	0,0153	0,0320

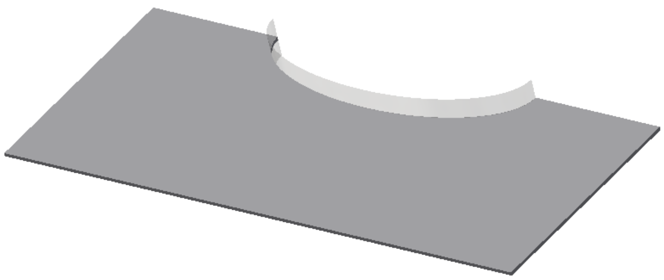

Fig. 5. Plate fin with elliptical tube

The relationship between the Colburn factor of each optimized model and the Colburn factor of a flat fin operating at the same inlet velocity is shown below. This graphic is made to determine which is the model increasing its heat transfer compare to the reference fin. The Model labeled 9, with alouver angle of 34,87 and an approximate velocity inlet of 4,7 m/s, it presents the best results doubling the Colburn factor. Although as shown in figure 6, the pressure drop increases more than 250 %, which would result in a model with a low performance. This low performance is consistent with the low value of area goodness presented by Model 9 in figure 7. This model, having a high louver angle, presents a greater redirection of the air flow that circulates through the channel formed between two consecutive fins. This phenomenon achieves a higher mixing of fluid with different temperatures and decreases the thickness of the thermal boundary layer developed in each louver.

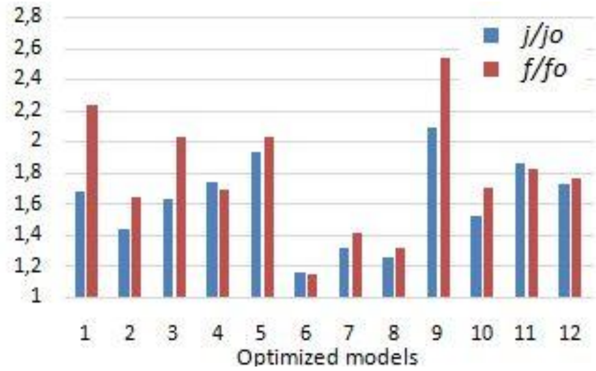

Fig.6. Ratio between the Colburn factor and friction factor of each optimized model and the Colburn factor of a flat fin operating at the same inlet velocities

The model with the least increase in pressure drop is the one with the smallest louver angle among the studied ones,as happen withthe Model 6, which has a louver angle of 17º and a velocity inlet of 0,5 m/s. This behavior was to be expected, since, being the louvers less confronted with the flow, they present less drag force and therefore less pressure drop.

It is a known fact that louvered fins have a higher heat transfer than plate fins (reference fin). Figure 6 shown, that all optimized models have a j/j_o coefficient higher than one. This increase however comes coupled with an increase in pressure drop, which generally is some percentages higher than the percentages of the heat transfer increase. In this work as the result of optimization there were three models obtained, which met the goal of increasing the heat transfer without a significant increase in pressure drop. In figure 7 it is shown that the models 4, 6 and 11 have *area goodness* values higher than one. It seems like the models with approximately 21° of louver inclination and velocities higher than 4,3 m/sare between the most effectives, according to the performance enhancement criteria.

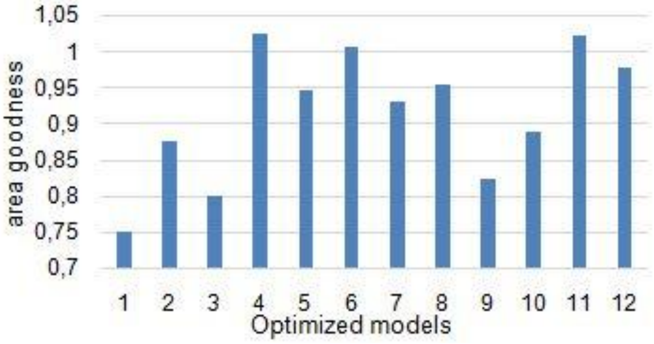

Fig. 7. Area goodness for every optimized model

Conclusions

The multiobjective optimization method of using genetic algorithms demonstrated to be a useful technique in these kind of researches. Three of the models in Pareto set, as a result of optimization, are showing *area goodness* values higher than one. Some Models achieved to increase j up to 200 %, compared to a plate fin. Finally, the objective of find the best combination between louver angles and velocity inlet was reached for 21° louver angle value and velocity inlet over 4,3 m/s. That combination produces models having the optimal thermo-hydraulics behavior.

Referencias

1. Burkov V Medvedskii VP Kochegarova IY, et al. Studies of heat transfer aerodynamics in bundles assembled from oval tubes. Thermal engineering. 2010;57(3):227-31.
2. Pérez RB, Yanagihara JI, González Bayón JJ. Thermal friction drop characteristic of heat exchangers with elliptical tubes smooth fins. Ingeniería Mecánica 2012;15(3):243-53
3. Han H, He YL, Li YS, et al. A numerical study on compact enhanced fin--tube heat exchangers with oval circular tube configurations. International Journal of Heat Mass Transfer. 2013;65:686-695.
4. Park YG, Jacobi A. Air-Side Performance Characteristics of Round- Flat-Tube Heat Exchangers: A Literature Review. Analysis Comparison in Air Conditioning Refrigeration Center. USA: University of Illinois; 2001 [Consultado el: 12 de octubre de 2018]. Disponible en: http://hdl.hle.net/2142/13360
5. Franzisku Gruger J, Menéndez A, Borrajo R. Thermal-hydraulic performance of a slit fin influence of the fin pitch. Ingeniería Mecánica. 2018;21(2):59-67.
6. Välikangas T, Singh S, Sørensen K, et al. Fin--tube heat exchanger enhancement with a combined herringbone vortex generator design. International Journal of Heat Mass Transfer. 2018;118:602-16.
7. Jang JY, Chen CC. Optimization of louvered-fin heat exchanger with variable louver angles. Applied Thermal Engineering. 2015;91:138-150.
8. Qian Z, Wang Q, Cheng J, et al. Simulation investigation on inlet velocity profile configuration parameters of louver fin. Applied Thermal Engineering. 2018;138:173-182.
9. Han H, He YL, Li YS, et al. A numerical study on compact enhanced fin--tube heat exchangers with oval circular tube configurations. International Journal of Heat Mass Transfer. 2013;65:686-95.
10. Wang CC, ChiK Y, Chang CJ. Heat transfer friction characteristics ofplain fin tube heat exchangers part II: Correlation. International Journal of Heat Mass Transfer. 2000;43(15):2693-2700.
11. Menéndez A, Borrajo R, Sacasas D. Análisis de la influencia del número de venecianas en intercambiadores de calor compactos. Ingeniería Mecánica. 2016;19(3):176-184.
12. Schmidt TE. Heat transfer calculationsfor extended surfaces. Refrigeration Engineering. 1949;57(4):. 351-57.
13. Wang CC, Chen KY, Liaw JS, et al. An experimental study of the air-side performance of fin--tube heat exchangers having plain louver semi-dimple vortex generator configuration. International Journal of Heat Mass Transfer. 2015;80:281-87
14. Li J, Dang C, Hihara E. Heat transfer enhancement in a parallel finless heat exchanger using a longitudinal vortex generator Part B: Experimental investigation on the performance of finless fin-tube heat exchangers. International Journal of Heat Mass Transfer. 2019;128:66-75.
15. Garelli L, Rodriguez GR, Dorella JJ, et al. Heat transfer enhancement in panel type radiators using delta-wing vortex generators. International Journal of Thermal Sciences. 2019;137:64-74.
16. Dezan DJ, Yanagihara JI, Jenovencio G, et al. Parametric investigation of heat transfer enhancement pressure loss in louvered fins with longitudinal vortex generators. International Journal of Thermal Sciences. 2019;135:533-545.