IOP Conf. Series: Materials Science and Engineering **227** (2017) 012103 doi:10.1088/1757-899X/227/1/012103

Study of a sea water heat exchanger using Computing Fluid Dynamics

A Preda[1], L L Popescu[2] and R S Popescu[2]
[1]Constanta Maritime University, Faculty of Naval Electro-Mechanics, 104 Mircea cel Batran Street, 900663, Constanta, Romania
[2]Technical University of Civil Engineering of Bucharest, 66 Pache Protopopescu Blv. Bucharest, Romania

E-mail: andrei.preda@yahoo.com

Abstract. In order to have a better visualization of the heat transfer that might occur in a tubular seawater heat exchanger, I have created a simulation using Ansysy Fluent program. The first stage of the simulation is the representation of the subject in the Geometry. The next step is the mesh in a finite number of cells. All settings and information defined in the first two phases are taken in the third (Setup), where are imposed boundary conditions (the input and output). The next step is the calculation (Solutions) where the problem is solved iteratively using one of the methods of calculation selected. The software can solve numerical equations of Navier-Stokes flow, continuity equation, energy conservation etc. that are available in the database of the program. The last part is especially designed for the analysis and interpretation of results (distribution of speeds, temperatures, pressures etc.). Due to the fact that many types of heat pump used as working fluid water, a mixture of water and glycol or Freon, I have required two ways of calculation: the first in which the working fluid is water and the second in where water is replaced by a solution consisting of 50% glycol and 50% water.

1. Introduction
Fluent program is used for various applications of engineering such as: modeling heat transfer and mixture of phases through or around bodies with different geometries and modeling of fluid flow (CFD). The program is versatile being written in the C programming language. It is characterized by a control and maneuverability, easy enough; memory being allocated dynamically and databases being some of the most effective . The program consists of five individual modules interconnected, namely: Geometry, Mesh, Setup, Solution and Results. Thus, following each module separately, we can get credible results.

2. Math presentation of turbulent model used in the program Ansysy-Fluent
In order to get results worthy of the truth we chose k-epsilon model turbulent; and to be able to define the flow of fluids we've used Navier-Stocks equations and the equation of continuity.

$$\bar{F} - \frac{1}{\rho}\nabla p + \upsilon\Delta\bar{v} = \frac{d\bar{v}}{dt} \tag{1}$$

$$\begin{aligned}
F_x - \frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon\left(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2}\right) &= \frac{\partial v_x}{\partial t} + \frac{\partial v_x}{\partial x}v_x + \frac{\partial v_x}{\partial y}v_y + \frac{\partial v_x}{\partial z}v_z; \\
F_y - \frac{1}{\rho}\frac{\partial p}{\partial y} + \upsilon\left(\frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2}\right) &= \frac{\partial v_y}{\partial t} + \frac{\partial v_y}{\partial x}v_x + \frac{\partial v_y}{\partial y}v_y + \frac{\partial v_y}{\partial z}v_z; \\
F_z - \frac{1}{\rho}\frac{\partial p}{\partial z} + \upsilon\left(\frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2}\right) &= \frac{\partial v_z}{\partial t} + \frac{\partial v_z}{\partial x}v_x + \frac{\partial v_z}{\partial y}v_y + \frac{\partial v_z}{\partial z}v_z.
\end{aligned} \tag{2}$$

$$\frac{d\rho}{dt} + \rho \cdot \nabla\bar{v} = 0 \tag{3}$$

Also, in order to define the k-epsilon model turbulent I used the following equation: equation for kinetic energy; Epsilon loss and equation which defines energy, at the same time respecting the equations of continuity and Navier-Stoks's.

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla\cdot(\rho_m \bar{v}_m k) = \nabla\cdot\left(\frac{\mu_{t,m}}{\sigma_k}\nabla k\right) + G_{k,m} - \rho_m\varepsilon \tag{4}$$

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla\cdot(\rho_m \bar{v}_m \varepsilon) = \nabla\cdot\left(\frac{\mu_{t,m}}{\sigma_\varepsilon}\nabla \varepsilon\right) + \frac{\varepsilon}{k}\left(C_{l\varepsilon}G_{k,m} - C_{2\varepsilon}\rho\varepsilon\right) \tag{5}$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla\cdot\left(\bar{v}(\rho E + p)\right) = \nabla\cdot\left(k_{eff}\nabla T - \Sigma_j h_j \bar{J}_j + \left(\bar{\tau}_{eff}\bar{v}\right)\right) + S_h \tag{6}$$

Where:

$$\rho_m = \sum_{i=1}^{N}\alpha_i\rho_i \tag{7}$$

$$\bar{v}_m = \frac{\sum_{i=1}^{N}\alpha_i\rho_i\bar{v}_i}{\sum_{i=1}^{N}\alpha_i\rho_i} \tag{8}$$

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon} \tag{9}$$

$$G_{k,m} = \mu_{t,m}\left(\nabla\bar{v}_m + \left(\nabla\bar{v}_m\right)^T\right) : \nabla\bar{v}_m \tag{10}$$

where:
k_{eff} - the effective conductivity,
J_j – flow diffusion,
S_h – the amount of heat due to chemical reaction.
In the 3rd equation we have:

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \quad (11)$$

h – enthalpy for ideal and real fluids

$$h = \Sigma_j Y_j h_j \quad (12)$$

$$h = \Sigma_j Y_j h_j + \frac{p}{\rho} \quad (13)$$

$$h_j = \int_{T_{ref}}^{T} c_{p,j} dT \quad (14)$$

3. Modelling of heat exchanger and its mesh

The geometry of the heat exchanger in Ansys-Fluent software was rather elaborate, since the module dealing with planning cabinets borrows many features from programs CAD (Computer Aided Design). For this simulation we built a multitubular counter current heat exchanger with two passes, composed of twelve copper tube bent so as to form each tube profile "U". Diameter pipes that make up the beam is 22 mm; and the number of input and output sockets is 42.4 mm (11/4 inches) and a length of 100 mm. The mantle of insertion pipe is made of steel beam with inside diameter of 108 mm. To intensify the heat exchange between the primary fluid, seawater and the secondary one, ware attached three baffles arc shape, with a thickness of 5 mm at a distance of 200 mm between them. These latter have a length equal to 2/3 of the internal diameter of the exchanger.

Figure 1. a) View of heat exchanger. b) Longitudinal section thru heat exchanger. c) Mesh of the heat exchanger.

Thus the heat exchanger, which relates to the heat pump, has the following geometrical characteristics: length: 660 mm, diameter 108 mm, the mantle: diameter copper tubes: 22 mm; shicane: 3 mm thick pieces positioned at 150 mm distance between them.

To be able to accomplish the mesh, in Ansys - Fluent, we have the option to create mesh networks to unstructured and structured; based on the set of cell types of discretization. For a computing easier and faster is always attempts to create a mesh network with a rectangular structure, based on a small number of cells, which consist the subject. The time is decreasing, in this case, due to the emergence of a small number of faces and vertices.

Since the geometry of the exchanger presents a regular shape, could conduct a structured network consists of: 1,032,574 nodes; 2,228,283 tetrahedrons; 1.248.473 outlines and 109,899 hexahedron. Due to the regular form, we managed to achieve a uniform distribution of the cells on field of fluid dynamics.

Thus the field is made up of very small cells in order to be able to view a as smooth and precise temperatures, speeds and pressures.

4. Calculation of heat transfer between the primary fluid, sea water, and the secondary one

Considering the fact that many types of heat pumps use water as the working fluid, a mixture of water and glycol or freon, ware imposed two directions of computing: the first in which the working fluid is water and the second the water is replaced with a solution consisting of 50% glycol and 50% water.

The flow of coolant through the pipes and piping among them (i.e. around them) is variable in time and produces turbulence. As models for solving them using Ansys-Fluent offers a series of three main methods: Scale Adaptive Simulation (SAS), Large Eddy Simulalion and RANS – Reynolds Average Navier Stokes that includes k-models, k-epsilon omega, Reynolds Stress Detached Eddy Simulation and k-kl-omega. In the first case studied, working fluids are water and water respectively

The conditions for this case are:
- Entry speed of sea water: 1[m/s],
- Speed of technological water: 0,1[m/s],
- Density of sea water: 1050 [kg/m3] ,
- Density of technological water: 1000 [kg/m3],
- Sea water temperature :22[◦C],
- Technological water temperature : 11 [◦C],
- Acceleration due to gravity: 9,81 [m/s2]
- Dynamic viscosity:1,003*10-3 [kg/ms],
- Thermal conductivity of water: 4182 [J/kg].

For the second case I used as primary fluid all sea water as the working fluid, and a mixture of 50% water and 50% glycol.

The conditions for this case are:
- The entry speed of sea water: 1 [m/s],
- Entry speed of mixture: 0.1 [m/s],
- Sea water density: 1050 [kg/m3],
- Mixture density: 1056 [kg/m3],
- Sea water temperature: 22 [◦C],
- Mixture temperature: 11 [◦C],
- Acceleration due to gravity: 9.81 [m/s2],
- Dynamic viscosity: 1.5*10-2 [kg/ms],
- Thermal conductivity of water: 4182 [J/kgK].

Considering the fact that in this case we have a turbulent flow (Reynolds numbers) solution would converge after a number of iterations, which implies a long calculation rather prolonged. For flow among the pipes have the relationship:

$$\mathrm{Re} = \frac{v \cdot d}{\nu} = \frac{1 \cdot 0.1 \cdot 10^6}{1.003} = 10^5 \tag{15}$$

$$\mathrm{Re} = \frac{v \cdot d}{\nu} = \frac{1 \cdot 0.022 \cdot 10^6}{1.003} = 22 \cdot 10^2$$

As to decrease convergence time I followed the following steps described below

a) I modeled the flow using the model of turbulent frictional (complying with his equations Navier-Stoks) for a non-uniform flow. Thus in the converged solution 2315 iteration but the cell fluid has not come to go through the whole heat exchanger. Transient flow (velocity varies in time and in space) characterizes the Navier Stokes's equation. For solving equations in numerical terms, you must set a time step, Δ t. The step value of time is chosen depending on the geometry and flow type. As the values of reference in the recommendations of the programme between five and ten iterations.

Considering the type of flow that occurs in the exchanger, a smaller time step will require a faster convergence and greater stability; While a larger step, exactly the opposite. Because the solution is not divergence must be complied with current number, $C = \frac{v \cdot \Delta t}{\Delta x}$ (16), where Δx represents the smallest cell size and v is the velocity. Number must not exceed the value 1. It is recommended that a value of 0.2
Such:

$$\Delta t = \frac{c \cdot \Delta x}{v} = \frac{0.2 \cdot 0.002}{0.1} = 4 \cdot 10^{-3} \qquad (17)$$

For greater safety, and I worked with a time step $\Delta t = 0.001$ so that the condition is fully satisfied. Using the data already obtained I managed to explain a distribution of velocities, pressures and temperatures; taking place in convergence of iteration.
b) Whereas I wanted a faster convergence of the solution, in the beginning I did not use the equations. They have been used after 3x104 iterations. I added his equations Navier-Stoks and energy equation and I have run once again the simulation.

We have adopted this solution because the solution diverges if the three were met at the same time. All simulations were run for both the summer and the winter, where temperatures of the two agents are much lower

5. Representation of heat exchange characteristics parameters

In figures 2 and 3 are temperature variations, along the direction of flow, on outer and inner part of the heat exchanger.

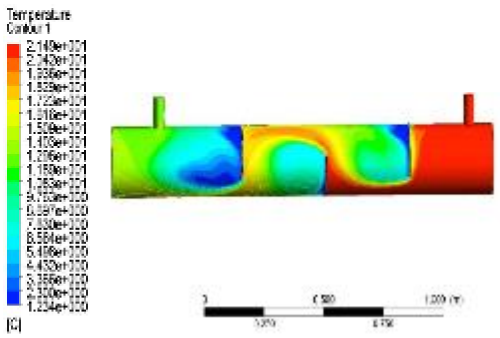

Figure 2. Summer variation of the temperature on the surface of the exchanger.

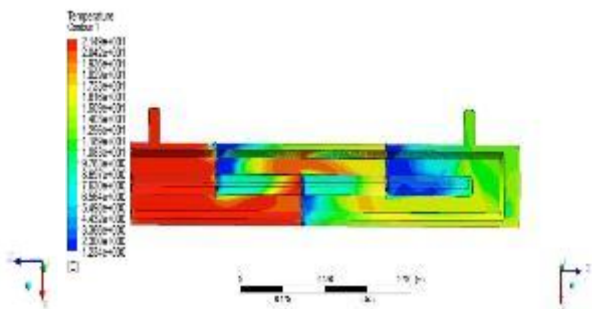

Figure 3. Temperature variation inside the exchanger.

The variation of the temperature on the outside of the copper pipes and inside them can be seen in figures 4 and 5. At the same time these two figures show, the heat transfer through the pipe structure.

IOP Conf. Series: Materials Science and Engineering **227** (2017) 012103 doi:10.1088/1757-899X/227/1/012103

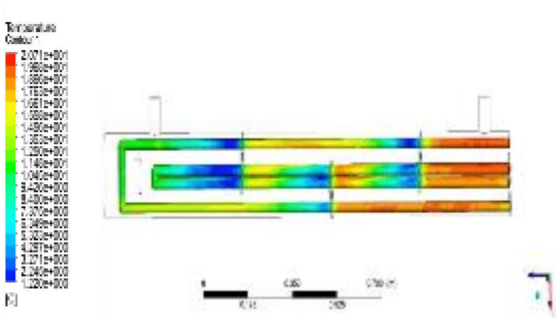

Figure 4. Winter temperature variation inside the exchanger.

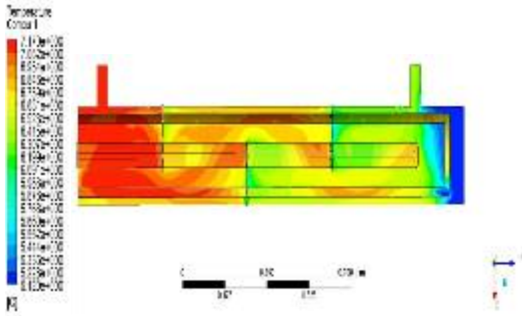

Figure 5. Winter temperature variation on exterior of copper pipes.

The simulation was made for the two seasons, summer and winter, in order to be able to detect the amount of heat that can be extracted from sea water, from the same depth-5 m, respectively. Figures 6 and 7 put in value the temperatures inside the exchanger and on the faces of the outer and inner copper pipes.

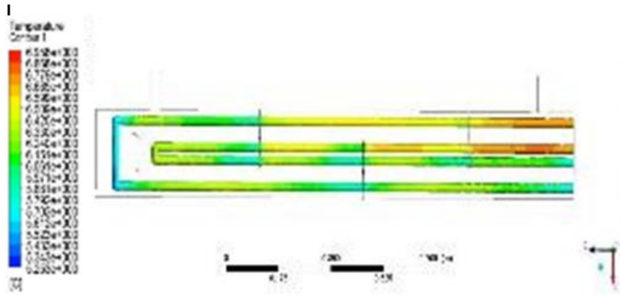

Figure 6. Winter temperature variation on the interior of copper pipes.

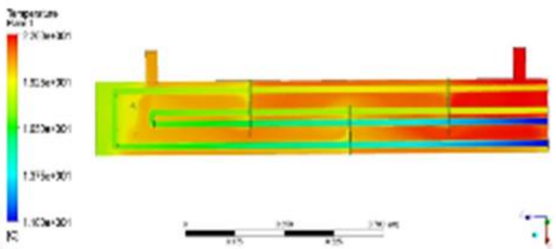

Figure 7. Temperature variation of the two agents.

Figure 7 shows us quite explicitly how the exchange of heat takes place. Comparing the way the distribution of temperatures in the exchanger, observe a variation of their fine and a different way of presenting. Thus we can conclude that the numerical program ANSYS-FLUENT offers a better picture of the turbulent flow the primary agent. This is due to the use of a structured mesh network. In addition the program also allows viewing and other important elements for heat exchange as well as current lines of particles as shown in figure 8, and particle velocity vectors you primary fluid in figure 9.

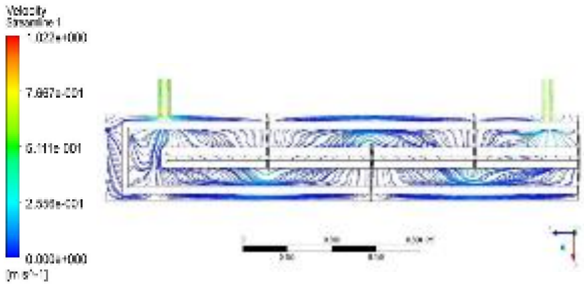

Figure 8. Primar agent speed vector variation.

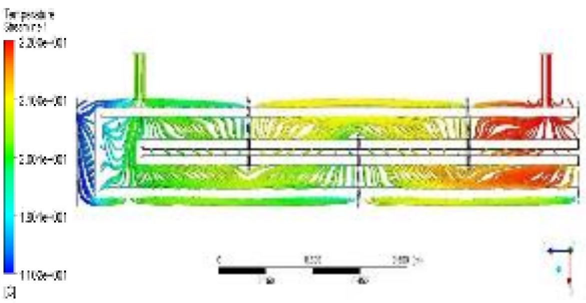

Figure 9. Primar agent temperature vector variation.

For a better example of heat exchange taking place between the two fluids I chose a line from the current passing through the middle of the pipe, and who follows them, with the two trail passes, until leaving the heat exchanger.

	Temperature [C]	
Z [m]	I	II
0	4.00002	6.35085
0.155556	4.23062	6.28052
0.311111	4.50793	6.10561
0.466667	4.72866	5.99856
0.622222	4.93008	5.86251
0.777778	5.08743	5.82772
0.933333	5.21343	5.77259
1.08889	5.33697	5.72005
1.24444	5.42724	5.6821
1.4	5.52694	5.55071

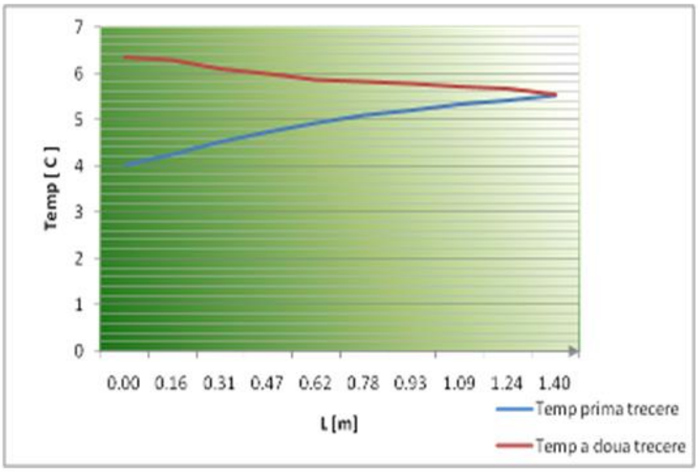

Figure 10. Winter temperature variation of the secondary agent , measured in current line.

Therefore, figure 10 illustrates the variation in temperature along the line of current in the two passages. It can be seen with temperature increases until the secondary agent elbow pipe, the return of stagnating along its, and then seems to continue increasing until exiting the exchanger. The same simulation is presented in figure 11 for the summer period.

	Temperature [C]	
Z [m]	I	II
0	11.00000	19.8125
0.155556	11.38842	19.58655
0.311111	11.95320	19.15442
0.466667	12.43880	18.68225
0.622222	12.85002	18.10251
0.777778	13.36859	17.5221
0.933333	13.99012	16.9325
1.08889	14.46621	16.56877
1.24444	15.01224	15.9772
1.4	15.40125	15.52772

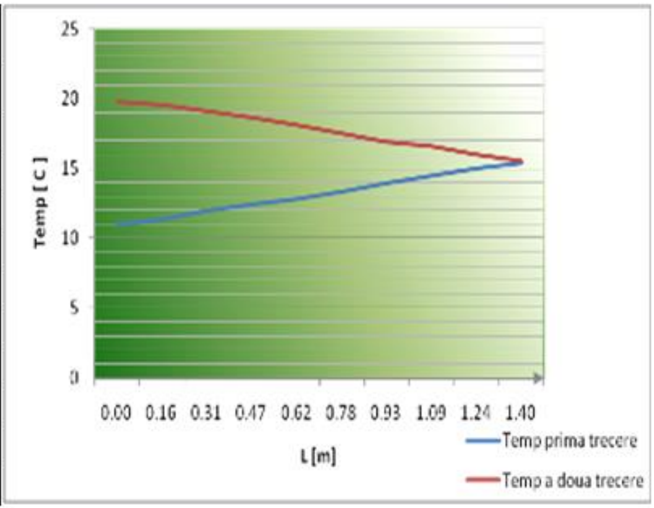

Figure 11. Summer temperature variation of the secondary agent, measured in current line.

6. Conclusions

Study of turbulent flow of a fluid whose Reynolds number is high, and the study of heat transfer, put issues of scientists since its inception; This phenomenon is quite difficult to implement in a numeric calculation program due to its complexity. The network conceived and presented in the program of calculation provides a clear vision of the parameters characterizing the thermal Exchange, the disadvantage represented by the smoothness of the outcome (good results but containing errors will be accepted). Ansys Fluent numerical program presents a very clear distribution of all parameters involved providing some spectacular results in terms of graphics. It offers the advantage of much better understanding of the phenomenon of heat transfer by all involved parameters: speed, temperature and density.

7. References

[1] Apetroaei Șt 1983 *Evaluarea şi prognoza bilanţului apei din sol (Evaluation and forecast of soil water balance)* (Bucharest: Ceres)
[2] Ataman H1991 Design of ground-source heat pump systems *MSc Thesis in Mechanical Engineering* (Turkey: Istanbul) 15
[3] Cataldi R H and Lund J W Stories from a Heated Earth *Geothermal Resources Council and Int. Geothermal Association*
[4] Cohut I and Bendea C 1999 Geothermal Development Opportunities in Romania *Geothermische Energie* **24-25** (Jahrgang Heft, Zuruck)
[5] Cui P, Yang H and Fang Z 2008 *Energy and Buildings* **40** 1060–1066
[6] Dinu D 1999 *Hydraulics and hydraulic machines* (Sigma Trading Metafora Publisher: Constanta)
[7] Dinu D 2010 *Mecanica fluidelor pentru navigator (Fluid mechanics for the navigator)* (Nautica Publisher, Constanţa) ISBN 978-806-8105-11-6
[8] Florescu I 2007 *Mecanica Fluidelor* (Alma Mater Publisher: Bacau)
[9] Fujii H and Akibayashi S 2002 Analysis of thermal response test of heat exchange wells in ground-coupled heat pump systems *Shigen-to-Sozai* 75–80
[10] Gavriliuc R 1999 *Pompe de căldură - de la teorie la practică (Heat pumps - from theory to practice)* (MatrixRom Publisher) ISBN 973-9390-55-2.
[11] Gehlin S 2002 Thermal response test-method development and evaluation Ph.D. Dissertation
[12] Iordache F and Băltărețu F 2002 *Modelarea și simularea proceselor dinamice de transfer de căldură (Modeling and simulation of dynamic heat transfer processes)* (MatrixRom Publisher)
[13] Scupi A-A 2013 *Research on Dynamics of Unconventional Traction Systems and Suspensions* PhD thesis, Constanta Maritime University, Constanta, Romania