Le 3ème Séminaire International sur les Energies Nouvelles et Renouvelables
The 3nd International Seminar on New and Renewable Energies
Unité de Recherche Appliquée en Energies Renouvelables, Ghardaïa – Algérie 13 et 14Octobre 2014

Heat transfer and pressure drop in a fluid/solar spiral tube exchanger

T. Arrif[1], A. Belaid, H. Merarda, A. Gama,R. Khelifi, F.Bedaouche

Unité de Recherche Appliquée en Energies Renouvelables, URAER, Centre de Développement des Energies Renouvelables, CDER, 47133, Ghardaïa, Algérie
[1]arriftou@yahoo.fr

***Abstract*—This work presents the results of simulation study performed on pressure losses and *Nusselt* number in laminar water-flow and spiral tube channel-coils with two different section that are widely used for industrial application. *Reynolds* number was applied to describe the influence of the velocity inlet. The results obtained are in accordance with those of other authors. In our case the vertical coil pressure drop in different sections is compared. Investigation indicates that the section geometry influence the pressure drop and there is pressure drop variation with respect to *Re*.**

Keywords: spiral tube, Pressurelosses, heat transfer

I. Introduction

In a solar thermal power tower system, the cylindrical receiver (heat exchanger tube) is a key component for solar concentration system. It is thereforereasonable tolearnalsomoreaboutthepressuredropintheabove mentioned spiral tube water-flow. Relationship between pressure drop and heat transfer is important for the efficient and solar thermal heating and cooling.

The main obstacle to the development of solar tower technology is the significant investment required by the construction of a field of heliostats and tower. Naphon [2] studied the characteristics of the heat transfer and flow of a horizontal spiral coil tube by numerical and experimental method.

In actual solar thermal power, because of the different concentrated solar radiation and different incident angle, the heat transfer is uneven along the circumference. The local temperature, heat flux and heat transfer coefficient of the tube receiver need to be obtained when the solar plant operates normally [3].

Concerns the solar heat exchanger fluid (water/solar) high temperature we designed a spiral tube geometry with two variable sections. The aim is to seek optimum designs of spiral ducts for solar applications where the cross section area is fixed. The computational domains in Figures 4 were created in *GAMBIT (2.3. 16)* software then simulated in *FLUENT(6.3.26)* and then will be integrated and simulated in *TRNSYS*. The solar receiver (heat exchanger) presented in this work consists of one single spiral tube arranged vertically. In this study we simulate heat transfer spiral tube operating with water as the working fluid and one half circumference of the tube receiver receives the solar radiation energy, and the second half is supposed adiabatic surface covered with heat insulator. To compare the two geometries, numerical simulations were carried out for fixed cross section area and pipe length. The results of numerical simulations in this document show the particular distribution of *Nusselt* number and temperature.

II. Model development

In this study, incompressible laminar Newtonian fluid flows in-side a spiral duct with two cross sections is taken into account. The configurations of in-plane spiral ducts and their respective cross section schematic are displayed in Figure.1and 2; the detailed geometric parameters are presented in Table I. In this work we consider one side of tube called the heating surface receives constant heat flux from the solar radiation Q_{Net} = 10000 W/m2, and the other side is called adiabatic surface covered with heat insulator. Figure. 2 shows coordinate system and thermal boundary conditions.

The water enters and flows through the spiral pipe with the velocity and the temperature inletU_{in} and T_{in} = 303 K, it absorbs heat and its temperature increases, and exits and flows through the outlet, in this study the properties of the water are constant. Conservation equations for mass, momentum and energy for the flow inside the ducts are given by [4],[6]:

$$\nabla.\rho u = 0 \tag{1}$$

$$\nabla.(\rho u \otimes u) = -\nabla p + \nabla[\mu(\nabla u)^T] \tag{2}$$

$$\rho c_p . \nabla(u.T) = k\nabla^2 T \qquad (3)$$

ρ: Density of liquid water,
at 30c° : ρ (kg/m3)= 995.6502
u,v,w: fluid Velocity in xyz
P : pressure, pas
μ : fluid dynamic viscosity
C_p: Constant-pressure heat capacity
at 30c° : CP (J/(g·K) = 4.1784
K: thermal conductivity of the fluid
K [$W \cdot m^{-1} \cdot K^{-1}$] =0.609
T: temperature.

TABLE I DIMENSIONS OF THE SPIRAL TUBE.

Parameters	Dimensions (mm)	
	circular Section	*semi circular Section*
Inner diameter of tube, d	20	28
Innermost spiral tube diameter,	140	140
Outermost spiral-coil tube diameter ,	680	720
Length of spiral tube,	6365	6365
Pitches,	30	30
Number of coil turns,	5	5

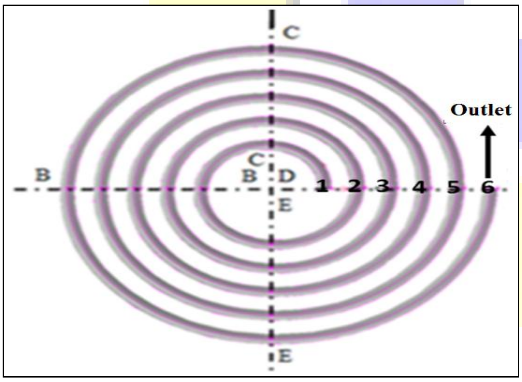


Fig. 1 Schematic diagram of the spiral tube.

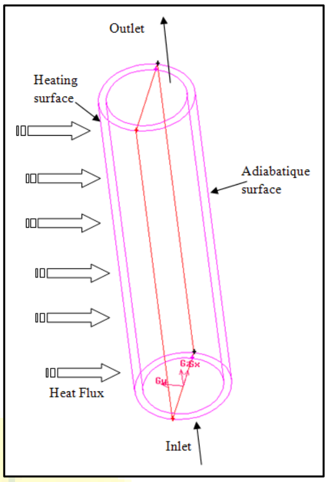


Fig.2 Schematic of coordinate system and thermal boundary conditions.

III. BOUNDARY CONDITIONS AND NUMERIC COMPUTATION:

GAMBIT (2.3.16) is used to mesh the two geometries shown in Figure 4 with a Tetrahedron volume element. The boundary definitions are *VELOCITY INLET* for inlet boundary, *OUTFLOW* for outlet boundary, *WALL* for other boundaries, *and FLUID* for water in the tube, and *SOLID* for tube wall.

The mathematical model given by Equations. (1)– (3), together with appropriate boundary condition and constitutive relations comprising five dependent variables – u, v, w, P and T – was solved using the commercial code *Fluent 6.3.26* based on finite volume method. The*SIMPLER* (*Semi-Implicit Pressure-Linked Equation Revised*) algorithm is employed to deal with the problem of velocity and pressure coupling *Patankar* (1980)[5], the steps of this algorithm are almost the same as the *SIMPLE* algorithm with the difference that he neglect least significant terms in the velocity correction equations. Second-order upwind scheme and tetrahedral grid are used to discretize the main governing equations as shown in Figure. 3.Thenumerical computation is ended if the residual summed over all the computational nodes satisfies the criterion 10^{-4} except for energy equation criteria for residual is 10^{-6}.

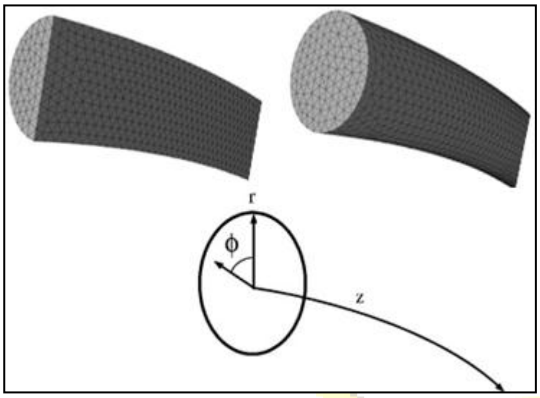


Fig.3 Schematic diagram of the tetrahedron grid system for spiraltube.

IV. Results and Discussion:

In order to evaluate the performance of heat transfer in spiral tubes exposed to a concentrated heat flux, numerical simulations were carried out for two different sections. The cross-sectional area is fixed so we can observe the effects of different cross sections in the flow.

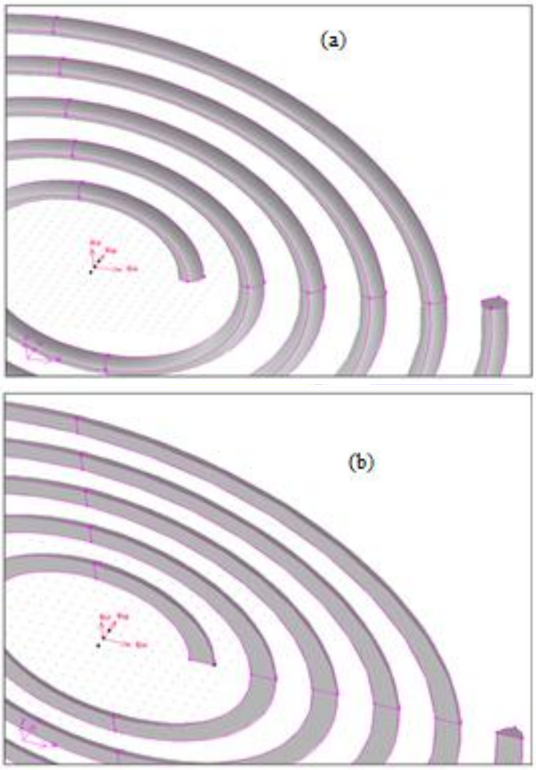


Fig. 4 Computational domain: (a) circular section (b) Semi-circular section.

A. Velocity:

The axial velocity tends to increase towards the outer side of the coil as shown in Figure 6, and it is almost zero in the vicinity of the inner wall and it is due to the centrifugal force which is generated by the spiral path, the strength friction are oriented outwardly.
The presence of the centrifugal force due to the curvature of the coil will induce a significant radial pressure gradient in the central flow region[3]. However, the velocity field and pressure and their profile must be exploited.

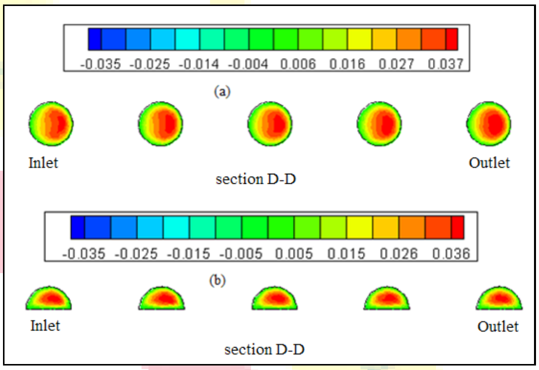


Fig.5Axial velocity, spiral tube: a) circular section b) Semicircular section.

B. Pressure:

Both Figures 6, 7 show that the pressure drop (pressure loss) in the radial direction of the 6-position for a fixed Reynolds number and inlet temperature is higher in the circular cross section than the semicircular section, so the geometry affects the pressure drop.

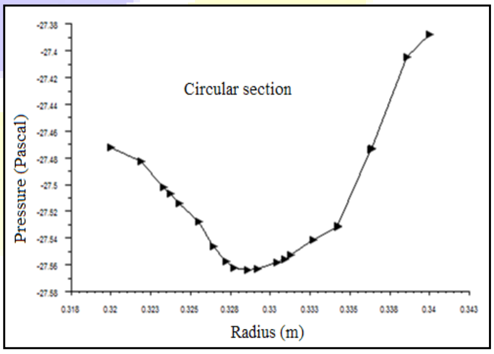


Fig.6 Pressure drop Outlet tube, section D-D, Y=0, Z=0.

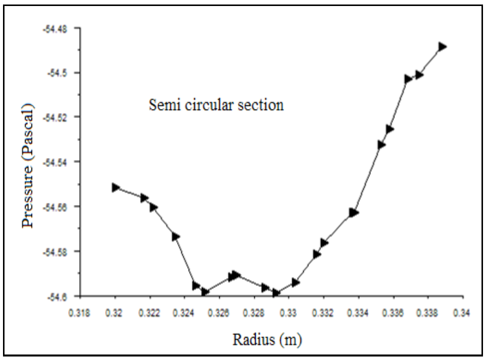


Fig.7Pressure drop Outlet tube, section D-D, Y=0, Z=0.

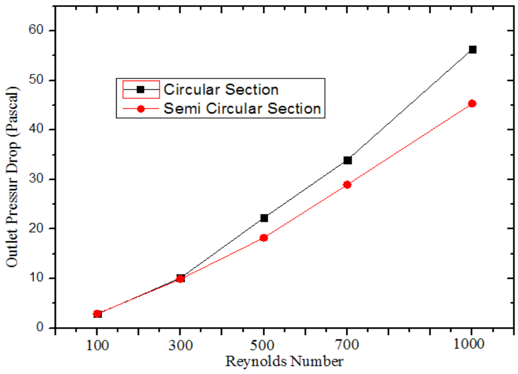


Fig8. Effect of Reynolds number on the vertical spiral tube pressure drop.

Figure 8. Represents the effect of *Reynolds* number on the vertical spiral tube pressure drop, it indicates that there is pressure drop variation with respect to *Re*, what makes the difference in the pumping.

C. Nusselt number:

In general, the coefficient of heat transfer through the fluid can be found by the formula [1]:

$$Nu=h.d/K \text{ and } q=h.(t_w-t_f) \quad (4)$$

Nu: *Nusselt*number
q : heat flux
t_w: inner wall tube temperature.
t_f :fluid average Temperature.

h :convective heat transfer coefficient
The water *Prandtl* number under laminar flow at temperature T_{in}= 303k is [6]:

$$P_r = \frac{\upsilon}{\alpha} = 5.43 \quad (5)$$

P_r:prindlt number

υ : water cinematic viscosity at 303k : υ= 0.804x10 $^{-6}$ m^2/s
α : thermal diffusivity m^2/s ($\alpha=K/\rho.C_p$)

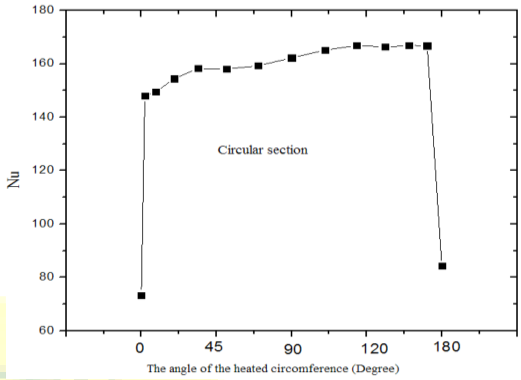


Fig.9Nusseltnumber along the heated circumference, section D-D (position 6), (Re=500).

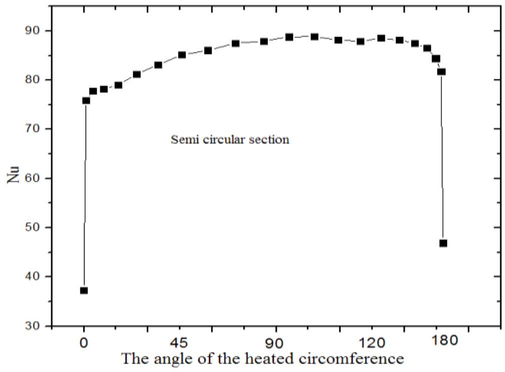


Fig.10Nusselt number along the heated circumference, section D-D (position 6), (Re=500).

Figure. 10 and 11 shows the Nu number change curve at a different angle direction at section D-D (position 6), whenRenumberis 500.

ThevalueofthelocalNunumber for circular section is higher than the one of semicircular section.

D.Pressure and temperature field in three dimension :

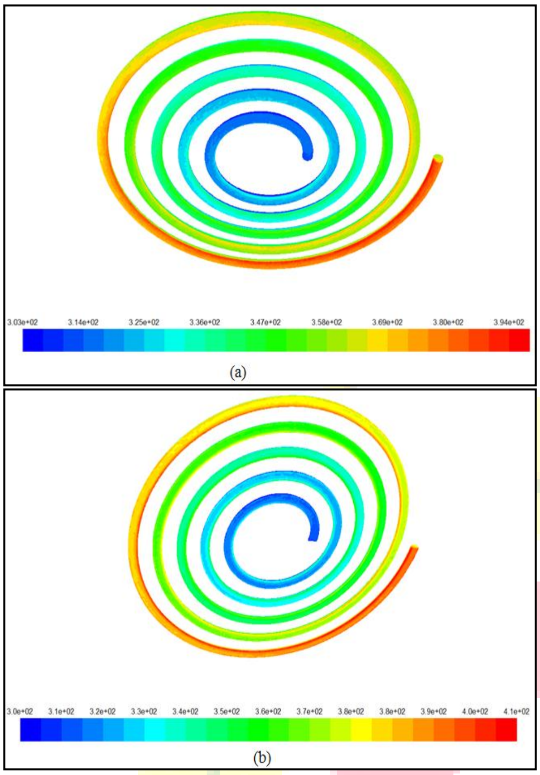
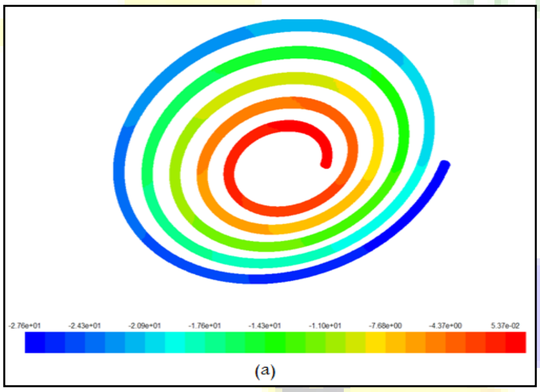


Fig.11Temperature profile: spiral tube: (a) circular section (b) Semi-circular section.



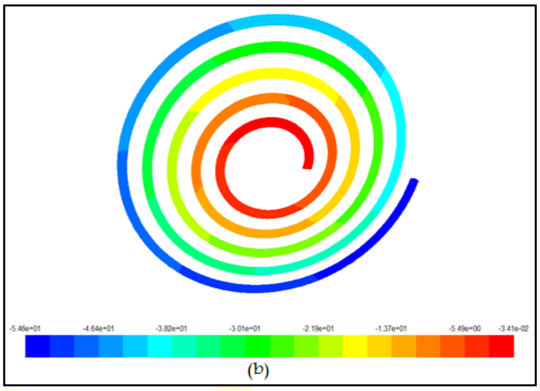


Fig.12Pressure drop (Pascal): spiral tube: (a) circular section (b) semi-circular section.

Figure 11 shows the increase in temperature due to the heat flux, and figure 12 shows the decrease in pressure due to the coil tube geometry.

V. CONCLUSION

In this work we have designed two three-dimensional geometries with *GAMBIT 2.3.16* software, one has a circular cross section and the second has semi-circular cross section, the three-dimensional water laminar flow was simulated with *FLUENT 6.3.26* commercial code that is based on the finite volume method in order to analyzing numerically the characteristics of the heat transfer and flow. These two geometries were imagined in order to be used for exchanger fluid / solar in a solar tower.

We introduced a comparison between two different cross sections of a spiral tube; it was found that section geometry influence the pressure drop and there is pressure drop variation with respect to Re.

The cross section geometry of the tube influences the local*Nusselt* number of the fluid in the heated face.

REFERENCES

[1] Xiaoping Yang ,Xiaoxi Yang, Jing Ding,Youyuan Shao , Hongbo Fan," Numerical simulation study on the heat transfer characteristics of the tube receiver of the solar thermal power tower" Applied Energy, vol. 90, issue 1, pp. 142-147, 2012.

[2] Naphon Paisarn. 'Study on the heat transfer and flow characteristics in a spiral- coil tube'. Int Commun Heat Mass Transfer vol 38: pp. 69–74, 2011.

[3] S.Vashisth, V.Kumar, K.D.P.Nigam, A review on the potential applications of curved geometries in process industry, Ind.Eng.Chem.Res. vol 47, pp. 3291–3337,2008.

[4] Reilly HE, Kolb W J. 'Evaluation of molten salt power tower technology based on The experience of solar two '.SANDIA Report SAND -3674, 2001.

[5] Patankar, S.V. 'Numerical heat transfer and fluid flow', hemisphere publishing corporation Taylor and Francis group, New York (1980).

[6] W. Kays, M. Crawford, B. Weigand, Convective Heat and Mass Transfer, fourth ed., McGraw-Hill, Singapore, 2005.