ИЗЫСКАНИЕ, ПРОЕКТИРОВАНИЕ, СТРОИТЕЛЬСТВО И МОНТАЖ ТЕХНОЛОГИЧЕСКОГО ОБОРУДОВАНИЯ ОБЪЕКТОВ АТОМНОЙ ОТРАСЛИ

УДК 621.039.5

АНАЛИЗ ПОЛНОСТЬЮ РАЗРАБОТАННОГО ЛАМИНАРНОГО ПОТОКА МЕЖДУ ПАРАЛЛЕЛЬНЫМИ УВЧ ПЛАСТИНАМИ С ИСПОЛЬЗОВАНИЕМ ПРОГРАММЫ ГИДРОДИНАМИЧЕСКОГО МОДЕЛИРОВАНИЯ ANSYS

© 2018 К.Дж. Одии

Томский национальный исследовательский политехнический университет, Томск, Россия

Ламинарный поток между двумя параллельно расположенными твердыми пластинами с постоянным тепловым потоком на краях пластин является идеальным способом моделирования потока хладагента между параллельными топливными пластинами, обычно используемыми в исследовательских реакторах. В этой работе мы упростили модель, используя условия границ. Мы использовали уравнения переноса для генерации распределения скорости и температуры, градиента давления и других профилей контура. Затем мы использовали Программный модуль ANSYS FLUENT для создания профилей контура, результаты обоих методов сравнивались, и было отмечено, что распределение скоростей было параболическим, что соответствовало аналитическому результату, который предсказывал, что скорость полного ламинарного поток является параболической. Было также отмечено, что потеря давления и повышение температуры между впускным и выходным потоками очень малы.

Ключевые слова: ламинарный поток, форма, тепло, флюс, скорость, давление, температура, ANSYS, гидродинамическое моделирование, Нуссельт, число.

Поступила в редакцию: 14.08.2018

1.1 Introduction

In the model, we considered the flow between two parallel plates separated by a distance $2H$ with a uniform heat flux imposed on both plates [1]. The fluid is driven between the plates by an applied pressure gradient in the x-direction. Assume the fluid is laminar and fully developed:

$$\frac{du}{dx} = 0, v = 0 \; and \; w = 0$$

Then we are to determine the fully developed velocity distribution and the fully developed temperature distribution, using ANSYS FLUENT CFD Simulation.

1.1.1 Input Parameters:

Inlet velocity $= 0.005 m/s$
Outlet Pressure $= 5.00 Pa$
Heat Flux $= 20.00 W/m^2$
Wall Thickness $= 0.05 m$
Inlet Temperature $= 300.00 K$

1.1.2 Basic Assumptions:

1. Steady flow
2. Constant properties
3. Newtonian fluid
4. Negligible radiation
5. Negligible gravity effects
6. Negligible viscous dissipation
7. Laminar flow
8. Fully developed
9. Negligible end effects
10. Conduction in y-direction much greater than that of x-direction

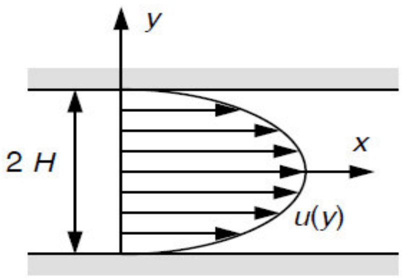

Fig. 1 – Laminar flow velocity distribution

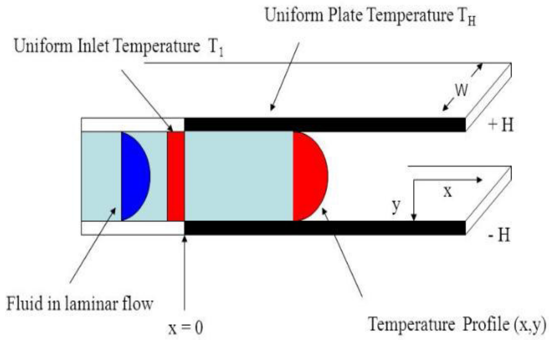

Fig. 2 – Diagram of Parallel plates Laminar flow

1.2 Analytical Model

1.2.1 Conservation of mass for constant property flow

$$\nabla \cdot V = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{1}$$

For fully developed flow, $v = w = 0$, therefore

$$\frac{\partial u}{\partial x} = 0 \tag{2}$$

Momentum equation for constant property flow of a Newtonian fluid:

$$\rho\left(\frac{\partial V}{\partial t} + V \cdot \nabla V\right) = \mu \nabla^2 V - \nabla p + \rho g \tag{3}$$

For this case the flow is independent of the z-direction, and the flow looks exactly the same at every position along the z-axis. Hence, all derivatives in the z-direction vanish [2].

x-component for 2-D steady flow in Cartesian coordinates:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) - \frac{\partial p}{\partial x} + F_x \tag{4}$$

$$\mu\left(\frac{\partial^2 u}{\partial y^2}\right) = \frac{\partial p}{\partial x} \tag{5}$$

y-momentum reduces to

$$\frac{\partial p}{\partial y} = 0$$

Integrate twice to get:

$$u(y)=\frac{1}{\mu}\frac{dp}{dx}\frac{y^2}{2}+A_1 y+A_2 \tag{6}$$

Impose boundary conditions: $u(-H) = 0\ and\ u(H) = 0$ (no slip condition at the wall)

$$u(-H)=\frac{H^2}{2\mu}\frac{dp}{dx}-HA_1+A_2=0 \tag{7}$$

$$u(H)=\frac{H^2}{2\mu}\frac{dp}{dx}+HA_1+A_2=0 \tag{8}$$

Adding these equations we get:

$$A_2=-\frac{H^2}{2\mu}\frac{dp}{dx} \text{ and } A_1=0$$

$$u(H)=-\frac{H^2}{2\mu}\frac{dp}{dx}\left(1-\left(\frac{y}{H}\right)^2\right) \tag{9}$$

Thus, the velocity profile is parabolic in nature.
Recalling the definition of mean velocity u_m and mass flow rate, $\dot{m}$:

$$\dot{m}=\rho u_m A=\int \rho u dA \tag{10}$$

$$u_m=\frac{1}{\rho A}\int \rho u dA \tag{11}$$

If the width of the flat plates perpendicular to the flow:

$$u_m=\frac{1}{\rho(2H)W}\int_0^W\int_{-H}^H \rho u dy dz=\frac{1}{H}\int_0^H\frac{H^2}{2\mu}\frac{dp}{dx}\left(\left(\frac{y}{H}\right)^2-1\right)dy \tag{12}$$

$$u_m=\frac{H^2}{2\mu}\frac{dp}{dx}\left(\frac{y^3}{3H^2}-y\right)_0^H=\frac{H^2}{2\mu}\frac{dp}{dx}\left(\frac{1}{3}-1\right)=-\frac{H^2}{3\mu}\frac{dp}{dx} \tag{13}$$

These observed trends of the pressure gradient can be intuitively interpreted as follows: [3]

$$\frac{dp}{dx}=\frac{3\mu}{H^2}u_m \tag{14}$$

Hence we can establish after substitution that the velocity distribution of the fluid flow is given by:

$$u(y)=\frac{3}{2}u_m\left(1-\left(\frac{y}{H}\right)^2\right) \tag{15}$$

It can be clearly seen that the velocity distribution of the flow is a parabolic curve and it is also a function of the mean velocity.

We will now decompose the Energy equation of fluid flow. We must say that the energy equation is needed for thermal analysis of the fluid [4]. The case where we do not need any thermal analysis, the energy equation is not used. This is also the case in our Simulation where we had to keep the energy equation on, to Simulate thermal properties.

1.2.2 Energy Equation for constant property flow of a Newtonian fluid:

$$\rho c_p\left(\frac{\partial T}{\partial t}+V\cdot\nabla T\right)=k\nabla^2 T+\mu\phi+\dot{q} \tag{16}$$

For 2-D steady flow in Cartesian coordinates:

$$\rho c_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)+\mu\phi+\dot{q} \tag{17}$$

$$\frac{u}{\alpha}\frac{\partial T}{\partial x}=\frac{\partial^2 T}{\partial y^2} \tag{18}$$

Where we assume $\frac{\partial^2 \mathrm{T}}{\partial \mathrm{y}^2} \succ\succ \frac{\partial^2 \mathrm{T}}{\partial \mathrm{x}^2}$

The temperature gradient along the x-axis determines the temperature distribution along the y-axis [4]

Substitute (15) into (18)

$$\frac{\partial^2 T}{\partial y^2}=\frac{3}{2}\frac{u_m}{\alpha}\frac{dT}{dx}\left(1-\left(\frac{y}{H}\right)^2\right) \tag{19}$$

Boundary conditions for temperature can be set in any part of the calculation domain boundary to one of the following types: constant value of temperature, constant value of heat flux, heat transfer with constant value of ambient temperature and linear profile of temperature [5]:

For constant surface heat flux: $\frac{dT}{dx}=\frac{dT_m}{dx}=const$

Integrating (19) twice we obtain:

$$T(y)=\frac{3}{2}\frac{u_m}{\alpha}\frac{dT_m}{dx}\left(\frac{y^2}{2}-\frac{y^4}{12H^2}\right)+A_1 y+A_2 \tag{20}$$

Imposing boundary conditions $T(-H)=T(+H)=T_s$ (note that T_s is unknown):

$$T(H)=\frac{3}{2}\frac{u_m}{\alpha}\frac{dT_m}{dx}\left(\frac{H^2}{2}-\frac{H^4}{12H^2}\right)+A_1 H+A_2=T_s \tag{21}$$

$$T(-H)=\frac{3}{2}\frac{u_m}{\alpha}\frac{dT_m}{dx}\left(\frac{H^2}{2}-\frac{H^4}{12H^2}\right)-A_1 H+A_2=T_s \tag{22}$$

Which transforms to:

$$A_1=T_s-\frac{3H^2}{2}\frac{u_m}{\alpha}\frac{dT_m}{dx}\left(\frac{5}{12}\right) \text{ and } A_2=0$$

$$T_s-T(y)=\frac{3H^2}{2}\frac{u_m}{\alpha}\frac{dT_m}{dx}\left(\frac{5}{12}-\frac{1}{2}\left(\frac{y}{H}\right)^2-\frac{1}{12}\left(\frac{y}{H}\right)^4\right) \tag{23}$$

But we know that the mean temperature is given as:

$$T_m=\frac{1}{\rho u_m A c_v}\int \rho u c_v T dA=\frac{1}{2u_m H}\int_{-H}^{H} uTdy \tag{24}$$

$$T_m=\frac{1}{u_m H}\int_0^H\left[\frac{3}{2}u_m\left(1-\left(\frac{y}{H}\right)^2\right)\right]\left[T_s-\frac{3H^2}{2}\frac{u_m}{\alpha}\frac{dT_m}{dx}\left(\frac{1}{12}\left(\frac{y}{H}\right)^4-\frac{1}{2}\left(\frac{y}{H}\right)^2+\frac{5}{12}\right)\right]dy \tag{25}$$

After transformation:

$$T_s-T_m=\frac{17H^2}{35}\frac{u_m}{\alpha}\frac{dT_m}{dx} \tag{26}$$

Dividing (23) by (26), we have:

$$\frac{T_s - T(y)}{T_s - T_m} = \frac{35}{136}\left[5 - 6\left(\frac{y}{H}\right)^2 + \left(\frac{y}{H}\right)^4\right] \tag{27}$$

From (27), we can see that the temperature distribution $T(y)$ is a function of the mean and surface temperatures. Limited comparison is made with data for flow in channels [6], this is because we are dealing with laminar flow which is a low Reynolds number flow.

Furthermore, we can theoretically calculate the Nusselt Number:

From heat transfer coefficient h:

$$h = \frac{k_f}{T_s - T_m}\left(\frac{\partial T}{\partial y}\right)_{y=H} \tag{28}$$

$$N_u = \frac{hD_h}{k_f} = \frac{D_h}{T_s - T_m}\left(\frac{\partial T}{\partial y}\right)_{y=H} \tag{29}$$

Due to the small hydraulic diameter used in micro-channel analysis [7], the hydraulic diameter in this case is just a measure of the distance between the plates.

Where

$$D_h = \frac{4A_s}{P} = \frac{4(2H)W}{2W} = 4H \tag{30}$$

$$N_u = \frac{4H}{T_s - T_m}\frac{\partial}{\partial y}\left(T_s - \frac{35}{136}(T_s - T_m)\left(\left(\frac{y}{H}\right)^4 - 6\left(\frac{y}{H}\right)^2 + 5\right)\right)_{y=H} \tag{31}$$

$$N_u = 4H\left(-\frac{35}{136}\right)\left(-\frac{8}{H}\right) = \frac{140}{17} = 8.2353 \tag{32}$$

D_h is called the hydraulic diameter, where w and H had been defined in the initial diagram.

After substituting the value of T in (27) into (29), the Nusselt number is calculated to be 8.2353.

1.3 Simulation results

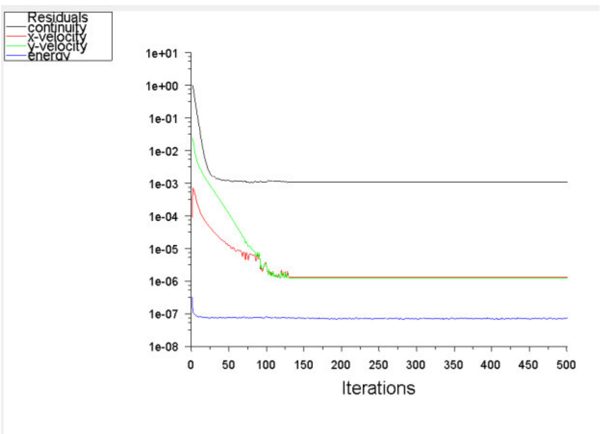

Fig. 3 – Iteration graph

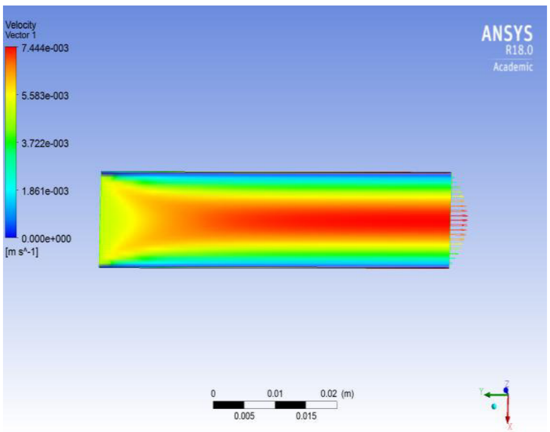

Fig. 4 – Velocity vector contour

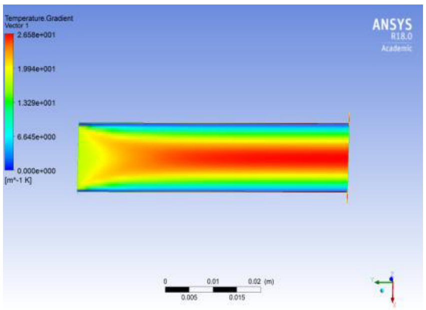

Fig. 5 – Temperature Gradient contour

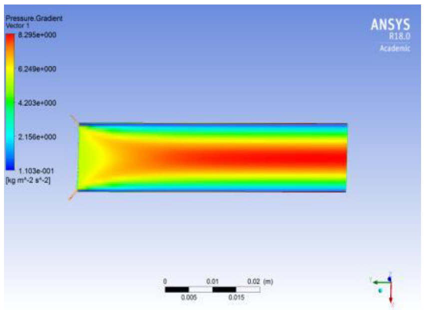

Fig. 6 – Pressure Gradient contour

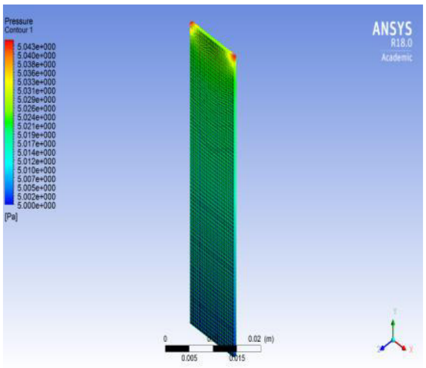

Fig.7 – Pressure contour

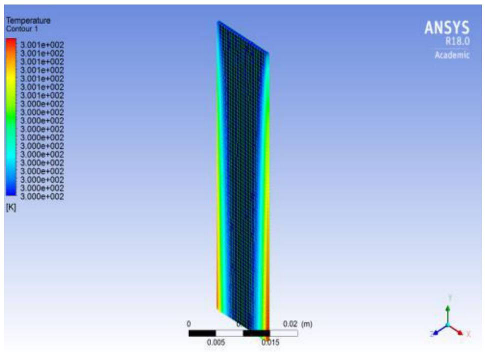

Fig.8 – Temperature contour

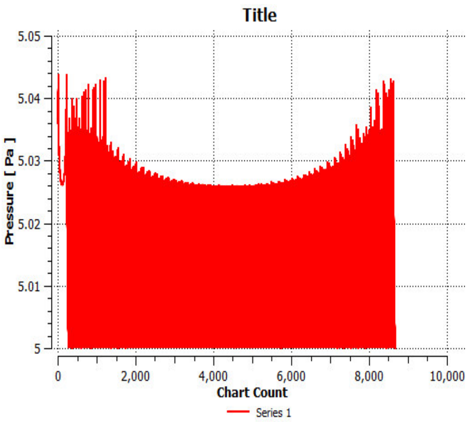

Fig. 9 – Pressure Distribution

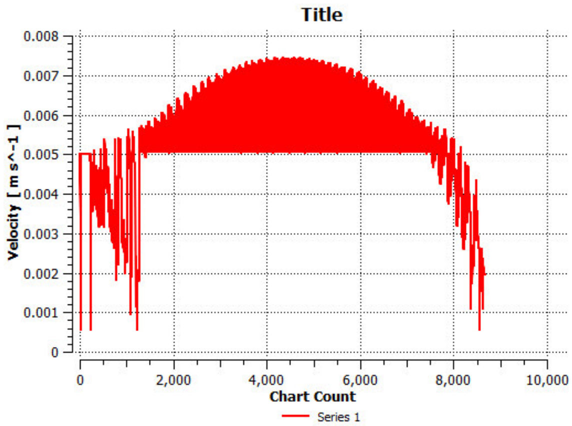

Fig. 10 – Velocity distribution

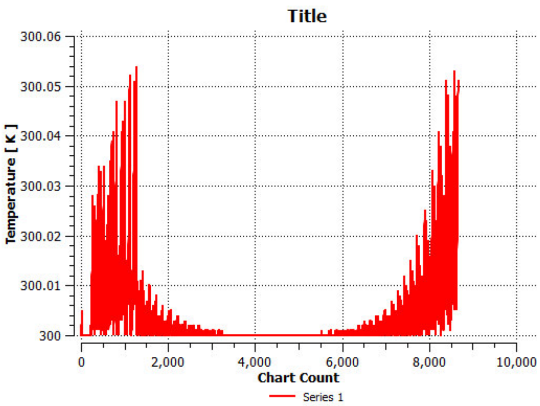

Fig. 11 – Temperature Distribution

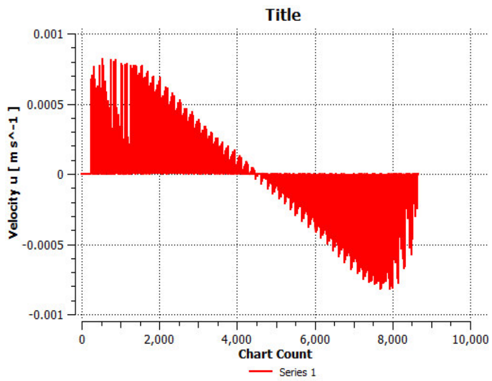

Fig. 12 – Velocity (u) Distribution

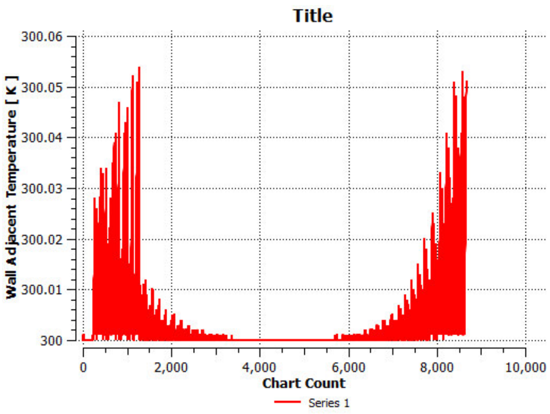

Fig. 13 – Wall adjacent Temperature Distribution

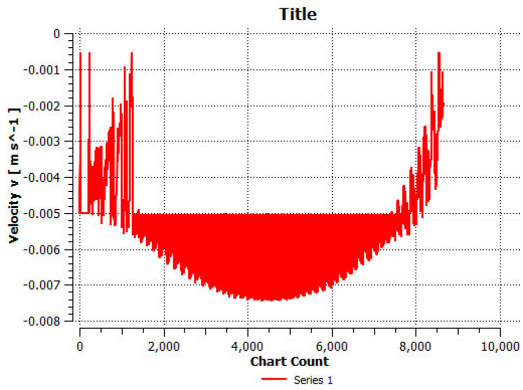

Fig. 14 – Velocity (v) Distribution

1.4 Analysis of Results

The following numerical results were obtained from the functional calculator of the ANSYS FLUENT CFD after Simulation.

Table 1 – Numerical

Parameters/values	Minimum	maximum	Average
Velocity (m/s)	0.000750	0.007500	0.005000
Temperature (K)	300.0000	300.0690	300.0180
Temperature gradient (K/m)	0.000000	26.58200	11.53400
Pressure gradient (Pa/m)	0.110000	8.295000	0.237000

From the table above, the minimum velocity is less than the inlet velocity, and this velocity was observed at the outlet.

The results of the table above can be directly interpreted from the various distribution contours in the Simulation results.

Thermal properties of water at 27℃ are as follows:

$$\rho = 996.5kg/m^3$$
$$k = 0.612W/mK$$
$$\mu = 0.00088kg/ms$$
$$Pr = 5.72$$

Dimension of the Flat plate; Length = 60mm, Width = 14mm

To compare the analytical Nusselt number with the simulation Nusselt number, and hence to calculate the convective heat transfer coefficient and the heat flux and compare with the analytical heat flux.

The uniform surface heat flux of laminar flow in a parallel flat plate, we have the correlation formular for determining the Prandtl number as:

$$N_{u_x} = 0.453 \mathrm{Re}_x^{1/2} \mathrm{Pr}^{1/3}$$

$$\mathrm{Re}_x = \frac{\rho u_m D_h}{\mu} = 113.23, \quad \mathrm{Re}_x^{1/2} = 10.64$$

$$N_{u_x} = 0.453 \mathrm{Re}_x^{1/2} \mathrm{Pr}^{1/3} = 8.6$$

Where $D_h = 20mm$

Also using the Churchill and Ozoe Correlation formular for laminar flow in parallel flat plate, we have:

$$N_{u_x} = \frac{0.886 \mathrm{Re}_x^{1/2} \mathrm{Pr}^{1/2}}{\left(1+\left(\frac{\mathrm{Pr}}{0.0207}\right)^{2/3}\right)^{1/4}} = 8.7$$

The heat transfer coefficient $$h_x = \frac{N_{u_x} k}{x} = 376.38 W/m^2 \,^\circ C$$

Therefore the numerical Heat flux is given thus:

$$q'' = h_x \Delta T = 376.38(0.055) = 20.70 W/m^2$$

Where ΔT was read directly from fig-11 (Temperature against chart count)

Since the heat flux is known, the heat transfer coefficient can be used to determine the local surface temperature.

$$T_s(x) = T_{in} + \frac{q''}{h_x} = 300 + \frac{20.5}{376.38} = 300.0559\ K$$

Calculations of temperature distributions, average temperatures, and heat transfer coefficients are presented for a rectangular geometry [8]. The maximum velocity can be calculated from the velocity distribution formula, and the result compared to the one on the table above. Since we already know the mean velocity (average velocity), we also know that in this case, the velocity depends on the transverse coordinate y [9]. We can easily check the maximum velocity, which we know occurs at the central axis of the flow where $y = 0$. Using equation (15):

$$u(y) = \frac{3}{2} u_m \left(1 - \left(\frac{y}{H}\right)^2\right)$$

For $y = 0$, we have

$$u(0) = u_{\max} = \frac{3}{2} u_m = \frac{3}{2}(0.005) = 0.0075 m/s$$

This is in good agreement with the CFD Numerical result in the table above.

The inlet pressure was also observed to be 5.05Pa from the Simulation contour.

We can conclude that the Simulation distribution contours clearly described the temperature and velocity distribution formulae modeled in this research. This can be verified by comparing the distribution contours and the distribution formulae.

1.5 Conclusion

The Simulation result of Laminar flow between two parallel plates separated by a distance of 2H, often referred to as Poiseuille flow [10], agreed with the analytical functional equations of Laminar flow between two parallel plates. The Simulated results produced distribution contours that describe the temperature and velocity distribution function of the standard Mass and Energy equations of Modeled Laminar Flow. Though there was slight difference between the analytical Nusselt number and the Numerical Nusselt number, the difference can be ascribed to error due to geometrical calculation in the Reynolds number, in particular the Hydraulic diameter (D_h). The slight temperature change in the flow is as a result of molecular interaction between water molecules during the flow, this is the reason why the outlet temperature is almost the same with the outlet temperature. The study does not consider heat source. In the case of heat source, the temperature difference would be higher.

REFERENCES

[1] ME 350, Heat Transfer, Prof. Shollenberger lectures Mechanical Engineering Department California Polytechnic State University. 2017. URL: www.calpoly.edu/~kshollen/ ME350/Examples/Example_14
[2] MIT Course Notes Spring 2009. URL: mit.edu/snively/www/2_006%20Coursenotes.
[3] Bonfanti F. et al (1979). Two Phase Pressure Drops in the Low Flow Rate Region. Energia Nucleare. 26 (10). pp 481-492.
[4] Alina Filip et al. Comparison of Two Phase Pressure Drop Models for Condensing Flows in Horizontal Tubes. Mathematical Modeling in Civil Engineering Vol 10. N 4. 2014.
[5] Jiri Brada. Mathematical Model of Two Phase Flows. WM, 01 Conference. Tucson. AZ. 2001.
[6] Chisholm D. (1973). Pressure Gradients Due to Friction during the Flow Evaporating Two-Phase Mixtures in Smooth Tubes and Channels. International Journal of Heat and Mass Transfer. 16(2). pp 347-358.
[7] Crowe C.T, (2006), Multiphase Flow Handbook, CRC: Taylor and Francis. Boca Raton. FL.
[8] Duda J.L and Vrentas J.S (1971). Heat Transfer in a Cylindrical Cavity. Journal of Fluid Mechanics. 45. pp 261-279.
[9] Satish Kandlikar et Al (2007). Heat transfer and fluid Flow in Minichannels and Mcrochannels. Elsevier publishers Oxford. UK. 576 p.
[10] Brian D. Storey (2015). Fluid dynamics and Heat Transfer. An Introduction to the Fundamentals. Olin College. 286 p.

Analysis of Fully Developed Laminar Flow between Parallel Plates with UHF Using ANSYS CFD

Odii Ch. J.

National Research Tomsk Polytechnic University, Lenin St., 30, Tomsk, Russia, 634050
e-mail: monoexiton@yahoo.com

Abstract – Laminar flow between two parallel placed solid plates with constant heat flux at the edges of the plates, is an idealize way of modelling the flow of coolant between parallel fuel plates commonly used in research Reactors. In this paper, we attempted to validate both the thermal and fluid properties of a fully developed laminar flow with uniform heat flux using ANSYS FLUENT. We used the transport equations to generate the velocity and temperature distribution, the pressure gradient and other profile contours. Then we used ANSYS FLUENT Simulation to generate profile contours, the results of both methods were compared and it was observed that the velocity distribution was parabolic from the Simulation, this was in agreement with the analytical result which predicted that the velocity of the fully developed Laminar flow is parabolic. Also the presure loss and the temperature rise between the inlet and outlet flow were observed to be very small.

Keywords: Laminar, uniform, heat, flux, velocity, pressure, temperature,ANSYS, CFD, Nusselts, number