Prediction of Turbulent Flow Using Upwind Discretization Scheme and k-ε Turbulence Model for Porous Heat Exchanger

assistant professor **Petre OPRIŢOIU***

*Technical University of Cluj-Napoca, Department of MTC, str. Observatorului, nr.72-74, 400363, Cluj-Napoca, Romania, E-mail: petre.opritoiu@mtc.utcluj.ro

Abstract: *The present investigation is concerned with the prediction of turbulent flow using upwind discretization scheme and k-ε turbulence model for porous heat exchanger application. Many modeling techniques exist for the analysis of porous foam. The Fluent porous media model employs a momentum equation which accounts for both viscous and inertial losses in the foam and includes an effective thermal conductivity in the energy equation calculated via volume-averaging the coolant and foam thermal conductivities. The FLUENT computational fluid dynamics (CFD) software v.6.0 is used for more detailed modeling of the porous heat exchanger.*

Keywords: *computational fluid dynamics, turbulence model, flow characteristics, heat transfer performance*

1. Introduction

Porous media can be used for modeling a wide variety of engineering applications. It is generally desirable to determine the pressure drop across the porous medium and to predict the flow field in order to optimize a given design. The general design strategy is to minimize the coolant flow path length in contact with the porous medium, and to minimize the friction factor in that zone while simultaneously maximizing the heat transfer coefficient.
The maximum heat flux that can be accommodated is limited by the heat transfer coefficient achievable with flowing air and the maximum allowable operating temperature of the structural materials. Porous metal heat exchangers have been studied in the past because of the large surface area they provide for heat transfer.
For a given particle dimension d_p, the pressure drop through a porous medium ΔP is highly dependent on the porosity ε, while the heat transfer coefficient tends to depend more on the specific surface area, σ. Such a porous foam would have high porosity (which governs the pressure drop) but with specific surface area (influencing the heat transfer) higher than those that a conventional packed bed of spheres can provide.

2. Model development

Most existing models for heat transfer through a porous medium seem to be based on a semi-empirical approach such as the following circuit-based model described in [1].

$$h_{eff} = \varepsilon h_p + \frac{1}{R_0 + \frac{1}{\sqrt{h_p k_p S_p}\tanh\sqrt{\frac{h_p S_p}{k_p}t}}} \tag{1}$$

where h_{eff} is the effective heat transfer coefficient, h_p is the local particle-to-fluid heat transfer coefficient, R_0 is the porous medium/wall interface resistance, k_p is the porous medium thermal conductivity and t is the porous medium thickness.
Such a model provides a quick and convenient means for estimating the overall heat transfer coefficient but is limited in its range of application, in particular to account for cases with large spatial variation of the microstructure characteristics (e.g. the porosity and directional thermal conductivity), for cases with high porosity, and/or for design configurations where entrance effects plays a major role.

It would be very useful to develop a more comprehensive and fundamental model which first calculates the velocity profile and then the corresponding temperature distribution in the porous region, with the capability to account for microstructure variation and to include potentially important processes such as the local heat transfer between solid and fluid and the effect of dispersion. This model would provide a much better tool to perform a more detailed assessment and optimization of porous media for high heat flux application.

3. Turbulent flow in open-cell aluminum foam heat exchanger - physical situation

The physical situation considered and coordinate system employed in the simulations are revealed in fig.1, with their appropriate physical quantities that characterize presented flow situation, implying a number of iterations cycles required by the computer code to attain a pre-specified level of convergence. In this case the physical situation considered corresponds to the experimental situation of [2].
The final overall dimensions of the compressed foam blocks used in pressure-drop and heat transfer simulations were 250mm×100mm×50mm, with the cross-sectional area normal to the flow direction measuring 250mm×100mm. To make them functional heat exchanger, each foam was brazed in a central position to an adjoining heat spreader plate made by solid aluminum.
A typical flow and heat transfer configuration is shown. A heat source is bonded or joined to a thin conductive substrate on which a block of open-cell aluminum foam of length L and thickness W is attached. The foam is then placed in a channel, and cooling fluid of velocity u_0 at a temperature T_∞ is pumped through the open celled material, thereby affecting heat transfer from the hot source to the cooling fluid [3,4].
Air enters the heat exchanger with a uniform velocity profile, and the flow of air develops along heat exchanger. In the inlet region, the free stream is completely surrounded by the growing boundary layer and accelerates as the thickness of the boundary layer increases.
In this work aluminum foam, with physical data: 40PPI porous density and porosity ε=0.927, acting as a heat exchanger, bonded to a heated wall set at 358K, with air initially at 295K and v=2.2m/s flowing through the foam. The coupled fluid flow-heat transfer calculation was then performed using SIMPLE algorithm and 2nd order upwind differencing on the momentum equation (fig. 2) [5].

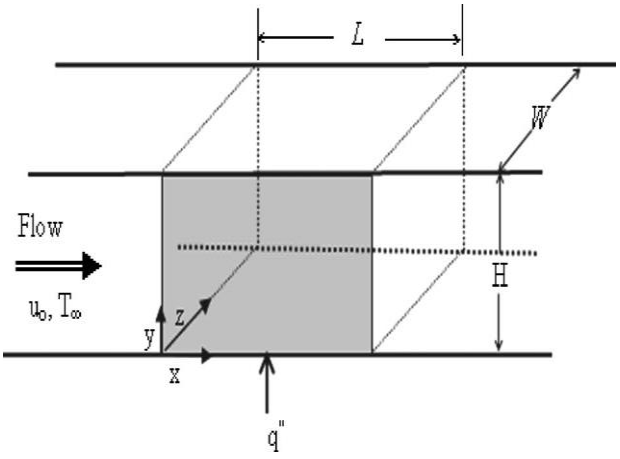


Fig. 1. Physical situations considered [5].

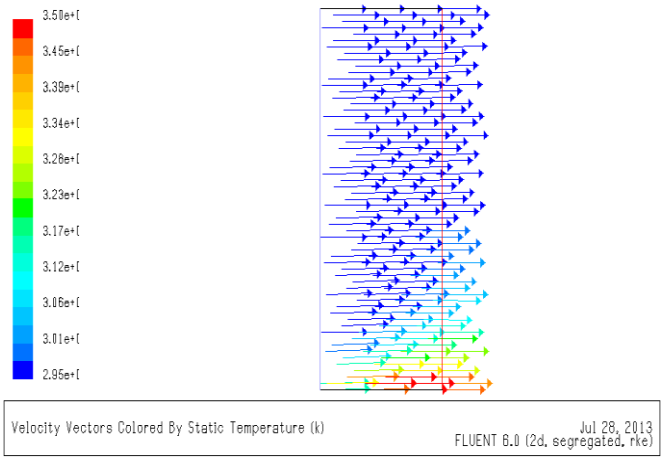
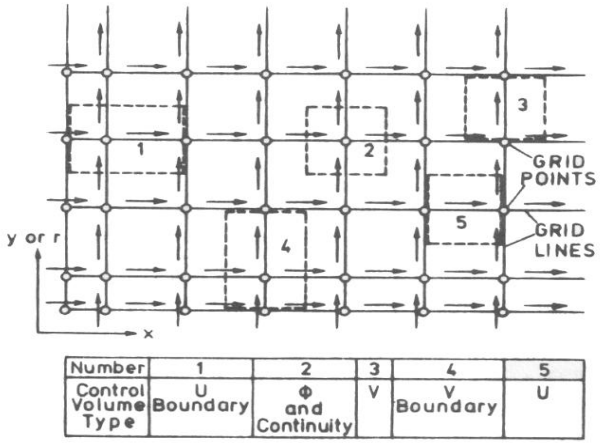


Fig. 2. Coupled fluid flow-heat transfer [5].

4. The finite difference discretization

In Fluent flow equation are solved, with their appropriate boundary conditions, by integrating them over finite–difference control volumes that form the physical integration domain considered, an example of the grid arrangement is depicted in fig. 3 where it is seen that the grid is staggered so that velocity components are situated mid-way between grid points [6].
The pressure, viscosity and any general scalar variable such as kinetic energy of turbulence, dissipation rate of turbulence kinetic energy are located at the grid points. The main advantage of

this arrangement is that the pressure difference between two adjacent grid points become the natural driving force for the velocity component located between these grid points.



Number	1	2	3	4	5
Control Volume Type	U Boundary	Φ and Continuity	V	V Boundary	U

Fig. 3. Control volume specification [6].

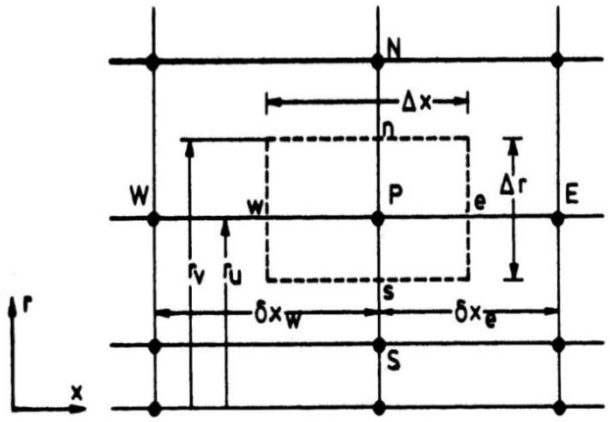


Fig. 4. Control volume for a scalar variable [6].

Now consider a single control volume for variable φ, as depicted in fig. 4, in which cell-face area at four points (e, w, n, s) is located mid-way between the grid points. The next step in the formulation of finite-difference equation is the assumption of the variation of φ between any two grid points. The diffusion terms are formulated using the central difference scheme. The schemes that are used to approximate the convection terms are only applied to the convected variable φ, the convecting velocity is discretized using the central-difference scheme.

In the ***central-difference scheme (CDS)*** the value of φ at an interface of the control volume is taken as the average value of the φ's at the grid points that lie on either side of the interface of the control volume. For velocities that are low enough, central-difference scheme is recommended, however it has been found that when the grid Peclet number is greater than 2, the coefficient matrix becomes non-diagonally dominant. As a consequence, a semi-implicit type numerical scheme, as normally used, becomes unstable.

The ***upwind-difference scheme (UDS)*** recognizes that the weak point in the CDS formulation is the assumption that the convected property φ at an interface is the average of the φ's at the grid points that lie on the either side of the interface, and it proposed a better resolution. That is, a piecewise-linear variation of φ between grid points is assumed for the diffusive flux, while for the convective flux the value of φ convected across an interface is taken to be the value of φ at the grid point on the upwind side of the face.

This approximation overcomes the stability problem associated with the use of the CDS. In the present study the upwind difference scheme is also employed for velocities that are low enough.

5. The wall functions

In the near-wall region, there is a steep variation in the fluid properties. To avoid the need for detailed calculations in these regions, algebraic relations are employed to relate the values of the dependent variables at a point on the wall to those at a point adjacent to the wall, a logarithmic layer is presumed to exist between these two points. FLUENT offers several discretization techniques for the convective terms of each governing equations. Using the segregated solver, the operator may choose to have either first or second order discretization of terms.

The first order method computes the solution at the center of each cell, while the second order method computes the solution at the center of each face. The first order discretization is generally acceptable for simple flow when the grid is aligned with the flow and a quadrilateral or hexagonal grid is in place.

A second order discretization method reduces errors over the first order methods, while generally increasing the difficulty of obtaining a convergent solution [7,8]. Reynolds number is used to determine y^+, a dimensionless distance from the wall. The implementation of wall functions is

necessary to overcome short falls in k-ε turbulence models. A criterion for the validity of wall functions exists. Acceptable range is $y^+ > 30$ (fig.5). Wall functions must also be implemented for the kinetic energy (k) and the turbulent dissipation rate (ε).

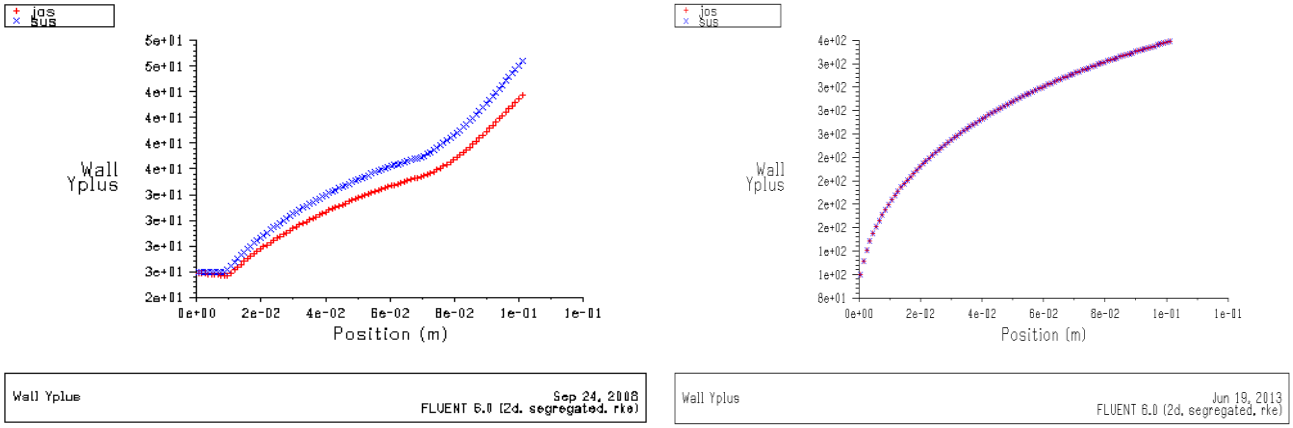


Fig. 5. Wall y plus function for open-cell aluminum foam heat exchanger: a. cooled by air, b. cooled by water [5].

6. Boundary conditions

In turbulent flow computations, additional boundary conditions for turbulence parameters need to be specified at inlet and outlet locations. This information can be supplied in the form of convenient, derived quantities such as turbulent intensity, length scale, viscosity ratio, hydraulic diameter, etc. It is important to review the default turbulent boundary condition (K=1.0, Epsilon=1.0). If these default settings are not representative for the flow field, error will be introduced into the solution.
The boundary condition set for the inlet is velocity. For this study, an inflow of mean velocity 2.2m/s was used. To generation turbulence an additional condition had to be applied. This involved turbulence intensity (I), which was calculated by eq. (2), [5,9].

$$I = 0.16\left(\mathrm{Re}_K\right)^{-1/8} \tag{2}$$

and the hydraulic diameter (D_h). The boundary condition set for the outlet is pressure. A viscous and an inertial resistance are applied in all directions. Porous media model is nothing more than an added momentum sink in governing momentum equations. Since the volume blockage that is present physically is represented in the model, FLUENT uses and reports a superficial velocity inside the porous medium, based on the volumetric flow rate, to ensure continuity of the following properties are required:
(a) Porosity (ε); (b) Viscous resistance (1/K), for aluminum foam:

$$\frac{1}{K} = \frac{32\pi}{\varepsilon \times d_p} \tag{3}$$

where K- permeability, [m^2] and d_p-pore diameter,[m]; (c) Inertial resistance (c_F):

$$c_F = \frac{2\left(\Delta P - \dfrac{\mathrm{v}L\mu}{K}\right)}{\mathrm{v}^2 \rho L} \tag{4}$$

The theoretical pressure drop per unit length for porous media was predicted following Forcheimer equation (1901):

$$\frac{\Delta P}{L} = \frac{\mu}{K} \cdot \mathrm{v} + \frac{c_F}{\sqrt{K}} \rho \cdot \mathrm{v}^2 \tag{5}$$

where Δp/L-pressure drop per unit length, [Pa/m], μ- fluid viscosity, [kg/m*s], ρ- fluid density, [kg/m^3], v-velocity, [m/s]. The hydraulic diameter is determined based on the size of the compressed porous cell, of the metal filament's diameter and the porous density. Boundary layers play an important role for heat transfer. The shape of the crossing section is determined by the metal filaments and it is meant to grow the local turbulence of fluid and heat transfer when it flows into the porous metal. Turbulence intensity values and resistance coefficients of viscous and inertial type, required for running the FLUENT software for a porous medium were obtained using MATHCAD program, starting from equations specific to Brinkman's porous environment [10].

7. Some computational details

A mesh which provides accurate results at laminar flow conditions may not be acceptable for turbulent flow situations. The turbulent boundary layer can be subdivided into several regions. Based on the region that needs to be resolved, the location of first cell adjacent to the wall is determined. When flow characteristics in the viscous sub-layer need to be captured,
Enhanced wall treatment should be used. Standard wall functions can be employed when the flow resolution starts from the log-layer region. Depending on the choice of near wall treatment, some constrains on the placement of the first cell from the wall are imposed, as prescribed in the following table.
When generating the mesh, care should be taken such that the first cell adjacent to the wall doesn't fall in buffer layer. Cell height calculations are based upon the cell centroid location. Enhanced wall treatment is recommended for the accurate prediction of frictional drag, pressure drop, separation, etc. Even here the mesh quality is acceptable, and no significant numerical issues have been encountered in solving the problem with a converged solution talking slightly over 200 iteration (fig.6a and b).

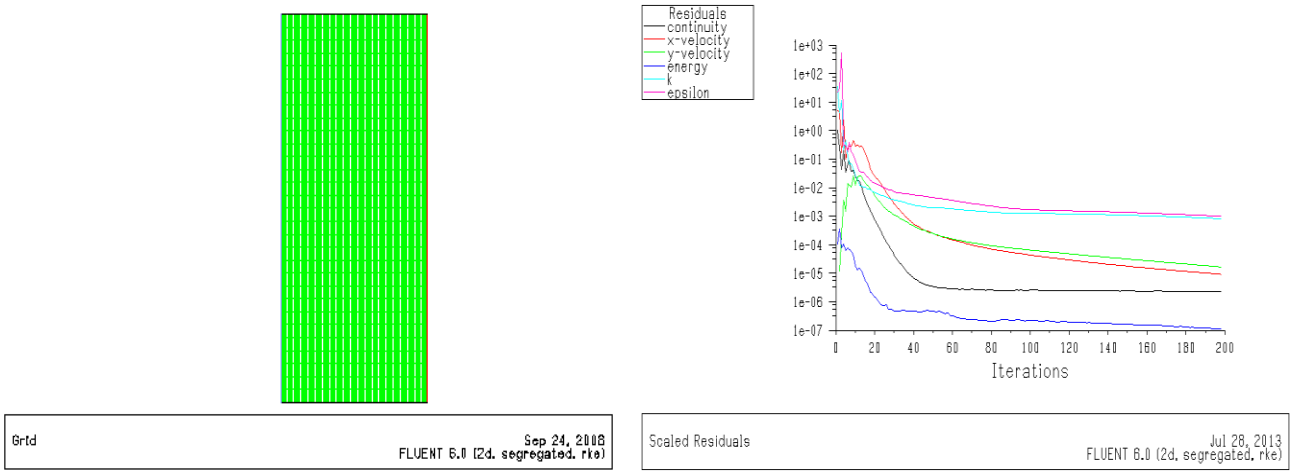


Fig. 6. Mesh (a) and plot of residuals (b) [4].

It is elegant and efficient to create mesh in one process. This also is more robust when topologically complex problems such as the open-cell metallic foam are being investigated. This means that the mesh generated required no intervention or correction so that solution is, as in flow/heat transfer presented here straightforward [4].

8. The turbulence model

The turbulence model employed in the present calculation is the k-ε model [5,7], and it necessitates the solution of two equation of transport for two turbulence quantities namely

turbulence kinetic energy k, and its rate of dissipation ε (fig. 7). Knowledge of the local values of k and ε allows the evaluation of a local effective viscosity from each the turbulent shear stresses are calculated.
A comparison of turbulence models was conducted. The principle underlying this was to compare the models and thus confirm their validity. k-ω Standard and k-ω SST models are not adequate for this case. Turbulence models offered similar results of simulation, with exception of Spalart-Allmaras model (S-A), Laminar model and Reynold Stress Model (RSM) [5]. The realizable model provides the best performance of all the k-ε model versions for several validations of separated flows and flows with complex secondary flow features (fig. 8).

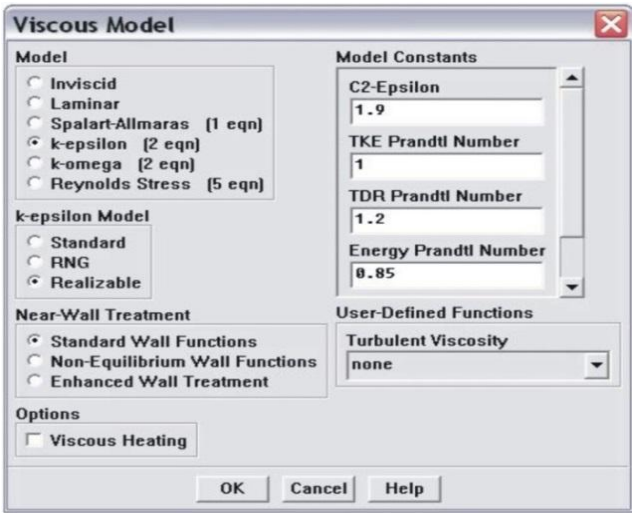


Fig. 7. k-ε realizable turbulence model [5].

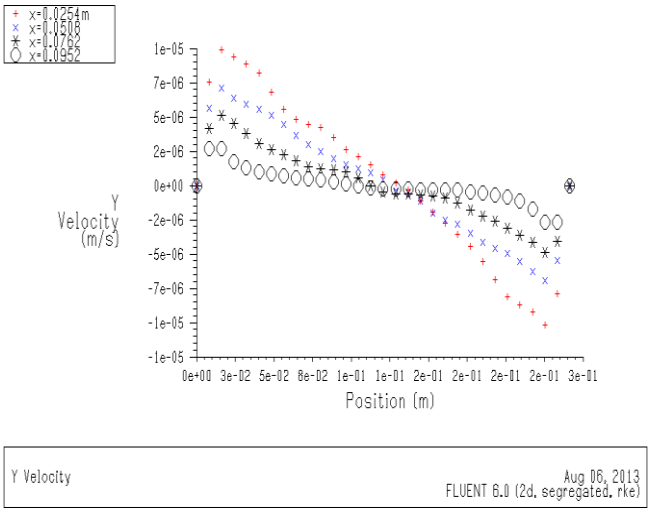


Fig. 8. k-ε realizable Model [5].

Developing turbulent heat exchanger flow is basically a transition from a boundary layer type flow at the entrance to a fully developed flow downstream. The free stream in the inlet region is completely surrounded by the boundary layer, which by diffusion of momentum through laminar and turbulent mechanisms grows in thickness as the distance from the inlet increases. The growing boundary layer accelerates the free stream which eventually losses its identity as the boundary layer merges with itself. Following the disappearance of free stream, further changes occur in the velocity distribution and turbulence structure until the flow attains fully developed state.

9. Numerical solution procedure

The numerical solution procedure employed to solve the finite-difference equations was the well know SIMPLE algorithm. This algorithm was embodied in the general two–dimensional computer code FLUENT [7]. In this computer code the flow variables are calculated in a semi-implicit line-by-line fashion over a staggered finite-difference grid system. Owing to the semi-implicit nature of the code, under relaxation factors are used. The first order discretization was used for all simulation in this study.
Pressure and velocity were coupled with the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm. SIMPLE uses a relationship between velocity and pressure corrections to obtain mass conservation and a pressure field. The pressure discretization scheme was the default value standard. Solutions control under-relaxation factors for porous media: Pressure 0.3, Density 0.95, Body forces 0.95, Momentum 0.7, TKE 0.6, TDR 0.6, Turbulent viscosity 0.6, Energy 1 (fig. 9).
The flow field of axisymmetric expansion in a porous heat exchanger is a complicated phenomenon characterized by flow separation, flow recirculation and flow reattachment. As illustrated in the fig.10 such a flow field may be divided by a dividing surface into two main regions, one being the region of recirculation flow, the other being the region of main flow.

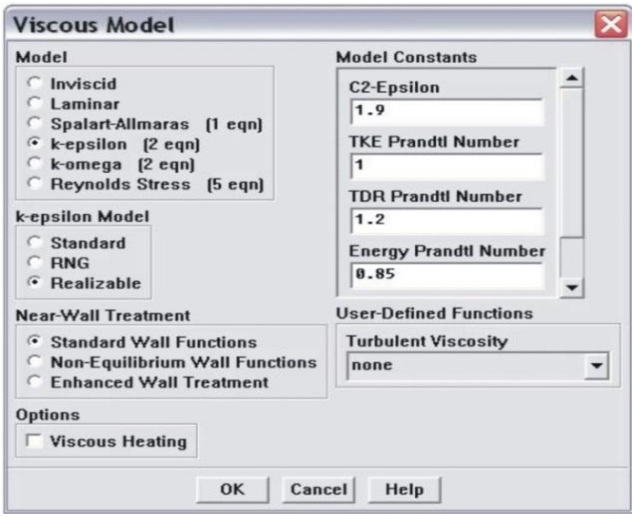


Fig. 9. Constants used in the k-ε turbulence model [5].

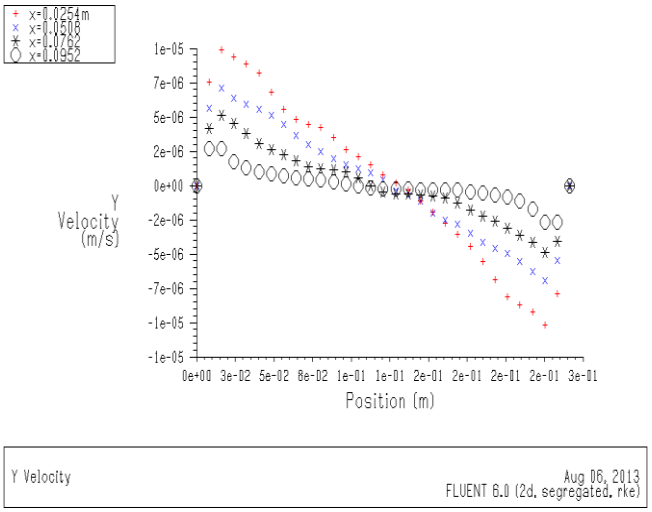


Fig. 10. Contours of velocity vectors vs turbulence intensity [5].

The point at which the dividing surface strikes the wall is called the reattachment point. In the recirculation zone, the high adverse pressure gradient results in reverse flow and promote turbulence. Eddies generated in the recirculation zone and in the vicinity of the reattachment point can be considered as highly concentrated source of turbulence. The subsequent convection, diffusion, and decay of turbulent eddies have a dominant influence on the characteristic of mean flow.

10. Model validation

The working method used, numerical simulation using Fluent software, for predicting turbulent porous heat exchanger flow was verified by comparison with experimental data reported in the literature [11]. The experimental work of [2,12] forms the basis for comparison.
One of the primary reasons to study forced convection in metal foams is to provide information necessary for the possible applications of these materials in electronic cooling and other thermal systems. The fibers of these materials could be thought of as a network of complex extended surfaces giving the advantage of increasing the interfacial area. In addition to the increased interfacial area the formation of eddies or fluid mixing promotes the heat transfer enhancement.
Validation of the flow is achieved through comparative research conducted on the distribution of pressure drop. Flow parameters are also validated, so the pressure drop calculated by simulation has a deviation of 1.31 – 8.27%, the relative error grows if the porosity has low values.

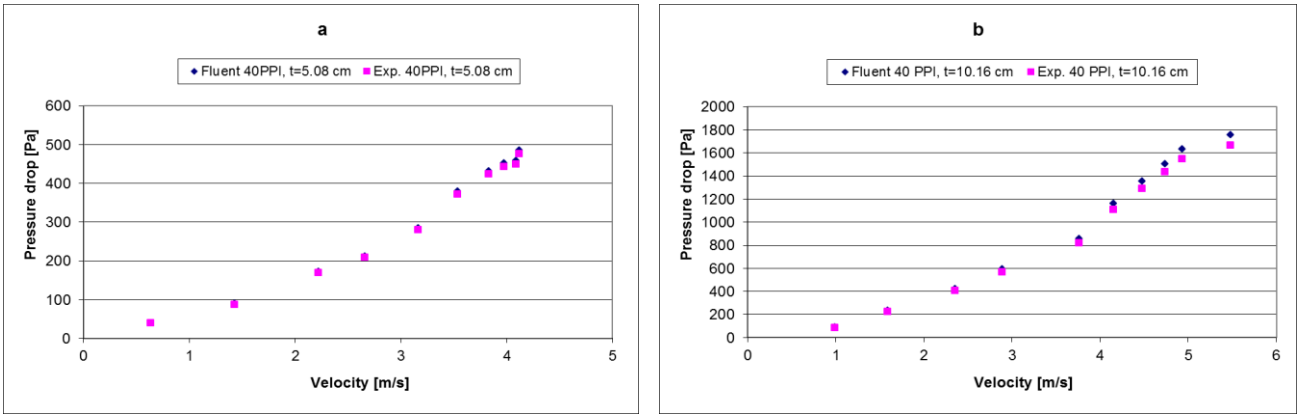


Fig. 11. Comparison between simulation and experimental pressure drop data at different velocities, for 40 PPI porous density, porosity ε=0.927 and heat exchanger thickness t= 5.08(a) and 10.16(b) [11].

The pressure drop is also influenced by the porosity, the density of porosity and the fluid velocity and thickness. At the same porous density and porosity the pressure drop is influenced by the flow rate of working agent and thickness. Relative error increased with thickness (fig. 11) [11].
Thermal convection coefficient values determined by simulation with FLUENT are also experimentally confirmed, the relative error between them is 0.54 ─ 10.23%. They depend on the flow rate, the nature of the agent flow (air or water), porosity, pore density and thickness of the heat exchanger.
The convective heat transfer is more intense near the limit surface between the solid aluminum heated board and the porous aluminum heated board, due to thermal contact resistance and low porosity in that area.
It is noticed that at the same porosity, the thermal convection coefficient is higher if the porosity density is higher, the difference between them increases with decreasing speed. Relative error is bigger if porous density decreased (fig. 12) [11].

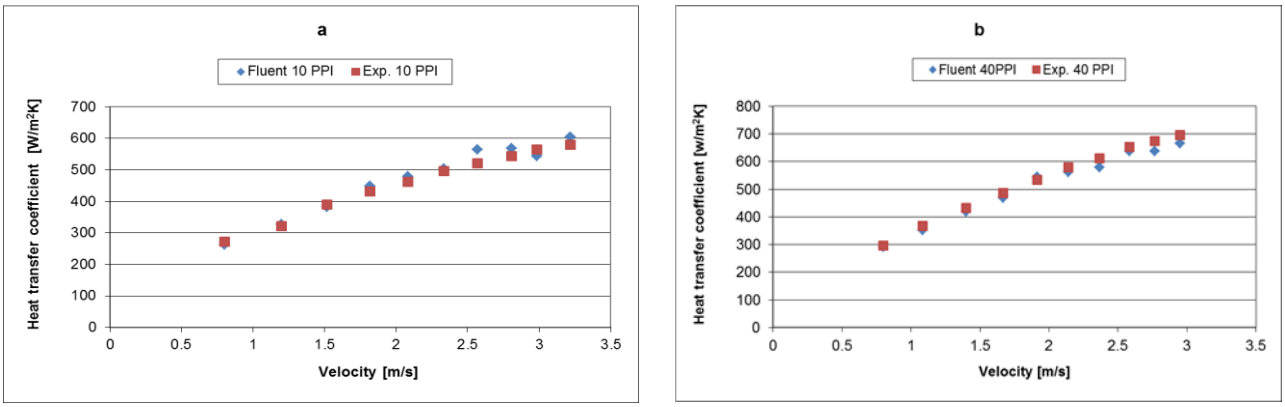


Fig. 12. Comparison between simulation and experimental heat transfer coefficient data at different velocities,for 10 and 40 porous densities and porosities ε=0.927(a) and ε=0.921(b) [11].

Comparing the values obtained by simulation with the experimental ones, relative errors resulted that were less than 5%, for most of the parameters studied, leading to the validation of simulation model developed in FLUENT by the experimental data.

11. Presentation and discussion of results

However, in our case with flow through a porous medium, researchers have shown that the localized heat transfer coefficient will vary with velocity, even at local Reynolds number less than 50.

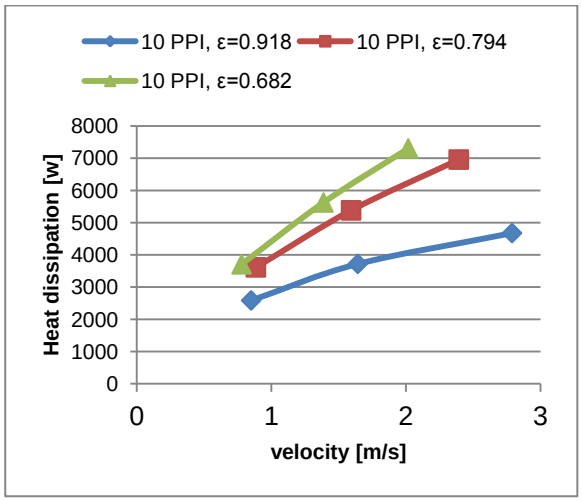


Fig. 13. Heat flux dissipation vs flow velocity [13]

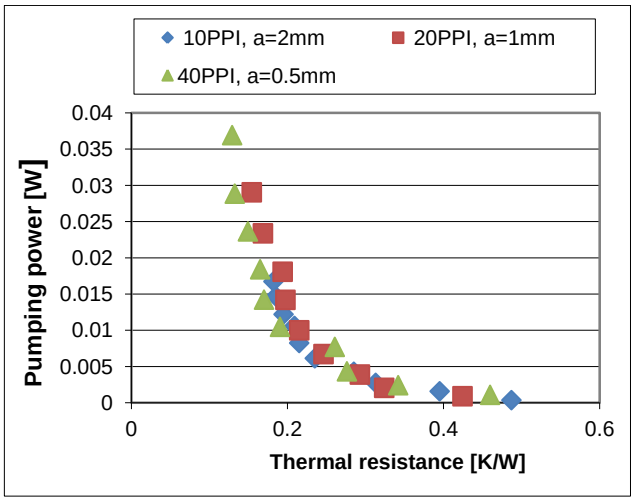


Fig. 14. Pumping power vs thermal resistance [13].

The heat transfer from the foam to the fluid will increase as either the porosity decreased (thus increasing surface area for heat transfer) or as the relative density (ρ) increases (thus increasing heat conduction through the ligaments) or as the velocity of the fluid increases (fig.13) [13].
However, all of these methods to increase heat transfer will result in increased pressure drop through the system. Thus, a delicate balancing act must be performed so as to maximize the ratio of heat output to pumping power[14,15]. In any heat exchanger design, the heat convection performance of the heat exchanger must be weighed against the energy required to operate the system, which is the pumping power in this configuration. In fig. 14 the optimal design is that which minimize the distance from the point to the origin of the plot. The simulated results of both pressure drop and heat transfer were compared with two of the best commercially available heat sinks with similar dimension to demonstrate both the advantages in thermal resistance and disadvantages of pressure drop increment with the use of highly porous media. In fig. 15 it could be seen that the metal foam heat sinks provide less thermal resistance than the best heat sink available from Thermaflo (Heat sink 1 and 2).

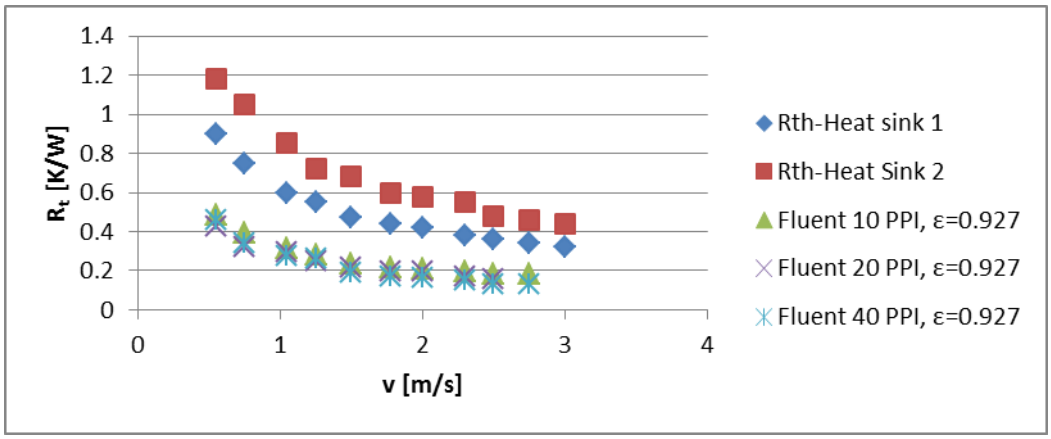


Fig. 15. Thermal resistance comparison plot.

Although there is a pressure drop increase for the use of porous materials as heat sinks. This pressure drop could be compensated by the increase in heat dissipation that the metal foams offer.

12. Conclusions

This model assumed heat transfer was dominated by the heat transfer of the fluid in the pores of the foam, rather than in the ligaments. In other words, since the foam is extremely conductive, the largest thermal resistance is at the local surfaces, rather than transmitting the heat through the foams. This model demonstrates several guiding principles of designing heat transfer devices utilizing cooling fluids and foams.

1. The highest conductivity ligaments are needed to transport the heat from the heat source rapidly into the foam so that it can then be transferred more uniformly to the cooling fluid.
2. While a turbulent fluid flow is desired to facilitate better mixing and high local heat transfer coefficients from the solid surfaces (ligaments) into the fluid, the effects of surface area are dominating as the surface area increases with the square of the change in pore diameter and the Reynolds number (turbulence) decreases monotonically with the change in diameter.
3. A low pressure-drop of the fluid through the foam is desired so that the required pumping power does not become overwhelming. Hence, low fluid viscosities are desired.
4. It is necessary to model the heat transfer of a system with a parametric study to optimize the various variables on the heat transfer. It was shown that there is a maximum pore diameter to optimize the ratio of dimensionless heat flux to pumping power. It was also shown that proper use of models could help optimize foam-based heat exchangers through parametric studies. For example, it was shown how the model could predict the effects of a geometric change in the system on reducing pressure while maintaining high heat transfer.

This type of model should be very useful in developing revolutionary new and exciting heat transfer devices. This work focused on developing an improved phenomenological thermo-fluid model in order to assess and optimize such porous heat transfer media with the intent of guiding the direction of future modelling, material and experimental.

References

[1] J. H. Rosenfeld, J. E. Toth, and A. L. Phillips, "Emerging applications for porous media heat exchangers", Proc. Int. Conf. on Porous Media and their Applications in Science, Kona, Hawaii, June 1996;
[2] E. Cruz, "Modeling of heat transfer in open cell metal foams', University of Puerto Rico, Mechanical Engineering Departament, 2004;
[3] P. Opritoiu, "Rans simulation of turbulent flow through porous media", Acta Tehnica Napocensis, nr.49., 2006;
[4] P. Opritoiu, "Fluid flow and pressure drop simulation in aluminium foam heat exchanger", Acta Tehnica Napocensis, nr.50, 2007;
[5] P. Opritoiu, "Modelling turbulence in open cell aluminum foam heat exchanger", Acta Tehnica Napocensis, nr.56, 2013;
[6] K. Tahir., "Predictions of some turbulent flows using upwind and hybrid discretisation schemes and two-equation turbulence model", Particulate phenomena and multiphase transport, vol. 5, Springer-Verlag, 1988;
[7] FLUENT v. 5/6, *Users manual,* Fluent inc.;
[8] Ajay Parihar, Shane Moeykens, *Flow through porous media*, Fluent inc., 2006;
[9] Michael Paul Scott, *Computational fluid dynamics study of flow through a shell and tube heat exchanger header with inlet modifiers*, Undergraduated thesis, University of Queensland, Australia, 2001;
[10] H. C. Brinkman, "On the permeability of media consisting of closely packed porous particles", Apl. Sci. Res., A1, pp.27-34, 1948;
[11] P. Opritoiu, "Validation of porous heat exchanger simulation model", Hidraulica, Magazine of Hydraulics, Pneumatics, Tribology, Ecology, Sensorics, Mechatronics, 2014;
[12] H. Alvarez, "Combined flow and heat transfer characterization of open cell aluminium foams', University of Puerto Rico, Mayaguez Campus, 2005;
[13] P. Opritoiu, "Heat transfer performance analysis in porous heat exchanger", Hidraulica, Magazine of Hydraulics, Pneumatics, Tribology, Ecology, Sensorics, Mechatronics, 2015;
[14] D. Opruţa, L. Vaida , A. Pleşa, "The simulation of fluid flow through a hydraulic resistance", Annals of DAAAM & Proceedings, 2008;
[15] D. Opruţa, L. Vaida, L. Naşcuţiu, A. Pleşa, "Coandă effect in oil flow through hydraulic command resistances", Conference: Annals of DAAAM and Proceedings of the International DAAAM Symposium, 2007.