CFD Simulation and Experimental Comparison of a Copper Wire Woven Finned Heat Exchanger to an Aluminium Flat Plate Finned Heat Exchanger

Dr John White

School of Mechanical Engineering, University of Birmingham UK.
Email:jxw998@bham.ac.uk

Received September 5th, 2013; Revised October 10th, 2013; Accepted October 17th, 2013

Copyright © 2013 John White. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

ABSTRACT

The performance of the adsorption cooling system is assessed in terms of the heat transfer coefficients of adsorbent bed heat exchanger. These parameters in-turn determine the specific cooling capacity (SCC) and the coefficient of the performance (COP) of the adsorption cooling system. It was observed that the overall heat transfer coefficients of the heat exchanger have a direct linear relation with the hot water inlet temperatures.

Keywords: Adsorption, CFD, Modelling, Silica gel, velocity

1. Introduction

An Adsorption cooling system is promising technology since it incorporates environmentally benign refrigerants and the industrial waste heat or low grade solar energy instead of mechanical power. However the low value of performance parameters (COP and SCP), high initial cost and the operating pressure, which is quite low these are the main issues that hold back the widespread application of the adsorption cooling systems. One of the efficient ways to improve the low SCP and COP of this system is the appropriate selection of the design and operating parameters of an adsorbent bed. The heat exchanger used as part of the adsorption system is one of the influential parameter in the heat transfer performance of an adsorption cooling system.

Studies relevant to this research span the last few decades. Chang et.al (2005) [1] experimental study that investigated the heat transfer efficiency of flat tube heat exchanger concluded that the flat tube heat exchanger thinner improves the mass transfer performance. InRiffel et.al (2007) [2] investigated the heat transfer effects of shell and tube heat exchanger for a solar adsorption cooling with an aim to improve the optimum performance of the system. Twenty years prior to this, Beecher and Fagan (1987) [3] designed and tested a heat exchanger for a surface temperature condition and in Ito et al.( 1977,) [4] applied the constant heat flux condition. They measured the heat transfer coefficient (h) since the fin efficiency (η) could be assumed as 100%.


The study of improving heat exchanger heat transfer efficiency has gained serious momentum over recent years due to increased demands by adsorption cooling industry for heat exchange equipment that is less expensive to build, and that operates better than standard heat exchange devices savings in heat exchanger materials. There is demand too, for adsorption cooling design with heat exchangers that are more compact and lightweight.

One way to increase the heat transfer of the heat exchanger and accomplish a lightweight design could be by changing the design configuration to increase the surface area, without increasing the size of the heat exchangers. There are three basic ways of accomplishing this [7,8]:

a. Increase the operational heat transfer surface area by using wire type copper finned compact heat exchanger. This change of design configuration and materials could considerably change
CFD Simulation and Experimental Comparison of a Copper Wire Woven Finned Heat Exchanger to an Aluminium Flat Plate Finned Heat Exchanger.

the heat transfer coefficient see table .1 for thermal conductivity of copper [7].

Increase heat transfer without significantly changing area. This can be accomplished by using a special channel shape, such as a wavy or corrugated channel, which would provide mixing due to secondary flows and boundary-layer separation within the channel. Vortex generators also increase heat transfer coefficient without a significant area increase by creating longitudinally spiralling vortices exchange fluid between the wall and core regions of the flow, resulting in increased heat transfer [1-7].

Increase both heat transfer coefficient and surface area of a heat exchanger by. Interrupted fins (i.e., offset strip and louvered fins) act in this way. These surfaces increase the effective surface area, and enhance heat transfer through repeated growth and destruction of the boundary layers [1, 2-7].

The use of any one of these heat transfer method mentioned could substantially increase the heat transfer fin efficiency and the motivation behind the design of new a adsorbent bed compact copper wire woven heat exchanger.

2.1 The heat exchange fin performance

Heat exchanger fin performance can be explained by three different methods. The first is fin effectiveness this is the ratio of the heat exchanger fin heat transfer to the heat transfer of the tube without fin [8]. The second is the fin efficiency and is defined as the ratio of the heat transferred through the actual fin to that transferred through a perfect fin.

A perfect fin is thought to be one made of a perfect or infinite conductor material. A perfect conductor has an infinite thermal conductivity, so in theory, the entire fin is at the base material heat transfer temperature [5, 8].

The equation for this is

\[ \varepsilon_f = \frac{q_f}{h A_c \kappa \theta_b} \]

Where is the heat exchanger fin cross-sectional area at the root. Fin performance can also be considered by heat exchanger fin efficiency this is the ratio of the heat exchanger fin heat transfer to the heat transfer of the fin if the entire fin were at the root heat [5-8].

\[ \eta_f = \frac{q_f}{h A_f \theta_b} \]  

The fin efficiency as

\[ \eta_f = \frac{q_f}{h A_f \theta_b} \]

\[ A_f \] This equation is equivalent to the surface area of the heat exchanger fin. The third method of heat exchanger fin performance can be explained as its entire surface efficiency.

\[ \varepsilon_f = \frac{q_f}{h A_f \theta_b} \]

The fin efficiency as

\[ \varepsilon_f = \frac{q_f}{h A_f \theta_b} \]

2.2 Common flat plat fin heat exchanger used in a adsorbent bed

![Figure 1: Common type of flat fin heat exchanger used in adsorption cooling system](image-url)
2.3 Different type of heat exchanger fin configuration that could be used in adsorbent beds

![Image of finned tubes](image1)


The figure 2 has eight different types of heat exchanger that could be used in an adsorbent bed to improve heat transfer and reduced the size of the heat exchangers (1) and (2) are typical circular fins. (3) and (4) serrated circular fins and dimpled spirally wound circular fins, both are design to improve convection (5) wire wound copper coil fin design to improve convection and increase surface area. (6) bristle fins, (7) spirally wound and (8) machined from base metal are also design to improve convection and increasing surface area.

2.4 Comparison of aluminum flat fins heat exchangers to copper wire woven fins heat exchanger

This section discusses a CFD simulation comparison study on a wire woven copper coil fin and a flat plat fin comparing the heat transfer performance. The CFD boundary conditions and properties taken from test rig experimental data research papers and Lienhard IV et al 2005 [1-7].

An important design aspect of adsorbent bed heat exchanger technology is deciding what materials are appropriate for the design of adsorbent bed heat exchanger. The common choice is aluminium alloy 1050A or 6063A flat fins. These types of aluminium have one of the highest thermal conductivity values, see table 6.1. Thus is a good material for removing heat from a system to an external environment. In the case of an adsorbent bed, the heat exchanger is packed with porous absorbent material and so the extended fins are used to increase and decrease the heat transfer rate from the fluid inside the heat exchanger to the surface of the porous absorbent.

The porous absorbent affects thermal conductivity of flat fins heat exchange. To overcome this problem, the heat exchanger designers design the heat exchanger fins larger to improve the heat transfer. This is one of the causes for the bulky size adsorption cooling system. One way designers could solve this problem is by changing the material and design configuration of the heat exchanger of the adsorbent bed.

In this chapter a copper wire woven fins heat exchanger is used for the adsorbent bed. Copper has around twice the thermal conductivity of aluminium see table for the properties of copper heat exchangers materials [1, 4-7].

<table>
<thead>
<tr>
<th>Material</th>
<th>At 25°C</th>
<th>Thermal conductivity (W/m·°C)</th>
<th>At 125°C</th>
<th>At 250°C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Iron</td>
<td>80</td>
<td>80</td>
<td>60</td>
<td></td>
</tr>
<tr>
<td>Low carbon steel</td>
<td>54</td>
<td>51</td>
<td>47</td>
<td></td>
</tr>
<tr>
<td>Stainless steel</td>
<td>16</td>
<td>17.5</td>
<td>18</td>
<td></td>
</tr>
<tr>
<td>Tungsten</td>
<td>180</td>
<td>180</td>
<td>150</td>
<td></td>
</tr>
<tr>
<td>Platinum</td>
<td>70</td>
<td>71</td>
<td>72</td>
<td></td>
</tr>
<tr>
<td>Aluminium</td>
<td>250</td>
<td>250</td>
<td>250</td>
<td></td>
</tr>
<tr>
<td>Gold</td>
<td>310</td>
<td>312</td>
<td>310</td>
<td></td>
</tr>
<tr>
<td>Silver</td>
<td>420</td>
<td>418</td>
<td>415</td>
<td></td>
</tr>
<tr>
<td>Copper</td>
<td>401</td>
<td>400</td>
<td>398</td>
<td></td>
</tr>
</tbody>
</table>

Copper wire fins heat exchanger is a small diameter wire fins. It has better rates of heat transfer than aluminium flat plate fins heat exchanger. The performance advantages of copper over aluminium as a heat transfer material include greater heat exchange, better long-term durability and resistance to corrosion.

Wire wound fins heat exchangers are made from one continuous piece of wire wound helically and root soldering onto a copper tube. The root soldering eliminates air gaps between the fins and the copper tube [7-11].

3. Discussion

3.1 Approaches for Heat Transfer CFD Modelling

In this study, it was assumed that the temperature and pressure may have some variation throughout the whole adsorbent bed and the system may have some heat losses to the environment. The following assumptions are made before establishing the CFD model:
CFD Simulation and Experimental Comparison of a Copper Wire Woven Finned Heat Exchanger to an Aluminium Flat Plate Finned Heat Exchanger.

- The temperature and pressure may have some variation in the silica gel adsorption bed.
- The water is adsorbed uniformly by the silica gel.
- The vapour inside the adsorption bed is saturated vapour.
- The system has no heat losses to the environment

In the (CFD) Computational fluid dynamics simulation the heat source temperature variation was taken from 30°C to 85ºC. A CFD simulation computer program was constructed to analyse the influence of hot and cooling water temperature on the adsorption/desorption performance of silica gel.

3.2 How the CFD Simulation was completed

In this phase of the model design the major entities are defined. The outer wall to the model is established, as well as, the internal structure, solid particles and fluid regions. Several entities, such as heat exchanger and silica gel are added to the geometry. Before exporting the fundamental geometry it is important all curves are defined as so-called B-Spline curves. These are mathematical descriptions of the specific curves. This conversion is necessary to be able to have the surface mesh form to the exact contours of the created geometry.

3.3 Boundary Conditions

The boundary conditions determine the flow and thermal variables on the boundaries of the physical model. There are a number of classifications of boundary conditions:

- Flow inlet and exit boundaries: pressure inlet, velocity inlet, pressure outlet.
- Heat exchanger fins wall, repeating, and limit boundaries: fins wall, symmetry.
- Internal fluid, solid
- Internal face boundaries: porous, fins wall, interior.

In the model, velocity was assigned to the flow inlet of the adsorbent bed; this boundary condition defines a flow velocity at the inlet of the bed. The flow exit boundary is defined as a pressure outlet and the outlet pressure is defined as atmospheric pressure. The bed and packing interior are defined as boundaries. The fins wall boundaries separate the fluid zone and vapour in between the silica gel granules from the fins wall zones [9, 18].

With the determination of the boundary conditions the physical model can be defined by a numerical solution. It was then necessary to determine how the solution will be established. This was be done by setting the iteration parameters. With all boundary conditions defined, a number of additional parameters and solving schemes where selected. An initial condition was assigned to the model and was used to help speed the convergence of the computation.

The computation is an iterative process that solves the governing equations for flow and energy in each simulated cell. Depending on the complexity of the model and the computer resources available, CFD simulation can take anywhere from minutes to days [6]. The results of the simulation can be viewed and manipulated with post-processing software once the simulation has converted to a solution.

3.4 Mesh

One of the most important parts of CFD modelling is the construction of the mesh topology. It has to be chosen with enough detail to describe the processes accurately and with a degree of smoothness that enables solution within a satisfactory period of time. When an optimal density has been found, refining this will increase the model size without displaying more flow detail [9, 18]. When it is coarsened the mesh will obscure, possibly essential, parts of the flow detail. The mesh determines a large part of creating an acceptable simulation.

3.5 Mesh Variations

This was created to focus on silica gel to silica gel contact points and silica gel-wall contact points. The study of the silica gel geometry in Flow simulation was also made to compare the results from CFD codes. When actual contact points are created, both surfaces that are contacting have one common node [9, 18]. In surface mesh creation this can be defined and does not pose any problems.

The 3D mesh can be created relatively easily by merging a number of nodes on the contacting surfaces. When, however, a solution is iterated convergence problems occur with the fluid elements around the contact point. In a laminar case, the solution parameters can be adjusted to get a converging solution, in a turbulent case this becomes impossible [9].
After it was found that actual contact points, as they were designed, with the Silica gel geometry, a turbulent solution of the model used for the CFD validation could not be established, a number of models were created to make a comparison between several sizes of gaps between the silica gel in the packing. Eventually, the appropriate gaps, facilitating both a turbulent flow solution, as well as, a sufficient stagnant fluid film around the silica gel for heat transfer to be simulated was created.

4. CFD Porous Medium Methodology

When using CFD it is possible to apply the porous medium simulation method to predict the effect heat transfer has on porous adsorbents. For the porous medium simulation method the CFD model has a mass of cells representing the fluid inlet [7]. This is followed by the porous adsorbent units which are used to model fluid flow through porous adsorbent [12, 13]. Full flow field predictions are possible with the porous adsorbent simulation method because the resistance of the porous adsorbent to flow is described by the equations:

\[
\frac{\Delta P}{L} = -\alpha U_s^2 - \beta U_s,
\]  

where the coefficient values and are assigned temperature dependent values that describe the performance of a porous adsorbent. High values of and preclude flow at right angles to the porous adsorbent. Upstream and downstream of the water vapour flow field is solved using the usual Reynolds averaged Navier–Stokes methodology.

4.1 Fluid Flow Fundamentals

For iteration CFD solvers use generalised fluid flow and energy balances based on the Navier Stokes equations. The balances are generalised so the user can influence which elements are added in the balance and which are not. [9, 18] The number of balances to be solved is also user defined; it can be advantageous to not solve all balances initially. The generalised balances that are used by the Flow simulation commercial CFD package are the Navier Stokes equations for conservation of mass and momentum, when it is set to calculate laminar flow without heat transfer. Additional equations are solved for heat transfer [8].

4.2 Solving the CFD Problem

When a mesh is completed with its density and all other complications resolved, the actual computational part of CFD can be started. At this point the completed geometry can be imported in the solver and the CFD simulation is started. Again, a series of steps are to be performed; first, the boundary conditions on the system need to be set, next the process iteration parameters need to be set. With the boundary conditions defined the simulation is performed. The final step in obtaining the desired data is the post-processing of the data in which the desired data sets are taken from the simulation data [7, 10].

4.3 Post-Processing the Simulation Data

When the simulation has converged the last data set is stored as a final solution. This data set has a record of the status of all elements in the model, temperature, densities, pressures, flow aspects, etc. To be able to interpret the data it needs to be ordered and reduced to comprehensible sizes [5,7-9]. This displaying of the data is called post-processing and makes it possible to compare the different simulations with each other and with external data. There was as many ways of displaying the data as there were data points so it was important to select the data representation that was required for the desired data comparison.

Some of the standard options available are contour plots and velocity vector plots. Contour plots will give a plot in the defined data point collection; this can be a plane or a volume, of contours of another variable. For example, a plane can be defined as a constant x coordinate plane (y-z plane), a contour plot can be made showing temperature contours in this plane.

In the same plane a velocity contour plot can be made showing absolute velocities of the fluid in the defined plane [7, 9].
5. BET Adsorption Equations

The concept of the BET adsorption equation is an expansion of the Langmuir theory, which is an equation for monolayer molecular adsorption to multilayer adsorption with the following assumption:

\[
\frac{1}{v [(\frac{P}{P_0}) - 1]} = \frac{c-1}{v_m c} \left( \frac{P}{P_0} \right) + \frac{1}{v_m c}
\]  

(5)

\( P \) and \( P_0 \) are the balance and the diffusion force of adsorbents temperature of the adsorption \( v \) is the adsorbed \( [10] \). Vapour capacity and \( v_m \) is the monolayer adsorbed vapour amount.

\( c \) is expressed by (6.3):

\[
c = \exp \left( \frac{E_1 - E_L}{RT} \right)
\]  

(6)

\( E_1 \) is the adsorption for the first layer, and \( E_L \) is that for the second layers \([10]\).

6. Heat Transfer in the Two Different Types of Heat Exchangers

The wire woven fin and flat fin heat exchanger have silica gel embedded between the fin spaces. Figure 4, shows the different heat transfer phenomena taking place. The phenomena of heat transfer in an adsorption bed are examined based on the following assumptions:

- Conduction inside the silica gel,
- Convective heat transfer,
- Silica gel, Fin Conduction and radiation,
- Pressure drop.

Figure 4 Different heat transfer taken into account in the packed bed modelling approach.

6.1 The thermal heat transfer of packed beds

Modelling the thermal heat transfer in a porous adsorbent bed includes two main studies (i) conduction between porous adsorbents and (ii) heat transfer through adsorbed water vapour and between porous adsorbent. The geometry of a common joint is shown in Figure 4 where spherical porous adsorbent are placed in contact. The gap between the contacting bodies is filled with a water vapour heat is transferred from one sphere adsorbent to another.

The heat transfer through a porous adsorbent bed can be affected by various variables. The type of porous adsorbent, the packing size, the packing ratio, the inlet water and vapour temperature, the velocity of the vapour fluid, also the type of refrigerant fluid used in the evaporator. These various variables will have an significant effect on the heat transfer phenomenon occurring in the porous adsorbent bed. [1-3].

Another area of particular difficulty in the modelling of heat transfer in a porous adsorbent bed is the heat transfer near the heat exchanger fins wall. Experimental data used in CFD simulation modelling are taken from thermocouples fixed to the surface of the heat exchanger fins. However, due to physical limitations the thermocouples near to the fins it can sometime be difficult to obtain accurate experimental recordings because of the adsorbent packing near the fins wall.
6.2 Near-Heat Exchanger Fins Wall Small Section Geometries

As mentioned before, computational fluid dynamics modelling is constrained by the available computational power and the required accuracy. In all modelling cases an appropriate balance between the two has to be found. For more accurate modelling a more detailed computational model has to be used, which increases the strain on the simulations. To be able to get more detailed views of certain areas in the simulation model, or to be able to simulate a specific section quicker, geometry can be created of a segment of the overall simulation model.

Since the major concern in this study is to find the performance of heat transfer from the heat exchanger fins to the porous adsorbent granules for two different types of heat exchangers, i.e., copper wire woven fins and aluminium flat plate, segment geometry can be used to generate simulation results more quickly, due to its limited size. One segment of fins with porous adsorbent spheres was simulated in order to predict the heat transfer from fin to porous adsorbents granules. Three dimensional model of a fins and silica gel adsorbent model can predict the distribution of the thermal heat transfer between fins and porous silica gel during heating desorption heat phases.

7 Heat Transfer Equations

The heat transfer equation is given as

$$\rho c_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda \frac{\partial T}{\partial z} \right) + s_v$$

The initial condition for Eq. (4) is

$$t = 0, T = T_0$$

and the boundary conditions are as follows: on the x, y and z symmetries,

$$-\lambda \frac{\partial T}{\partial n} = 0$$

on the surfaces of silica gel

$$-\lambda \frac{\partial T}{\partial n} = q_{evap, sur} + q_r$$

where $q_r$ is heat loss by radiation on the surface of the fins of the heat exchanger which is calculated based on the Stefan–Boltzmann law:

$$q_r = \frac{Q_r}{A} = \sigma \varepsilon_r (T^4_{sur} - T^4_w)$$

The water vapour rate $s_v$, is,

$$S_v = h_{fg} f_v$$

where $f_v$ is the water vapour generation rate for the control volume.

The latent heat of vaporisation of water $h_{fg}$ which can be calculated by the following formula [7 , 14].

$$h_{fg} = 2500.8 - (T - 273.15) \times 2.422449$$

The Fick’s second law of diffusion can be used to describe mass transfer phenomenon.

$$\frac{\partial c}{\partial t} = \frac{\partial}{\partial n} \left( D \frac{\partial c}{\partial n} \right)$$

For mass transfer through the silica gel porous medium, Eq. (11) can be rewritten as

$$\frac{\partial c}{\partial t} = \frac{\partial}{\partial n} \left( D_{eff} \frac{\partial c}{\partial n} \right) = \frac{c}{\tau} \frac{\partial}{\partial n} \left( D \frac{\partial c}{\partial n} \right)$$

Since concentration is proportional to the partial pressure, when considering water vapour through a porous medium, concentration can be normally expressed as the equilibrium relationship in terms of partial pressure [7-12]. Noting that for an ideal gas the concentration units can be converted to partial pressure units as follows:

$$C = \frac{p}{R_0 T}$$

Then the inner mass transfer rate is
The mass transfer happens in the area within the adsorbent bed where water vapour is actually being adsorbed on the silica gel. The mass transfer moves from the input end toward the output end of the adsorbent bed during operation. That is, as the silica gel near the input becomes saturated with water vapour, the vapour moves toward the output end of the bed where the silica gel is not yet saturated.

\[ P = P_{\text{sat,0}}, \quad T = T_0 \]  

(18)

And the boundary conditions are as follows: on the x, y and z symmetries,

\[ \frac{\partial p}{\partial n} = 0 \]  

(19)

on the surfaces,

\[ P = P_{VC} \]  

(20)

The vapour transport within the silica gel surface can be treated as vapour movement through the porous medium [4], which can be expressed as a transfer of vapour in the pores with inner vapour generation rate.

\[ f_V = \frac{-\varepsilon}{\tau} \]  

\[ \chi \left( \frac{\partial}{\partial x} \left( K_{xV} \frac{\partial p}{\partial x} \right) + \frac{\partial}{\partial y} \left( K_{yV} \frac{\partial p}{\partial y} \right) + \frac{\partial}{\partial z} \left( K_{zV} \frac{\partial p}{\partial z} \right) \right) \]  

(21)

where \( k_v \) is the mass transfer coefficient of vapour in the vapour-filled pores, which is given by

\[ k_V = \rho_v \frac{d_a^2}{32u} \]  

(22)

### 8. Results

#### 8.1 Comparative study

In heat exchangers there may be different thermal heat transfer owing to different type of heat exchanger design, creating different behaviour in the heat exchanger. Therefore, CFD thermal heat transfer simulation analysis will help the design engineer to predict the behaviour of the different types of heat exchanger performance.

The thermal heat transfer CFD simulation performance results from a 3D CFD simulation model of a wire woven finned heat exchanger and 3D CFD simulation model of flat fin heat exchanger shown in figure 5 and 6 were compared. The heat exchanger with the capability of transferring more heat to its tip will have greater heat transfer efficiency.

#### 8.2 Experimental Setup and Procedure

The experiment apparatus used to measure the different types of heat exchanger fin efficiency consists of adsorbent bed hot water tank and a water pump as shown in figure 5. This experimental apparatus was design to perform heat transfer testing of heat exchangers. The adsorbent bed section is removable this is to allow changing of the different heat exchanger.

Figure 5 Heat exchanger test rig.

There is one adsorbent bed in the test apparatus and the function of the bed is to house the heat exchanger. In the apparatus of the experiment the heat exchanger is attached to the hot water tank so as to allow the hot water to be pump into the bed. The heat is conducted along each fin and is transferred by natural convection to the adsorbent packing.

Thermocouples are embedded at intervals along each fin so that temperature is known at selected points as shown in figure 5. It is at these points where the convection coefficient will be determined. Temperature readings were monitored at 60 seconds intervals to determine the efficiency of the different heat exchanger fin as shown in table 2.
Figure 6 Setting up wire fin for heat transfer experiment.

The heat transfer coefficient and the effectiveness of the flat plate finned heat exchanger and wire woven heat exchanger are found based on different correlations from the experimental test rig results figure 7 show the flat finned heat exchanger.

Figure 7 Setting up wire fin for heat transfer experiment.

Table 2 shows the heat exchanger heat transfer efficiency test at temperature of 85°C.

<table>
<thead>
<tr>
<th>Test (Sec)</th>
<th>Wire Woven fin T-hot in</th>
<th>Wire Woven fin T-hot out</th>
<th>Flat fin T-hot in</th>
<th>Flat fin T-hot out</th>
</tr>
</thead>
<tbody>
<tr>
<td>60</td>
<td>85</td>
<td>83.9</td>
<td>85</td>
<td>73.5</td>
</tr>
<tr>
<td>120</td>
<td>85</td>
<td>84.1</td>
<td>85</td>
<td>73.5</td>
</tr>
<tr>
<td>180</td>
<td>85</td>
<td>83.9</td>
<td>85</td>
<td>73.6</td>
</tr>
<tr>
<td>240</td>
<td>85</td>
<td>83.7</td>
<td>85</td>
<td>73.6</td>
</tr>
<tr>
<td>300</td>
<td>85</td>
<td>83.7</td>
<td>85</td>
<td>73.2</td>
</tr>
<tr>
<td>360</td>
<td>85</td>
<td>83.5</td>
<td>85</td>
<td>73.2</td>
</tr>
<tr>
<td>420</td>
<td>85</td>
<td>83.4</td>
<td>85</td>
<td>73.1</td>
</tr>
<tr>
<td>480</td>
<td>85</td>
<td>83.4</td>
<td>85</td>
<td>73.1</td>
</tr>
<tr>
<td>540</td>
<td>85</td>
<td>83.2</td>
<td>85</td>
<td>73.1</td>
</tr>
<tr>
<td>600</td>
<td>85</td>
<td>81.1</td>
<td>85</td>
<td>73.1</td>
</tr>
<tr>
<td>660</td>
<td>85</td>
<td>80.2</td>
<td>85</td>
<td>73.1</td>
</tr>
<tr>
<td>720</td>
<td>85</td>
<td>80.1</td>
<td>85</td>
<td>72.9</td>
</tr>
</tbody>
</table>

The highlighted column indicates the values used for CFD simulation of the hot water cycle.

9 Computational Fluid Dynamics Simulation of the two Different Heat Exchangers

9.1 The Physical Models

The flow and thermal variables on the boundaries of the physical model there are a number of classifications of boundary conditions:

1. Flow inlet and exit boundaries: Temperature inlet, velocity inlet, temperature outlet.
3. Internal fluid, solid
4. Internal face boundaries: porous, fins wall, interior

In the model, velocity was assigned to the flow inlet of the adsorbent bed; this boundary condition defines a flow velocity at the inlet of the bed. The flow exit boundary is defined as a pressure outlet and the outlet pressure is defined as atmospheric pressure. The bed and packing interior are defined as boundaries. The fins wall boundaries separate the fluid zone and vapour in between the silica gel granules from the fins wall zones [9]. With the determination of the boundary conditions the physical model can be defined by a numerical solution. It was then necessary to determine how the solution will be established. This was be done by setting the iteration
parameters. With all boundary conditions defined, a number of additional parameters and solving schemes where selected. An initial condition was assigned to the model and was used to help speed the convergence of the computation.

The computation is an iterative process that solves the governing equations for flow and energy in each simulated cell. Depending on the complexity of the model and the computer resources available, CFD simulation can take anywhere from minutes to days [10]. The results of the simulation can be viewed and manipulated with post-processing software once the simulation has converted to a solution.

One of the most important parts of CFD modelling is the construction of the mesh topology. It has to be chosen with enough detail to describe the processes accurately and with a degree of smoothness that enables solution within a satisfactory period of time. When an optimal density has been found, refining this will increase the model size without displaying more flow detail [11-14]. When it is coarsened the mesh will obscure, possibly essential, parts of the flow detail. The mesh determines a large part of creating an acceptable simulation see figures.8 and figures 9 illustrates the stages followed to achieve the results that was discussed in this section.

9.2 Creating the Silica Gel Porous Medium

To create a silica gel porous medium for the adsorption bed the need is to, first specify the porous medium’s properties (porosity, permeability type, etc.) in the Engineering Database and then apply the porous medium to the spheres in the packed bed assembly. The data shown in figure 9 were those specified in this simulation.

\[ \frac{\Delta P}{L} = -\alpha U_s^2 - \beta U_s, \]

(23)

where the coefficient values $\alpha$ and $\beta$ are assigned temperature dependent values that describe the performance of a porous adsorbent. High values of $\alpha$ and $\beta$ preclude flow at right angles to the porous adsorbent. Upstream and downstream of the water vapour flow field is solved using the usual Reynolds averaged Navier–Stokes methodology.

9.3 Thermophysical Properties

Once the model was modelled, boundary conditions were assigned to each section of the model and were used to simulate the actual conditions set by the adsorption cooling system. Examples of common boundary
conditions include velocity inlet, pressure inlet, pressure outlet, temperature profile.

Table 3 Thermophysical properties of copper tube

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density, ρ (kg/m³)</td>
<td>1000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Specific heat capacity, C_p (J/kg K)</td>
<td>4200</td>
<td></td>
</tr>
<tr>
<td>Thermal conductivity, k (W/mK)</td>
<td>0.61</td>
<td></td>
</tr>
<tr>
<td>Dynamic viscosity, μ (kg/ms/Î•)</td>
<td>0.96172</td>
<td></td>
</tr>
</tbody>
</table>

Table 4 Thermo-physical properties of silica gel.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific surface area (m²/g)</td>
<td>650</td>
<td></td>
</tr>
<tr>
<td>Porous volume (m³/g)</td>
<td>0.36</td>
<td></td>
</tr>
<tr>
<td>Average pore diameter (Å)</td>
<td>22</td>
<td></td>
</tr>
<tr>
<td>Apparent density (kg/m³)</td>
<td>730</td>
<td></td>
</tr>
<tr>
<td>pH value</td>
<td>5.0</td>
<td></td>
</tr>
<tr>
<td>Water content (wt.%)</td>
<td>&lt;2.0</td>
<td></td>
</tr>
<tr>
<td>Specific heat capacity (kJ/kg K)</td>
<td>0.921</td>
<td></td>
</tr>
<tr>
<td>Thermal conductivity (W/m K)</td>
<td>0.174</td>
<td></td>
</tr>
<tr>
<td>Mesh size</td>
<td>10-40</td>
<td></td>
</tr>
</tbody>
</table>

Table 5 Thermo-physical properties of alumina fins

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic Modulus</td>
<td>1.1e11 N/m²</td>
<td></td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>0.37</td>
<td>NA</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>4e+10</td>
<td>N/m²</td>
</tr>
<tr>
<td>Density</td>
<td>8500</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Tensile Strength in X</td>
<td>3984380000 N/m²</td>
<td></td>
</tr>
<tr>
<td>Compressive Strength in X</td>
<td>3650</td>
<td>N/m²</td>
</tr>
<tr>
<td>Thermal Expansion Coefficient</td>
<td>2.4e+005</td>
<td>K/K</td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>380</td>
<td>W/(m K)</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>380</td>
<td>J/(kg K)</td>
</tr>
<tr>
<td>Material Density</td>
<td>1.6</td>
<td></td>
</tr>
</tbody>
</table>

10 Surface area and Volume of Flat Fin Results Generated by CFD

For most finned heat exchangers heat transfers are determined by the materials thermal conductivity and the structures surface area to volume ratio. As mentioned, before increasing the surface of a heat exchanger wills also increases the heat transfer of the heat exchangers. To study the surface area to volume ratio and heat transfer of heat exchangers, two different types of heat exchangers were selected and compared.

The length and the height of the aluminium flat fin are 32mm length and 32mm height as shown in figure 10.

Figure 10 Drawings dimension for aluminium flat finned heat exchanger used to generate volume and surface area of fin.

Figure 11 Single flat fin volume and surface area generated by assigning mass properties to 3D CAD model.

A mesh was created to focus on silica gel to silica gel contact points and silica gel heat exchanger fin wall contact points. The study of the silica gel geometry in Flow simulation was also made to compare the results from CFD codes. When actual contact points are created,
both surfaces that are contacting have one common node [15].

In surface mesh creation this can be defined and does not pose any problems. The 3D mesh was created relatively easily by merging a number of nodes on the contacting surfaces. When, however, a solution is iterated convergence problems occur with the fluid elements around the contact point. In a laminar case, the solution parameters were adjusted to get a converging solution [7,15-17].

Figure 12 Aluminium flat plat finned heat exchanger figure (c) and (d) presents the flat fin mesh.

The fin root temperature was set from 85°C and the model was run until the solution approached the steady state behaviour after 750 seconds. The fin root temperature changed from 85°C to 73°C as heat was conducted to the top edge of the fin.

This shows a reduction in temperature of the aluminium flat plat fin was by 12°C. This reduction in temperature is partly due to the heat transfer from the aluminium fin to porous adsorbent see figure 13 and 14.

Figure 13 The contour thermal heat transfer temperature of the flat finned heat exchanger the centre of the fin is 85°C and fin edge is at 73°C.

Figure 14 Changes of fin heat transfer efficiency over time due to the heat transfer of the heat exchanger the temperature decreases as it flows through the flat plate fin.

Temperature distribution figure 14 presents the temperature supplied along the length of the rectangular fin. A higher incline can be detected near the base of the fin due to the concentrated temperature difference between fin surface and the surrounding porous adsorbents covering the fins.

10.1 Surface area and Volume of Wire Fin Results Generated by CFD

Figure 15 Drawing dimension for copper wire woven finned heat exchanger used to generate volume and surface area of fin.
10.2 Wire finned heat exchanger surface area and volume

The surface area of one copper wire woven fin coil is 3079.53 square millimetres. This comprises 0.7mm copper wire of 1400mm length woven coiled into 70 loops as shown figure 15 and 16.

10.3 The Wire Woven Fin Adsorbent Bed

The method used to carry out the simulation of this type of adsorbent bed is the same as that described above. To reduce the time for CFD simulation a small geometries of the copper fin with adsorbent packed between one pitch was model for the CFD simulation.

11 One Pitch of Woven Wire Finned Heat Exchanger Mesh

The mesh for the one pitch of woven wire finned heat exchanger contains 187365 nodes and 84383 elements, see figure 18. The CFD programme creates the mesh based on the input parameters enter into the user define function menu. As can be seen in figure 18 the tetrahedral mesh is unstructured.

Figure 18 Nodes 187365 and elements 86483.

The first mesh to be simulated was a coarse mesh saving computations time on the CFD simulation. As the simulation was to be limited to a one pitch heat exchanger finned model it was determined that a fine mesh could be used as shown in figure 19.

Figure 19 CFD Representations showing a mesh detail of the wire woven fin pack with adsorbent.

The CFD mesh quality can be identified through two main factors. The first factor is that the CFD simulation meshes must have a sufficient number of points in the internal of the computational domain to describe the physical domain correctly. The second factor requirement for CFD mesh quality is that a sufficient number of points must be specified on the boundary to represent it accurately. This requires the number of boundary points to adapt according to the model surface geometry [19].
CFD Simulation and Experimental Comparison of a Copper Wire Woven Finned Heat Exchanger to an Aluminium Flat Plate Finned Heat Exchanger.

11.1 One pitch of wire woven finned heat exchanger with adsorbent packing

Figure 20 Displays the heat transfer efficiency of the wire fin heat exchanger root contour temperature is at 85°C and the fin tip temperature is at 81°C.

Figure 20 and 21 presents the temperature supplied along the length of the wire woven fin. A higher incline can be detected near the base of the fin due to the concentrated temperature difference between fin surface and the surrounding porous adsorbents covering the fins.

![Temperature Contours](image)

**Figure 21** Changes of fin heat transfer efficiency over time due to the heat transfer of the heat exchanger the temperature decreases as it flows through the wire woven fin.

The CFD simulation method for the wire woven fin was as described for the flat fin heat exchanger. The fin root temperature of the wire fin was set from 85°C and the model was run until the solution approached the steady state behaviour (after 750 seconds), as shown in figure 21. The fin root temperature decreased to 81°C from 85°C as the heat transferred to the top edge of the wire fin. This shows the change in temperature of the wire woven fin was by 4°C. This low reduction in temperature is to some extent to do with the heat exchanger been design from a continuous wire copper coil, which contributes to the good heat transfer, also, being made of copper and not aluminium increase the heat transfer to the adsorbent. The thermal conductivity of copper compared to aluminium is shown in table 1.

### 12 Model Validations

Several checks were performed in order to verify the generated results the adsorbent bed heat transfer performance was observed to ensure that the results satisfied the boundary conditions. The resulting file generated by Cosmos flow upon the completion of each run was carefully examined and analysed. The surface area to volume ratio of the heat exchangers was verified by comparing simulation result to surface area to volume simulation. The heat transfer results were compared to data from experiment.

### 13 Conclusions

CFD as a design tool for adsorbent bed design proves to be useful in estimating wall to fluid heat transfer parameters. It makes possible the modelling of a realistic case of an adsorbent bed using wire woven fin and flat fin heat exchangers. From the simulation comparisons of the copper wire woven finned and aluminium flat finned heat exchangers the wire finned heat exchanger has three advantages over the aluminium flat fin.

The first advantage was the greater surface area as seen in figure 16 the second was a better heat transfer efficiency as seen in figure 21 and the third advantage was the smaller volume of the heat exchanger as showed in figure 16.

It was observed from CFD simulation carried out on heat transfer on contact points between fin and porous adsorbents that the contact points between the adsorbent and heat exchanger had a significant effect to heat transfer. A poor heat transfer flow distribution can result in a lower heat transfer rate. Therefore, the optimisation of flow distribution is an essential step in heat exchanger design optimisation. CFD methods have demonstrated to have great potential in predicting the performance of both existing and newly developed adsorbent beds designs.

### REFERENCES


Chiller.,  *Heat SET, Heat Transfer in Component and Systems for Sustainable Energy Technologies*, 18-20 April, Chambéry, France


