Gas flow and heat transfer CFD modeling in microchannels

Abdelhamid BLIDIA[1*], Jean KOVCHAR[2], François LANZETTA[1], Magali BARTHES[2], Michel DE LABACHELERIE[2]

[1]FEMTO-ST, FEMTO-ST Institute, Univ. Bourgogne Franche-Comté, CNRS, ENERGY Department, Parc technologique, 2 avenue Jean Moulin, 90000 Belfort.
[2]FEMTO-ST, FEMTO-ST Institute, Univ. Bourgogne Franche-Comté, CNRS, MN2S Departement, 15B avenue des Montboucons, 25030 Besançon.
[*](Corresponding author: abdelhamid.blidia@femto-st.fr)

Abstract – In this study, a CFD analysis was performed to investigate the heat transfer and fluid friction characteristics of a micro regenerator in order to be integrated in a micro Stirling engine. The simulations were conducted using the commercial software ANSYS-FLUENT. The gas flow is set to be incompressible and viscous under laminar unidirectional steady flow conditions for low Reynolds number (Re < 100). The results are validated by comparing the friction factor and the heat transfer coefficient predicted by simulation with available experimental results.

Keywords: Microfluidics; Heat transfer; Pressure drop; Numerical Simulations

Nomenclature

C_f	Darcy friction factor
C_p	specific heat capacity, J. kg^{-1}. K^{-1}
D_h	hydraulic diameter, m
ΔP	pressure drop, bar
h	heat transfer coefficient, W.m^{-2}.K^{-1}
Kn	Knudsen number
l	length, m
Ma	Mach number
Nu	Nusselt number
P	pressure, bar
Pe	Peclet number
Pr	Prandtl number
Re	Reynolds number
T	temperature, K
u	velocity, m.s^{-1}
Greek symbols	
α	thermal diffusivity, m^2.s^{-1}
μ	dynamic viscosity, kg.m^{-1}.s^{-1}
ρ	bulk density, kg.m^{-3}
λ	thermal conductivity, m^2.s^{-1}
ν	kinematic viscosity, m^2.s^{-1}
Indices and exhibitor	
f	fluid
m, n	vector component
w	wall

1. Introduction

With the intensifying of global energy crisis and environmental deterioration, the development of renewable energy technologies becomes necessary [1]. Among various technical solutions that are being developed, Stirling engines have attracted widespread attention in recent years since they have wide energy adaptability as external combustion engines, and have a series of advantages including low noise, high theoretical efficiency, low environmental pollution, and simple structures [2]. However, the Stirling engines performances are relatively lower than those of internal combustion engines due the numerous irreversible losses, which has limited their further development and application. Therefore, the design and optimization of Stirling engines with high efficiency are of great value for facilitating the application of Stirling engines, and thus promoting the utilization of renewable energy and alleviating energy and environmental problems [3]. Researchers tried various ways to improve

https://doi.org/10.25855/SFT2022-014

Stirling engines and proposed several design methods, while these methods always need correlation equations to predict the performances of Stirling engines [4]. One of the concepts that is currently under study is the miniaturization of Stirling Engine which has been considered since the development of Micro Electro-Mechanical Systems (MEMS) [5]. Some components of a Miniaturized Stirling engine such as Hybrid Fluid–Membranes [6] and Micro-Regenerators [7] have already been developed, yet the study of the flow through Micro-Regenerators is still unclear and need further investigations.

In the literature, a vast amount of results can be found on microscale gas flow, although there are inconsistencies in the experimental data reported by the different researchers. Several factors can be attributed to this inconsistency: rarefaction (the slip on the surface), surface roughness, entrance effect, minor losses and compressibility.

Peiyi and Little [8] conducted one of the preliminary investigations of fluid flow characteristics in microchannels. The study was conducted using nitrogen, hydrogen and argon as working fluids, with eight different microtubes having diameters ranging from 55.81 to 83.08 µm. The conclusion of this study was that the effect of surface roughness still affects the value of the friction factor even under laminar flow conditions. The friction factor for bends (angle 90° and 135°) were also found with respect to Reynolds number.

Asako *et al.* [9] [10] investigated numerically pressure drop characteristics for two-dimensional compressible flow in both parallel plates and microtubes. Air and nitrogen were used as working fluid, and the hydraulic diameter was ranging between 10 and 100 µm. The computations were performed for a low range of Reynolds number (Re < 500) and Mach number ($Ma < 0.4$), and for both no heat conduction and isothermal flow conditions. The friction factor for both microtubes and parallel plates was correlated in terms of Mach number.

Turner *et al.* [11] performed experiments to measure the friction factor of a laminar gas flow through microchannels, having a hydraulic diameter ranging from 5 to 96 µm, with helium and nitrogen as working fluids. It was found that the friction factor was in close agreement, in the limiting case of low Ma and low Kn, with the incompressible continuum flow theory.

Kohl *et al.* [12] conducted experiments to investigate pressure drop in straight microchannels. The channels were developed with integrated pressure sensors, and with hydraulic diameters ranging from 25 to 100 µm. the result of the study showed that the friction factor can be accurately determined by a standard numerical simulation that considers compressibility and entrance effects.

Yang *et al.* [13] investigated experimentally pressure drop and heat transfer characteristics of air flow through microtubes with inside diameter of 86 and 920 µm. The experimental results showed that the conventional heat transfer correlation for laminar and turbulent flow can be well applied for predicting the fully developed gaseous flow heat transfer performance in microtubes.

Kim [14] performed an empirical study to explore the validity of theoretical correlations based on conventional sized channels for predicting heat transfer characteristics in microchannels. The flow resistance and thermal behavior of laminar flow through 10 different rectangular microchannels with hydraulic diameters of 155–580 µm and aspect ratios of 0.25–3.8 at Reynolds numbers ranging from 30 to 2500 were investigated. The results showed that Nusselt number began to exceed the theoretical values for $Re > 180$.

These studies indicated that scaling effects have important impacts on heat transfer and fluid flow characteristics. While there is no widely recognized conclusion, especially in correlations for heat transfer coefficient and friction factor. Besides, most of the above experimental studies were carried out using a steady unidirectional flow, which is not the one commonly used in the Stirling engine, on the contrary of the unsteady oscillating flow. The initial purpose of the current investigation was to determine the validity of the standard continuum-based models for microchannel flows where $Kn < 0.01$ by comparing with the available data in the literature. Additionally, the proposed model was used to estimate heat transfer and fluid flow characteristics at low Reynolds number $Re < 100$ for different microchannels.

2. Geometry configuration and CFD modeling

2.1. Description of computational model

The computational model used for the analysis of heat transfer and fluid flow characteristics in microchannels is based on the following assumptions:

- steady state flow,
- laminar flow,
- incompressible flow,
- single phase flow,
- radiation heat transfer is neglected.

The computational domain consisted of a straight main channel with a square-shaped cross-section placed between a contraction and expansion regions at the inlet and the outlet of the microchannel. Figure 1 shows a schematic of the computational domain. The hydraulic diameter D_h and the length l of the duct varied from 100 µm to 1 mm and from 10 mm to 50 mm, respectively. The working fluids used for the numerical calculation are: Air, Helium, Hydrogen and Nitrogen.

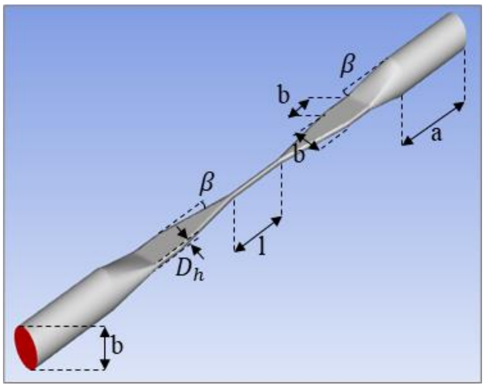


Figure 1: *Computational fluid-dynamics domain (a=20 mm, b=5 mm, β=10°)*

2.2. CFD setup and Boundary Conditions

The numerical investigations are performed by using the commercial software ANSYS. ANSYS Design Modeler is used to prepare the fluid domain for the numerical analysis. The grid is generated with ANSYS MESH, and then the meshed model is exported to ANSYS FLUENT. The element generated for this simulation is tetrahedral/hybrids mesh with non-uniform distribution and boundary layer mesh at the wall. The pressure-based type and absolute velocity formulation is chosen as a solver in the general setup. The convergence limit is taken of 10^{-6} for relative deviation of continuity, velocity components and energy to ensure sufficient accuracy for numerical results. Isothermal and adiabatic conditions are applied at the wall to study both heat transfer and fluid flow characteristics, respectively. A uniform velocity is imposed at the inlet and a fixed pressure of 1.01325 bar is applied at the outlet. The temperature inside the microchannel is taken as 300 K, at the beginning.

2.3. Governing equations

The steady incompressible flow in microchannels is governed by the continuity, momentum, and energy equations. With assumptions of steady state and viscous gas flow, the governing equations for the CFD analysis can be described as follows [15].

Continuity equation:

$$\frac{\partial u_n}{\partial x_n} = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial x_n}\left(\rho_f u_m u_n\right) = -\frac{\partial p}{\partial x_m} + \frac{\partial}{\partial x_n}\mu_f\left(\frac{\partial u_m}{\partial x_n} + \frac{\partial u_n}{\partial x_m}\right) \tag{2}$$

Energy equation:

$$\rho_f C_{p_f} u_n \frac{\partial T}{\partial x_n} = \frac{\partial}{\partial x_n}\left(k_f \frac{\partial T}{\partial x_n}\right) \tag{3}$$

The physical properties of the fluids are supposed to be constant (Table 1) except the density for which the ideal gas approximation is considered.

	Specific heat capacity C_p ($J.kg^{-1}.K^{-1}$)	*Dynamic viscosity* μ (10^{-5} $kg.m^{-1}.s^{-1}$)	*Thermal conductivity* λ ($W.m^{-1}.K^{-1}$)
Air	1006.5	1.789	0.024
He	5193.0	1.990	0.152
H_2	14283.0	0.841	0.167
N_2	1139.7	1.802	0.026

Table 1: *Thermophysical properties of working gases*

3. NUMERICAL RESULTS AND DISCUSSION

3.1. Comparative analysis and Grid independence of the solution

In order to ensure the relevance of the proposed model, a comparative analysis is performed between the numerical results and experimental data obtained in the literature. Three geometries are investigated: parallel plates, microtube and microchannel with rectangular cross-section. Figure 2 and 3 illustrate the variation of the friction factor in terms of Reynolds number and Mach number for the different geometries. The results show a good agreement with the correlations obtained in the literature (the average relative difference is less than 5%). Therefore, the proposed model can be well applied for predicting the fluid flow and heat transfer in microchannels.

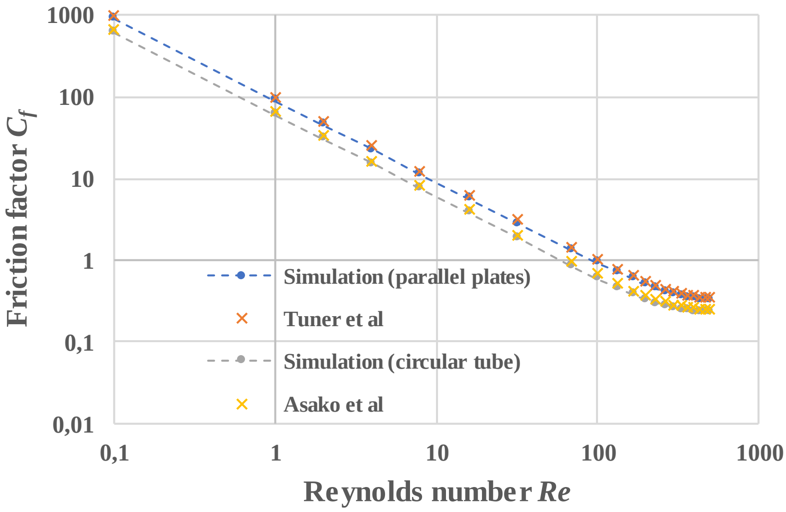


Figure 2: *Variation of the friction factor C_f in terms of Reynolds number Re*

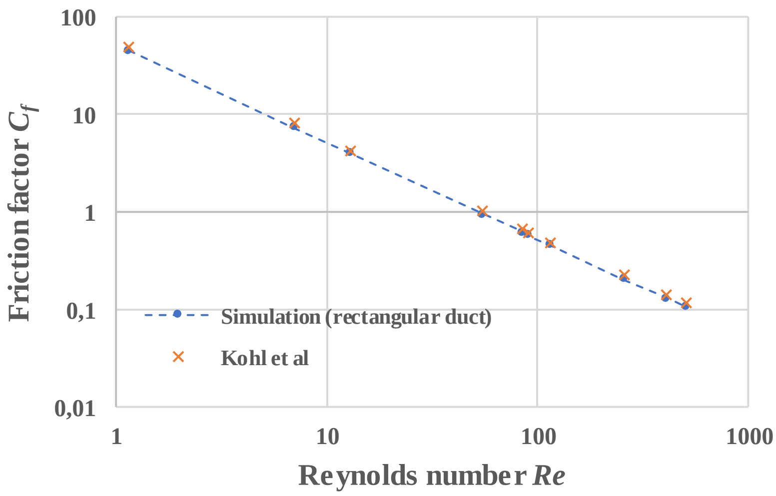


Figure 3: *Variation of the friction factor C_f in terms of Reynolds number Re*

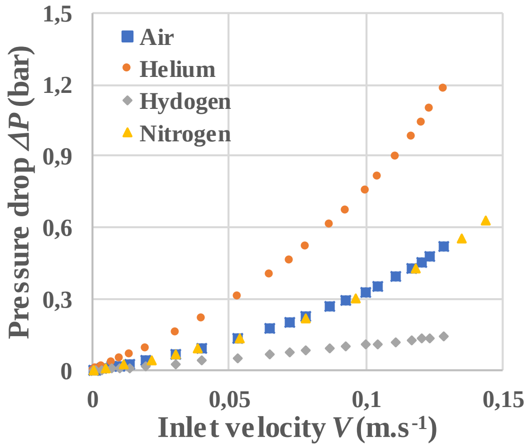
In order to consider appropriate grid size for the present study, a grid independent test is performed with different sizes of meshes and Reynolds number of 10. First, the initial mesh is generated with 25000 elements. Subsequently, the mesh is refined until the relative difference between the two successive results reached below 5%. The variation of the average velocity of the gas in the microchannel is chosen as criterion to determine the adequate mesh size. It has been concluded from this study that a mesh with 1.6 million elements allows to obtain mesh-independent results for cases under investigation.

Mesh	Element size (mm)	Number of layers at the wall
1	0.1	10
2	0.075	13
3	0.05	15
4	0.025	17
5	0.01	20

Table 2: *Element size and number of layers at the wall for the different sizes of mesh*

3.2. Fluid flow

Figures 4 and 5 show the pressure drop along the microchannel in terms of inlet velocity for different working fluids and the friction factor in terms of Reynolds number *Re* for different hydraulic diameters D_h, respectively.



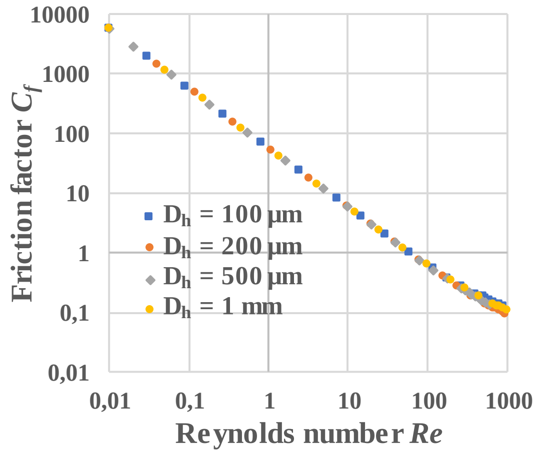


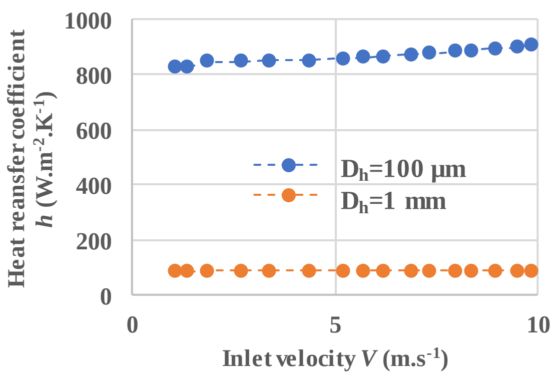
Figure 4: *Pressure drop versus inlet velocity for different working fluids*

Figure 5: *Friction factor in terms of Reynolds number for different* D_h

According to the results obtained, it clearly appears that helium has the highest pressure drop among the four gases since it is the most viscous gas.

3.3. Heat transfer

In this section, the heat transfer characteristics in the microchannel are presented as an average value of the heat transfer coefficient and the Nusselt number *Nu* calculated using the log-mean temperature difference. It should be noted that a constant temperature condition was applied at the wall boundaries.



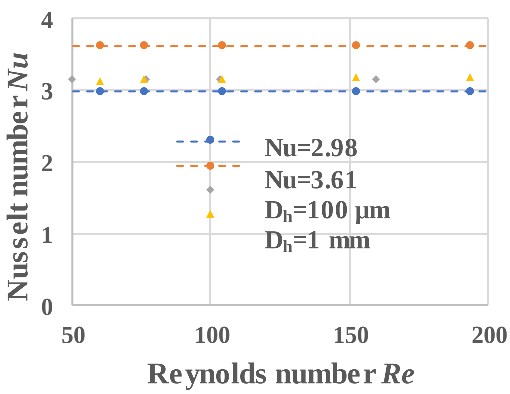


Figure 6: *Heat transfer coefficient versus inlet velocity V for different working fluids*

Figure 7: *Nusselt number in terms of Reynolds number for different* D_h

Figure 6 shows the convective heat transfer coefficients with constant wall temperature at various inlet velocity for microchannels with 100 µm and 1 mm of D_h. According to the results, the maximum heat transfer coefficient *h* obtained is about 900 $W.m^{-2}.K^{-1}$ at the wall temperature of 90°C and a hydraulic diameter of 100 µm as illustrated in Figure 6.

Figure 7 shows the Nusselt number in terms of Reynolds number, compared with theorical values with a constant wall temperature ($Nu = 2.98$) and a constant heat flux ($Nu = 3.61$). From the results, it can be observed that the Nusselt number is relatively in good agreement with the values predicted by the correlations for the flow in macro channels.

4. Conclusion

In this paper, a numerical study was investigated aimed at determining the fluid friction and heat transfer characteristics in square cross-cross section microchannels at different inlet conditions. ANSYS-FLUENT software was used to carry out the computational fluid dynamics modelling. The numerical model was first compared with some previous results obtained in the literature in order to improve the accuracy of the numerical simulations before being used in the current study.

By comparing the results, it can be concluded that there are many factors which affect the value of the friction factor in small channels. For low Reynolds number ($Re < 100$), it was found that there is no significant variation of the friction factor for the different channels simulated but as the Reynolds number increases it appears a certain dispersion between the values. For a given channel, it was observed that the friction factor increases beyond the analytic prediction for laminar flow as the Reynolds number increases.

As for the heat transfer, it was found that the conventional correlation for laminar flow can be well applied for predicting the fully developed air flow heat transfer in microchannel. It was also observed that the total temperature is higher than the wall temperature this is caused by the additional heat transfer between the gas and the wall since gas static temperature decrease because of conversion of thermal energy into kinetic energy.

References

[1] E. Santoyo, R. Best and M. L. de Haro, The third International Symposium on Renewable Energy and Sustainability (ISRES 2015), Elsevier, 2017.

[2] M. Yu, F. Xin, X. Lai, H. Xiao, Z. Liu and W. Liu, Study of oscillating flows through a novel constructal bifurcation Stirling regenerator, *Applied Thermal Eng.*, 184, (2021), 116413

[3] B. Kongtragool and S. Wongwises, A review of solar-powered Stirling engines and low temperature differential Stirling engines, *Renewable and Sustainable Energy Reviews*, 7, (2003), 131-154.

[4] G. Xiao, H. Peng, H. Fan, U. Sultan and M. Ni, Characteristics of steady and oscillating flows through regenerator, *Int. J. of Heat and Mass Transfer*, 108, (2017), 309-321.

[5] L.-L. Lemaire, Miniaturized Stirling engines for waste heat recovery, McGill University (Canada), (2012).

[6] R. Chutani, F. Formosa, M. De Labachelerie, A. Badel and F. Lanzetta, The design, fabrication and characterization of fluidic membranes for micro-engines with the aim of frequency lowering, *J. of Micromechanics and Microengineering*, 26, 12, (2016), 124009.

[7] E. Dellali, S. Bégot, F. Lanzetta, E. Gavignet and J.-Y. Rauch, Pressure drop analysis of oscillating flows through a miniature porous regenerator under isothermal and nonisothermal conditions, *Exper. Thermal and Fluid Science*, vol. 103, 2019.

[8] W. Peiyi and W. A. Little, Measurement of friction factors for the flow of gases in very fine channels used for microminiature Joule-Thomson refrigerators, *Cryogenics*, 23, 5, (1983), 273-277.

[9] Y. Asako, T. Pi, S. E. Turner and M. Faghri, Effect of compressibility on gaseous flows in micro-channels, *Int. J. of Heat and Mass Transfer*, 46, 16, (2003), 3041-3050.

[10] Y. Asako, K. Nakayama and T. Shinozuka, Effect of compressibility on gaseous flows in a micro-tube, *Int. J. of Heat and Mass Transfer*, 48, 23-24, (2005), 4985-4994

[11] S. E. Turner, L. C. Lam, M. Faghri and O. J. Gregory, Experimental investigation of gas flow in microchannels, *J. Heat Transfer*, 126, 5, (2004), 753-763.

[12] M. J. Kohl, S. I. Abdel-Khalik, S. M. Jeter and D. L. Sadowski, An experimental investigation of microchannel flow with internal pressure measurements, *Int. J. of Heat and Mass Transfer*, 48, 8, (2005), 1518-1533.

[13] C.-Y. Yang, C.-W. Chen, T.-Y. Lin and S. G. Kandlikar, Heat transfer and friction characteristics of air flow in microtubes, *Exp. Thermal and Fluid Sci.*, 37, (2012), 12-18.

[14] B. Kim, An experimental study on fully developed laminar flow and heat transfer in rectangular microchannels, *Int. J. of Heat and Fluid Flow*, 62, (2016), 224-232.

[15] I. ANSYS, «ANSYS fluent 12.0 User's Guide,» New Hampshire: ANSYS INC, 2009.

Acknowledgments

This work has been supported by the EIPHI Graduate School (contract ANR-17-EURE0002), and the Région Bourgogne-Franche-Comté.