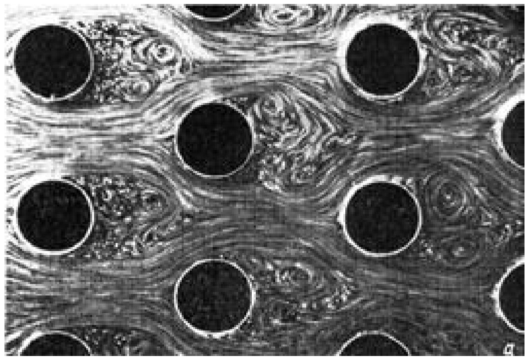
3rd Task: CFD Analysis
Turbulent flow and heat transfer in a tube bank

Authors: Julen Arruti
Victor Navarrete
Date:25/12/2018
Subject: Numerical Methods Of Heat And Mass Transfer
Professor: Ekaterina Kitanina

1. - Problem Definition



➢ Physical Description

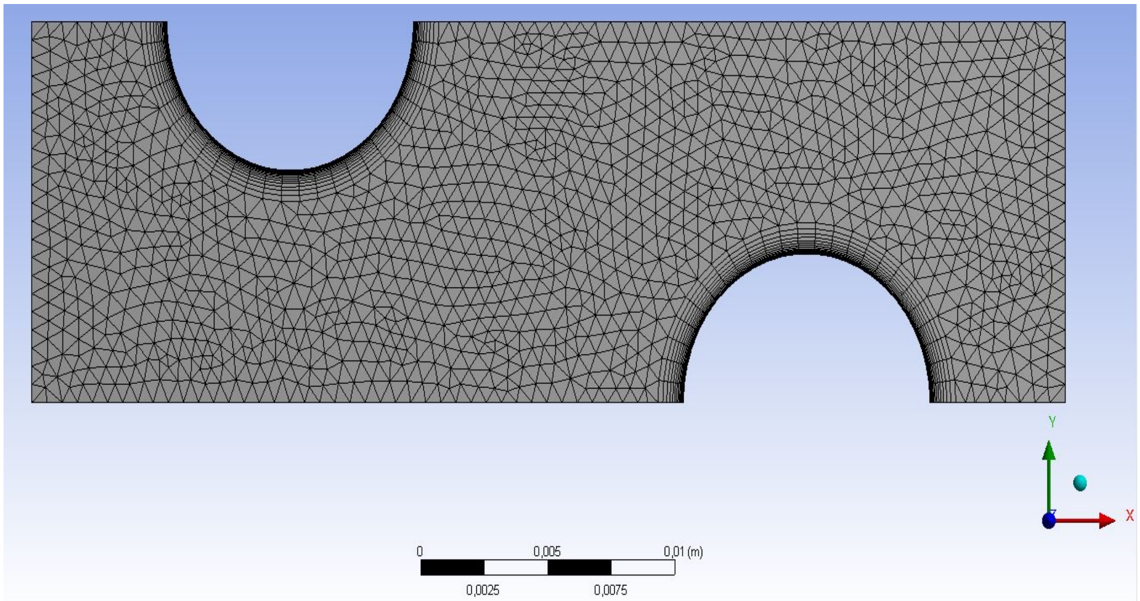
We are going to simulate the turbulences produced inside the case and among the tubes of a heat exchanger. We will modify the velocity of the flow to analyze the turbulences. Moreover, the walls of the tubes inside the case will have a temperature of 400k while the flow's temperature is 300k. In addition, we are going to suppose that the volume we are modeling would be the third row of the tubes due to the fact that the first two rows of the heat exchanger are really unstable.

➢ Numerical Description

We are discretizing the volume that we are analyzing into triangle greed and by the numerical methods we are going to calculate the different values of velocity vectors across the case, the method we are going to use the k-Ɛ method to simulate the turbulences.

2. - Results of the simulation:

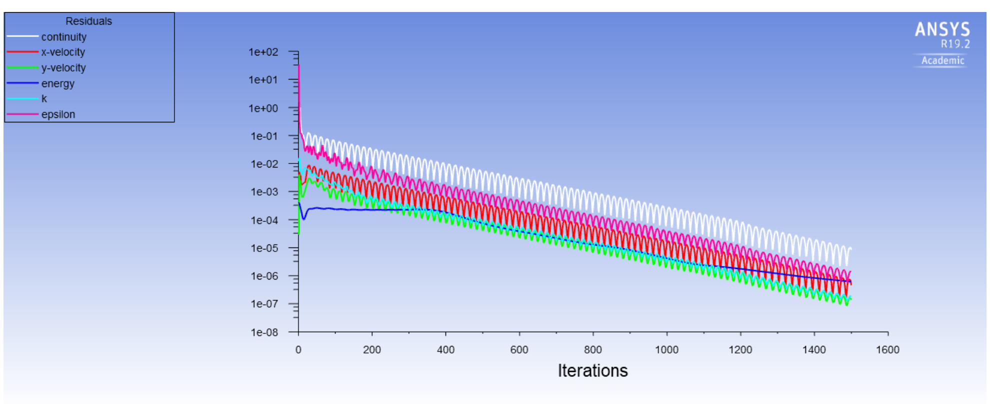
Figure 1.0 Meshes



Source: Ansys simulations solutions (2018)

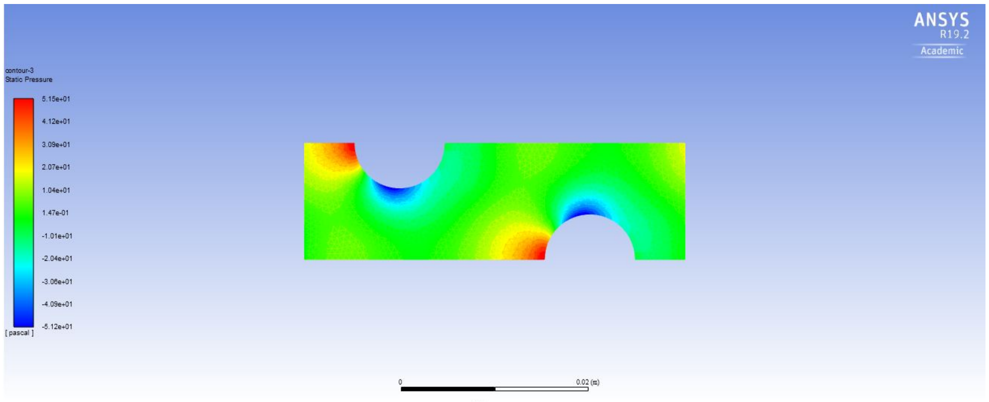
1st Simulation:

Figure 1.1 *Residual Results.*



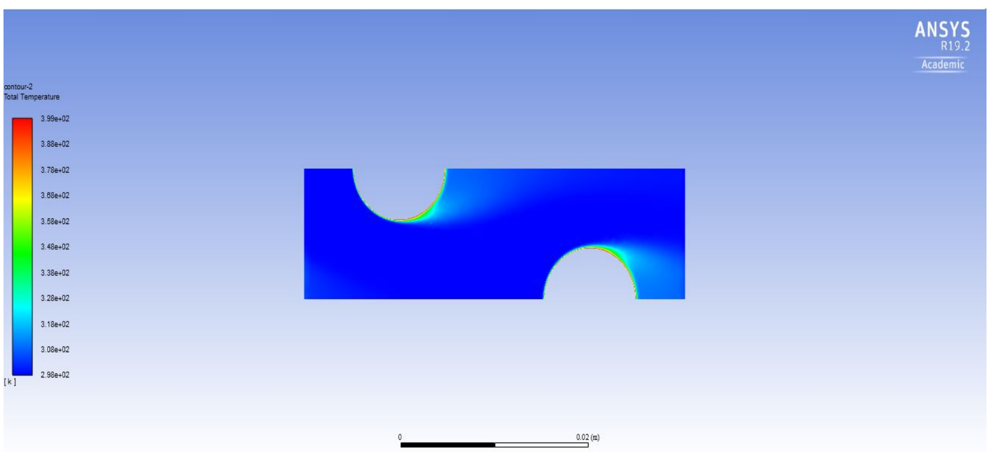
Source: Ansys simulations solutions (2018)

Figure 1.2 Static Pressure



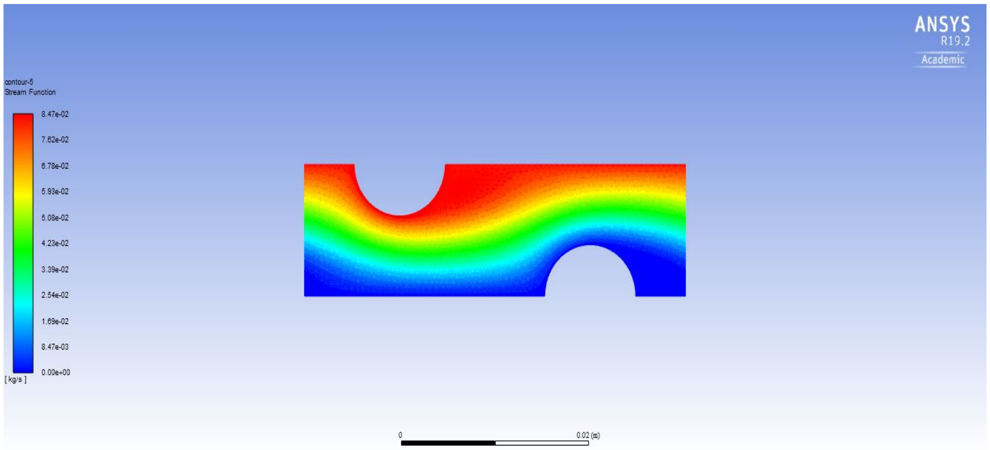
Source: Ansys simulations solutions (2018)

Figure 1.3 Temperature Field



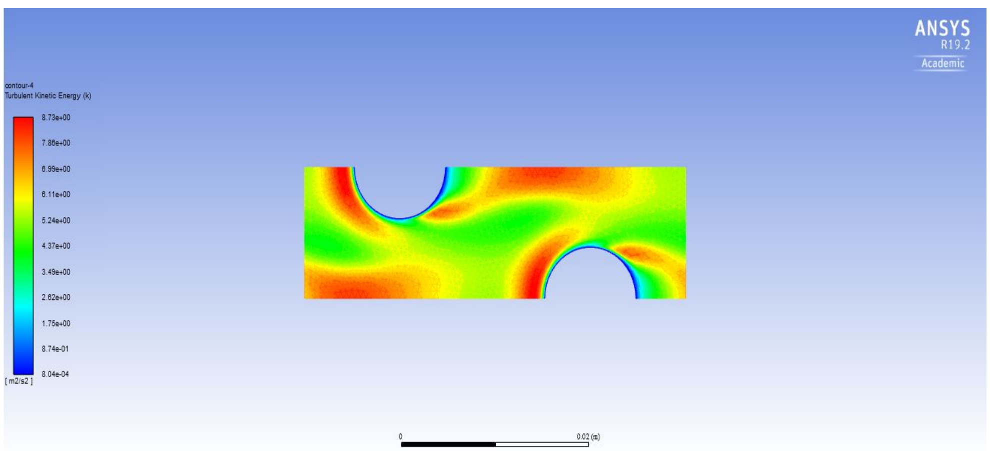
Source: Ansys simulations solutions (2018)

Figure 1.4 Stream function field



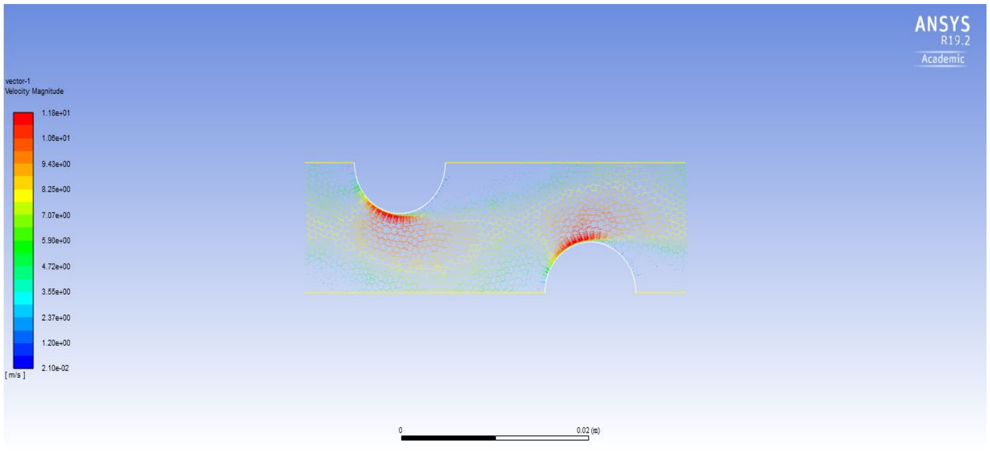
Source: Ansys simulations solutions (2018)

Figure 1.5 Turbulence field



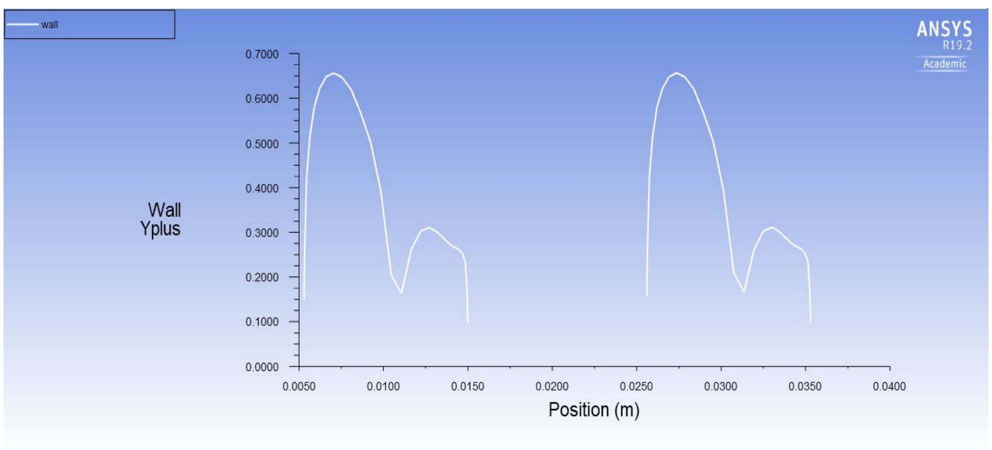
Source: Ansys simulations solutions (2018)

Figure 1.6Velocity vectors



Source: Ansys simulations solutions (2018)

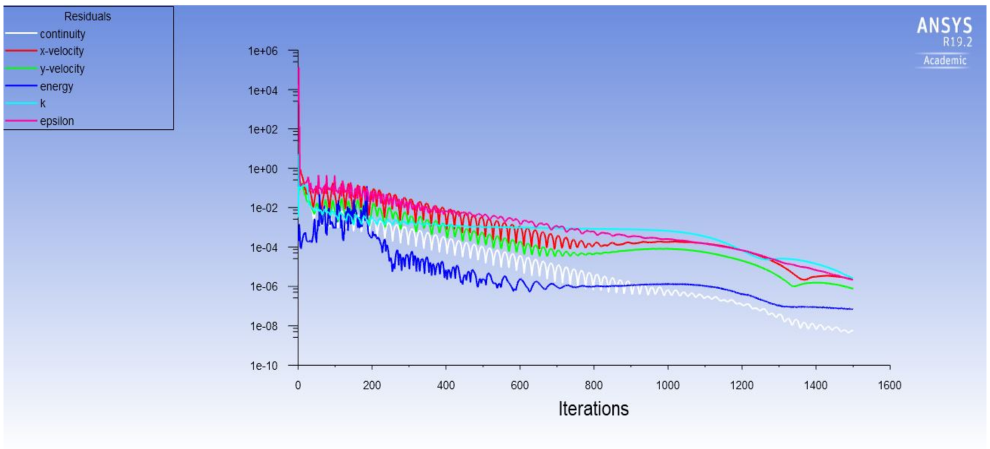
Figure 1.7 Wall Y+



Source: Ansys simulations solutions (2018)

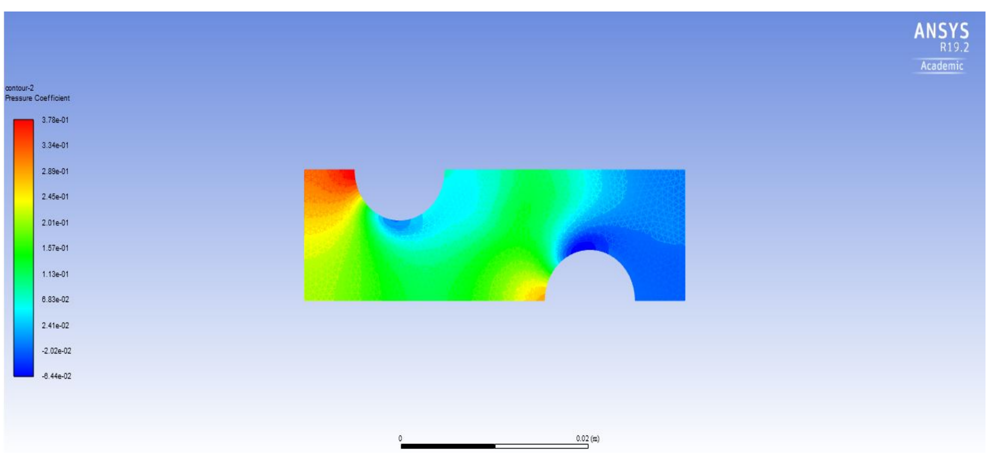
2ndSimulation:

Figure 2.1 Residual Results



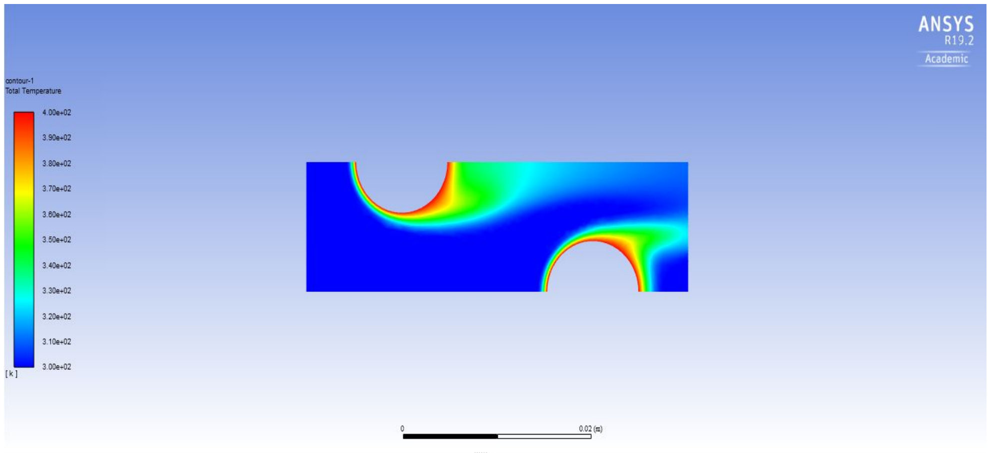
Source: Ansys simulations solutions (2018)

Figure 2.2 Static Pressure



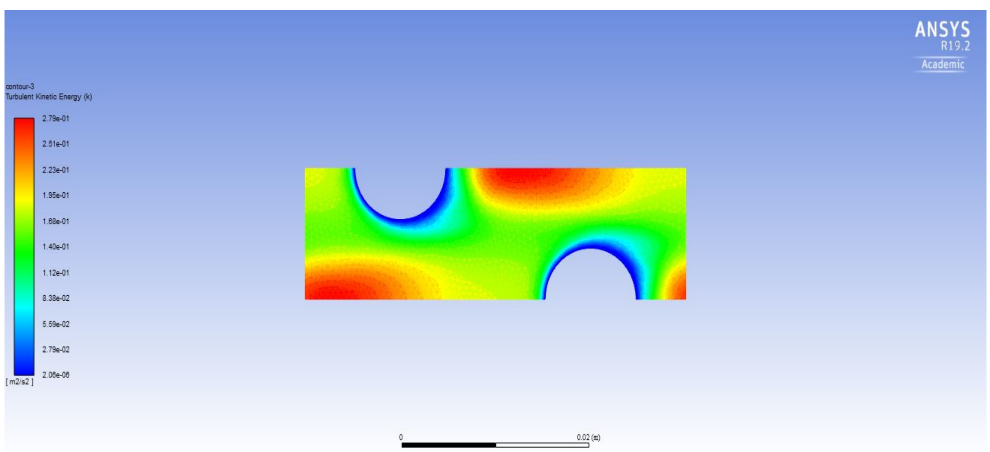
Source: Empirical results (2018)

Figure 2.3: Temperature Field



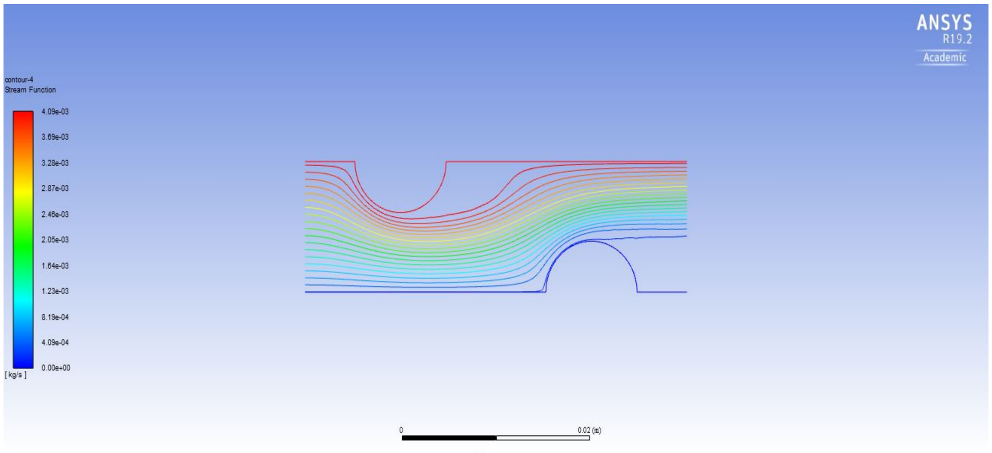
Source: Ansys simulations solutions (2018)

Figure 2.4: Turbulence Field



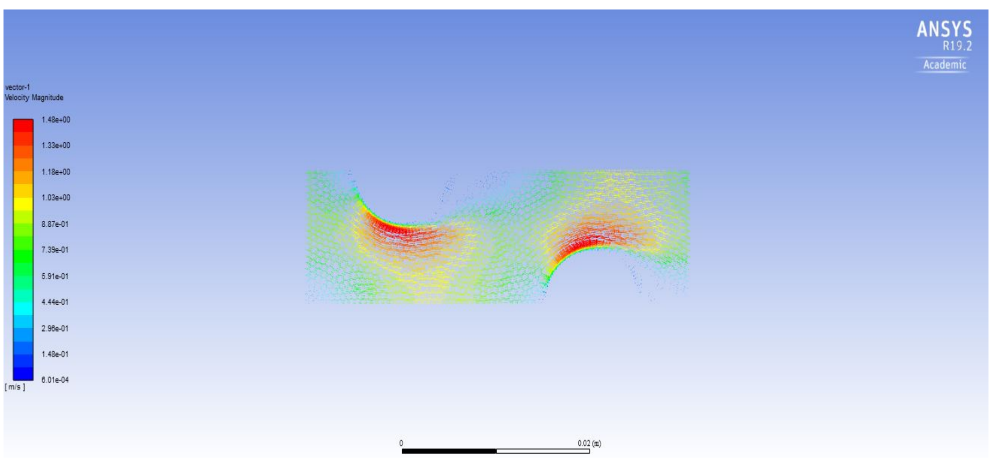
Source: Ansys simulations solutions (2018)

Figure 2.5 Stream function field



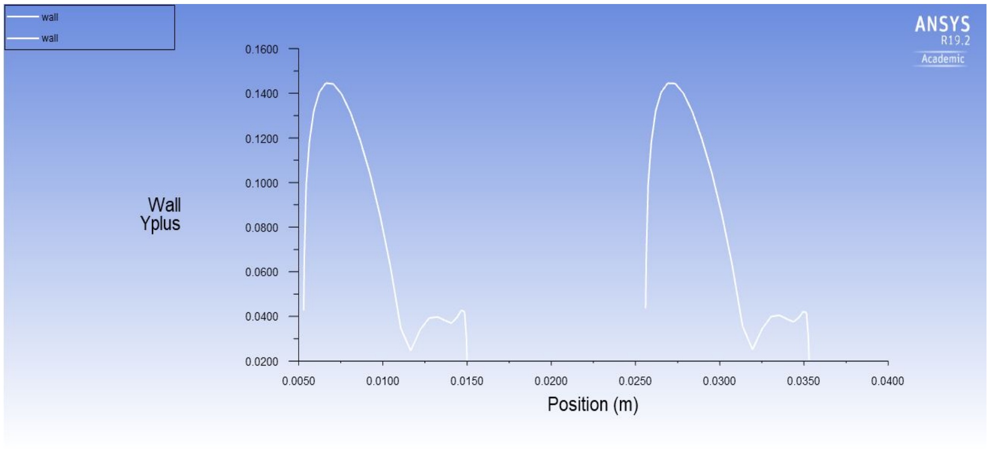
Source: Ansys simulations solutions (2018)

Figure 2.6 Velocity vectors



Source: Ansys simulations solutions (2018)

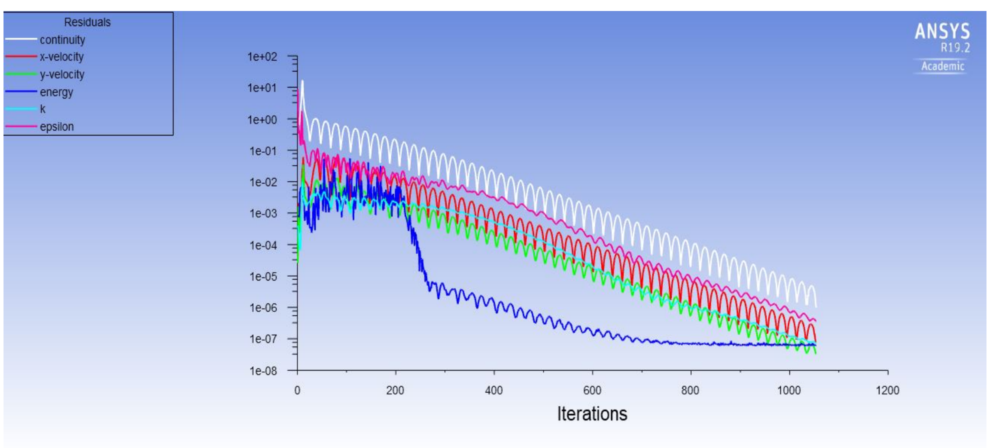
Figure 2.7 Wall Y+



Source: Ansys simulations solutions (2018)

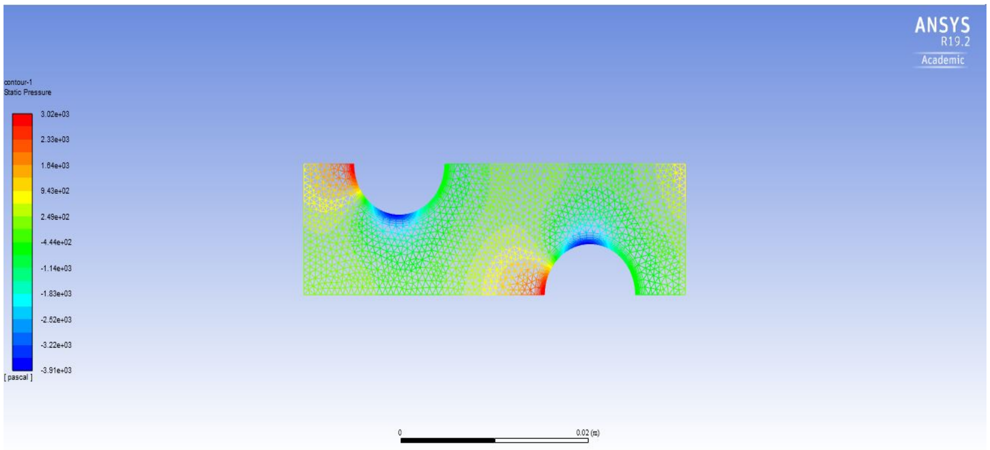
3ndSimulation:

Figure 3.1 Residual Results



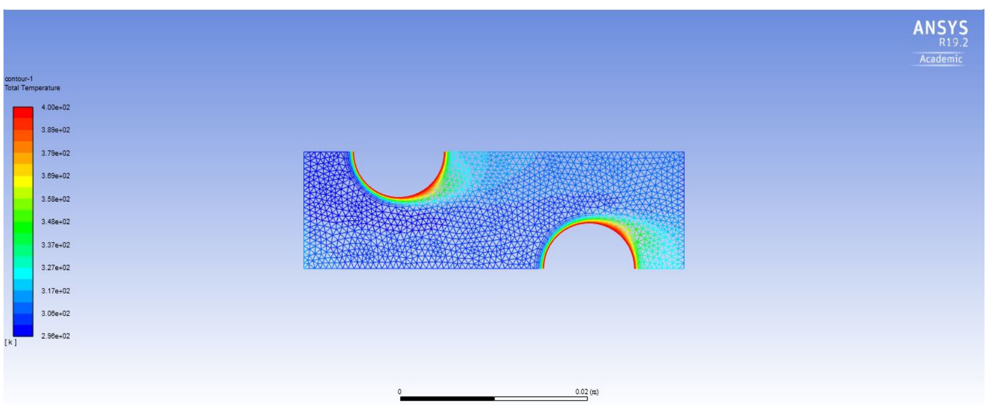
Source: Ansys simulations solutions (2018)

Figure 3.2 Static Pressure



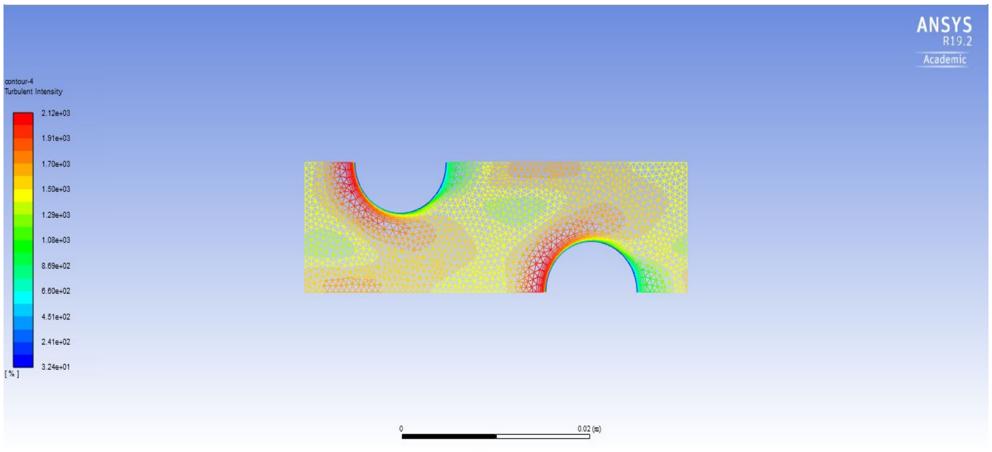
Source: Ansys simulations solutions (2018)

Figure 3.3 Temperature Field



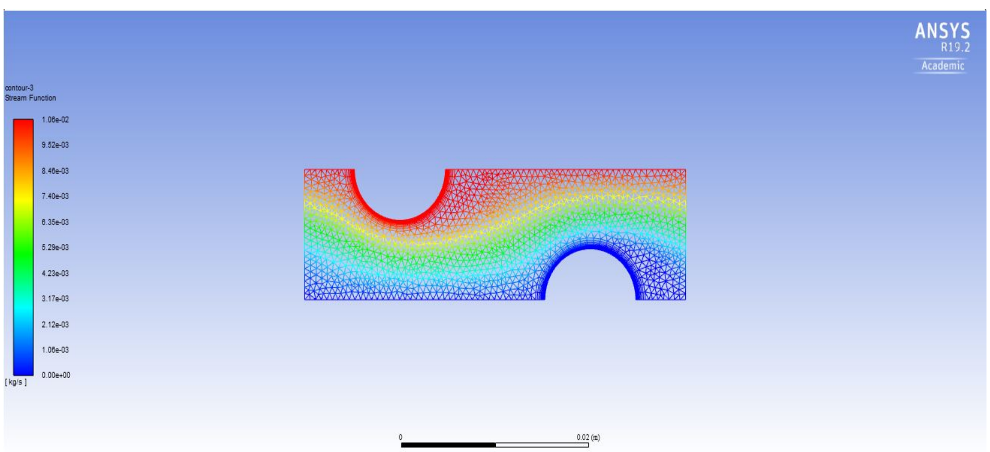
Source: Ansys simulations solutions (2018)

Figure 3.4 Turbulence Field



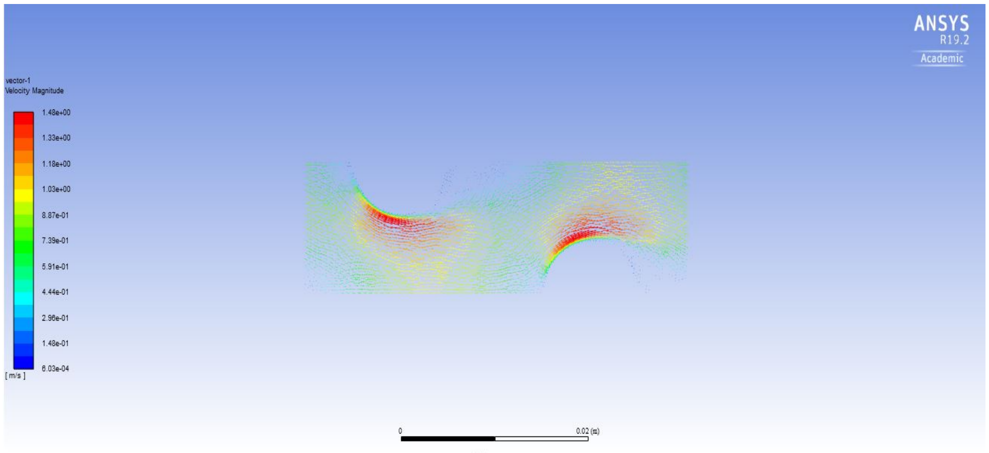
Source: Ansys simulations solutions (2018)

Figure 3.5 Stream function field



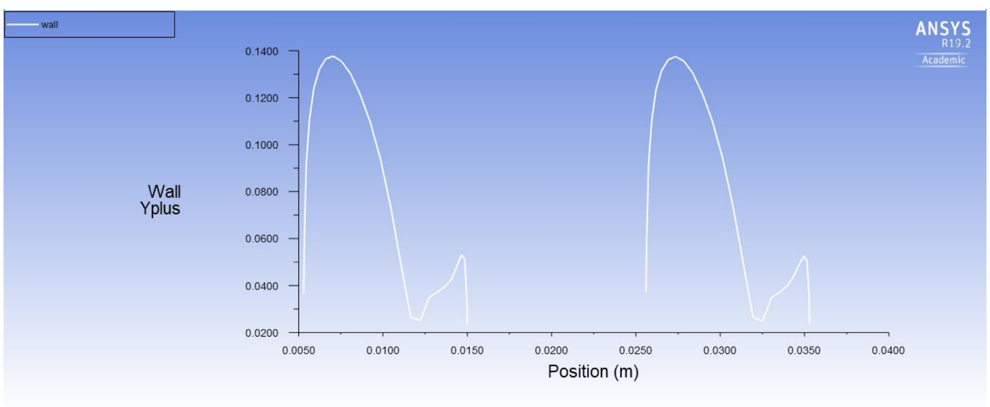
Source: Ansys simulations solutions (2018)

Figure 3.6 Velocity vectors



Source: Ansys simulations solutions (2018)

Figure 3.7 Wall Y+



Source: Ansys simulations solutions (2018)

3. - Analysis of the data:

- **Discretization of the control volume:**

We have designed a triangular fine mesh, with inflation in the near wall region of the tubes and with a maximum size value of the greed of 6e-4mm. We made such a fine greed in order to obtain reliable results. Nevertheless, the fine greed has sometimes produced instabilities.

- **Analysis of the residuals:**

If we watch the results obtained in the simulation of the residuals show us that the value of the residuals obtained in the chart of the experiment, *Fig. 1.1, 2.1 and 3.1* the values are not as small as we would like (1e-6) but they almost reached them after 1500 iterations. To reduce the lack of stability, we were forced to modify the relaxation factors.

- **Static Pressure:**

As we can see in the *Fig. 1.2, 2.2 and 3.2* before the wall there is an increase in the local pressure. In this region water flow is slowed down by the walls and that is why this raise of pressure occurs. On the top of the tube walls, separation happens and due to that separation, a decrease of the pressure is produced.

- **Temperature field**

In regard with the pictures of the temperature fields, we can say without a shadow of a doubt that as the mass flow and velocity were increased, the transmission of heat also increased. Therefore, we can conclude that in heat exchanger as the flow is risen the heat transmission will also increase, due to the fact that the turbulences enhanced the spread of the heat.

- **Turbulence field**

In *Fig. 1.4, 2.4 and 3.4* we can observe that as the Re is increasing the high energy turbulence zones are more defined in specific regions and obviously, the amount of energy increases as the Re increase.

Stream Function Fields

In *Fig. 1.5, 2.5 and 3.5* is showed that the kinetic energy increases in the upper region of the control volume. We believe that this phenomenon is caused because of the higher temperature in this region.

Nusselt Number

The Nusselt numbers obtained in the simulations increases proportionally with the raise of Re as it is shown in the equations however, there is an error between the engineering formulas and the results obtained in Ansys' simulations due to the approximations done in the simulation; 2D, small control volume and other simplifications done in the simulations and the use engineering equations that are more appropriate for practical use than for research appliances. Therefore, the variation between both Nu numbers is understandable.

$$Re_1 = \frac{v \cdot d}{\mu} = \frac{5{,}266 \cdot 0{,}0097}{1{,}460 \ \cdot 10^{-5}} = \boxed{3.497{,}298}$$

$$m = u \cdot \rho \cdot 0{,}5 \cdot S1 =$$

$$Nu_1 = 0{,}36 \cdot Re^{0{,}6} \cdot Pr^{0{,}33} \cdot Cs = \boxed{43{,}096}$$

$$Nu_{Ansys\ 1} = \boxed{49{,}56}$$

$$Cs = \left(\frac{\sigma_1 - 1}{\sigma_2 - 1}\right)^{0{,}1} = 1{,}006$$

$$\sigma_1 = \frac{S_1}{d} = 2{,}557$$

$$\sigma_2 = \frac{S_2}{d} = 2{,}093$$

$$\sigma'_2 = \sqrt{\frac{{\sigma_1}^2}{4} + {\sigma_1}^2} = 2{,}452$$

$$Re_2 = \frac{v \cdot d}{\mu} = \frac{16{,}45 \cdot 0{,}0097}{1{,}460 \ \cdot 10^{-5}} = \boxed{10.929}$$

$$Nu_2 = 0{,}36 \cdot Re^{0{,}6} \cdot Pr^{0{,}33} \cdot Cs = \boxed{85{,}37}$$

$$Nu_{Ansys\ 2} = \boxed{99{,}69}$$

$$Re_3 = \frac{v \cdot d}{\mu} = \frac{42{,}79 \cdot 0{,}0097}{1{,}460 \ \cdot 10^{-5}} = \boxed{28.415{,}54}$$

$$Nu_3 = 0{,}36 \cdot Re^{0{,}6} \cdot Pr^{0{,}33} \cdot Cs = \boxed{151{,}472}$$

$$Nu_{Ansys\ 3} = \boxed{204{,}089}$$

flow m	velocity	Re	Nu cal	Nufluent	dNu	α	αfluent
0,08	5,266623	3497,298	43,09614	49,56412	13%	107,5182	123,6548
0,25	16,4582	10929,06	85,37848	99,69	14%	213,0061	248,7111
0,65	42,79131	28415,54	151,472	204,0891	26%	377,8993	509,1707