24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2161
COMPUTATIONAL ANALYSIS OF VORTEX SHEDDING PAST A PIN FIN CONFINED IN A MICROCHANNEL

Philippe R. d'Egmont
Vinícius Z. Martins
Carolina P. N. Cotta
Fernando P. Duda
Renato M. Cotta
Laboratory of Nano & Microfluidics and Microsystems - LabMEMS
Mech. Eng. Dept. and Nanoeng. Dept., POLI & COPPE/UFRJ, Federal University of Rio de Janeiro, Brazil.

Manish K. Tiwari
Stavroula Balabani
University College London, UK.

Abstract. *The study of fluid flow over confined obstacles in a microchannel is relevant for a variety of microfluidics applications such as microreactors, micromixers, electronics cooling, etc. A numerical study based on the finite volume method has been conducted to investigate the flow behavior through a microchannel with a cylindrical micro-pin insert. Two different configurations were investigated; first, the symmetry condition was used on the channel's lateral walls, and second, the no-slip condition was used on the channel's walls, which accounts for lateral confinement effects. The vortex shedding behavior, as well as the confinement effect due to the channel's stationary walls on Strouhal number and drag coefficient, were analyzed and a good agreement with the literature was observed. As the lateral confinement increases, the Strouhal number goes up significantly. Furthermore, the increase of microchannel vertical wall confinement acts as a buffer, favoring the formation of two fixed and symmetrical vortices behind the cylinder and suppressing vortex shedding. Finally, a brief comparison with experimental results was performed, and the same vortex shedding behavior, was observed, offering validation of the proposed model and respective numerical simulation.*

Keywords: *Vortex shedding, Microchannel, Pin fin, Lateral/vertical confinement, Numerical simulation.*

1. INTRODUCTION

A large number of numerical and experimental studies have been conducted in recent years to investigate the flow around a confined cylinder, emphasizing the importance of this topic and the need to better understand the involved physical behavior towards different engineering applications, such as in the design of heat exchangers and microreactors for heat transfer and mixing enhancement, respectively. In this context, this work aims to numerically analyze the influence of confinement induced by both lateral and upper/lower walls, in order to understand the onset of vortex shedding at the microscale.

A few studies have been performed in order to investigate the vortex shedding phenomenon and heat transfer enhancement in micro-systems with a set of pins for two and three-dimensional cases. The numerical study carried out by Renfer *et al.* (2013a) was aimed at understanding the flow behavior and heat transfer through a microchannel populated with single and multiple pin arrangements. The numerical simulations were conducted on a representative portion of a 3D integrated chip stack with water cooling, applying 2D and 3D conjugated heat transfer models in order to analyze the vortex shedding and its effect on the heat transfer. In addition to the theoretical and numerical study, some experimental works have been undertaken in order to investigate the vortex shedding and thermal transport. Renfer *et al.* (2011) conducted an experimental work using Particle Image Velocimetry (μPIV) measurements to analyze the vortex shedding and flow transition in microchannels with cylindrical pins. Renfer *et al.* (2013b) analyzed the hydrodynamics in microchannels populated with cylindrical pins using dynamic pressure measurements. Microchannels with different height-to-diameter and pitch-to-diameter ratios were tested, confirming an increase in the Strouhal number with increased confinement.

Several other studies have been performed, originally formulated for the macro-scale, in order to investigate the flow dynamics itself and the vortex shedding over single cylinders in two and three-dimensional configurations. One of the first studies of numerical characterization of the flow over circular cylinders was carried out by Son and Harranty (1969). In this work, a finite difference formulation to solve the 2D time dependent equations of motion for the flow around a cylinder for Reynolds numbers up to 500 was proposed and the results were compared to experimental data, showing good agreement with numerical simulations. Chakraborty *et al.* (2004) numerically investigated the steady incompressible flow past a circular cylinder confined in a plane rectangular channel for uniform inlet velocity and different values of Reynolds number and width-to-diameter ratio. Two and three-dimensional numerical simulations were performed by Rajani *et al.* (2009) for different laminar flow regimen past a circular cylinder. The numerical results were validated against measurement data for mean surface pressure, force coefficients and also for Strouhal frequency of vortex shedding. Kanaris *et al.* (2011) also presented a two and three-dimensional numerical study of the flow around a circular cylinder placed symmetrically in a plane channel, for different values of Reynolds number up to 390 and width-to-diameter ratio of 0.2. The simulations were performed in order to investigate the confinement effect due to the walls on the force coefficients and the associated Strouhal number.

The confined flow behavior over obstacles within microchannels differs greatly from the unconfined flow, due to the influence of the channel's lateral and bottom/up walls. The lateral and the vertical confinement provided by the stationary no-slip walls of the channel affects the nature and the stability of the flow. Such flow configurations can be present in several industrially significant micro-scale flows, where they can be used to enhance heat transfer and mixing, for instance, in the context of common applications such as microreactors, micromixers, and electronics cooling. Two different configurations were analyzed here. First, symmetry conditions were used on the channel's lateral walls and a good agreement with the results in Renfer *et al.* (2013a) was observed, and second, the no-slip condition was used on the channel's walls which represents the confinement situation, also showing a good agreement with experimental results (*Zhang et al.*, 2017). The aim of the present work is thus to verify and validate a computational simulation based on the ANSYS CFX platform, which will allow the design and optimization of a novel pin fin microreactor for biodiesel synthesis (Costa Junior *et al.*, 2015; Pontes *et al.*, 2016).

2. COMPUTATIONAL PROCEDURE

The finite volume method is used for the discretization of the governing incompressible Navier-Stokes equations in the CFD simulations. The flow through the microchannel with cylindrical pins was simulated using the commercial platform ANSYS CFX 17.2. The predicted drag coefficients and Strouhal numbers were then investigated. Fig. 1 illustrates the implemented physical model, where d corresponds to the diameter of the pin, H represents the height of the channel, W is the width, and L is the length of the channel.

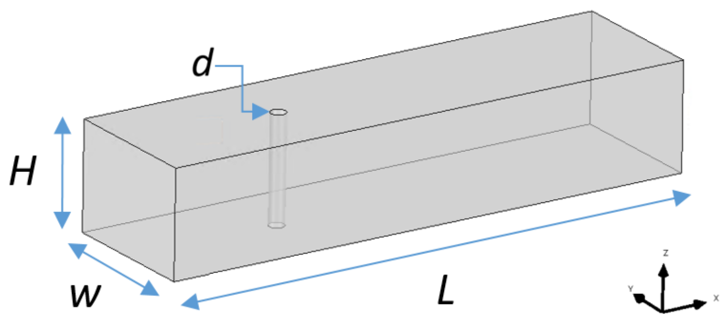

Figure 1. Schematic representation of the implemented physical model.

A convergence analysis was carried out for the drag coefficient computed through increasingly refined meshes, as shown in Tables 1 and 2, in order to attain a mesh size capable of providing precise results within moderate computing time. The geometry was finally meshed with 2,820,000 elements and 2,905,980 nodes, which corresponds to Mesh 3 in Tables 1 and 2 below.

Figure 2 shows a schematic view of the mesh density and orientation near the cylinder walls and along the microchannel.

Table 1. Convergence analysis for no-slip conditions case, with $w^* = 5$, $h^* = 1.6$ and Re=39.

Mesh	C_d	Number of nodes
1	4.053	1,491,580
2	4.048	2,080,304
3	4.047	2,905,980
4	4.047	3,611,664

Table 2. Convergence analysis for no-slip conditions case, with $w^* = 5$, $h^* = 1.6$ and Re=59.

Mesh	C_d	Number of nodes
1	3.319	1,491,580
2	3.317	2,080,304
3	3.315	2,905,980
4	3.315	3,611,664

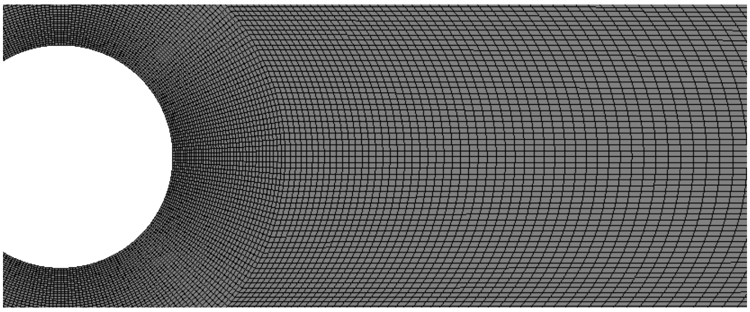

Figure 2. Schematic view of the adopted mesh (Mesh 3).

3. MATHEMATICAL MODEL

The laminar flow model was used in Ansys CFX software to numerically simulate the vortex shedding phenomenon in a microchannel with a confined micro-pin. The governing equations for this problem are the continuity and Navier-Stokes equations, given as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{v}) = 0 \quad (1)$$

$$\rho \left[\frac{\partial}{\partial t} \boldsymbol{v} + (\boldsymbol{v} \cdot \nabla) \boldsymbol{v} \right] = \rho \boldsymbol{g} - \nabla p + \mu \nabla^2 \boldsymbol{v} \quad (2)$$

where ρ is the fluid density, t is the time, p is the pressure, μ is the dynamic viscosity, $\boldsymbol{v}$ is the velocity vector with components $\boldsymbol{v}_x$, $\boldsymbol{v}_y$, and $\boldsymbol{v}_z$, and g is the gravitational acceleration vector.

The flow is assumed incompressible and the physical properties required in the governing equations were taken in the numerical simulations as ρ=1000 kg/m^3 and μ =10^{-3} Pa·s.

4. RESULTS AND DISCUSSION

First, a periodicity condition was set at the lateral walls for a Reynolds number equal to 100 and the effect of the lateral distance between the micropins on Strouhal number was investigated, where w^* is the lateral confinement (the ratio between W to d) and h^* is the vertical confinement (the ratio between H to d). As can be seen from Table 3, there is a good agreement between the present and previously published Strouhal number results (Renfer *et al.*, 2013a).

Table 3. Comparison of predicted Strouhal number with literature (Renfer *et al.*, 2013a).

Relative Width, *w**	Strouhal (present work)	Strouhal (*Renfer et al., 2013a*)	Relative deviation (%)
10	0.1767	0.1771	0.2257
5	0.2034	0.2000	1.7090
3.33	0.2341	0.2316	1.0691
2.5	0.2666	0.2688	0.8034
2	0.3071	0.3107	1.1679
1.66	0.3666	-	-

The case with *w**=1.66 was not investigated by Renfer *et al.* (2013a), however it presents the vortex shedding case with the highest Strouhal number studied in the present work. Fig. 3 shows the vorticity distribution for this case.

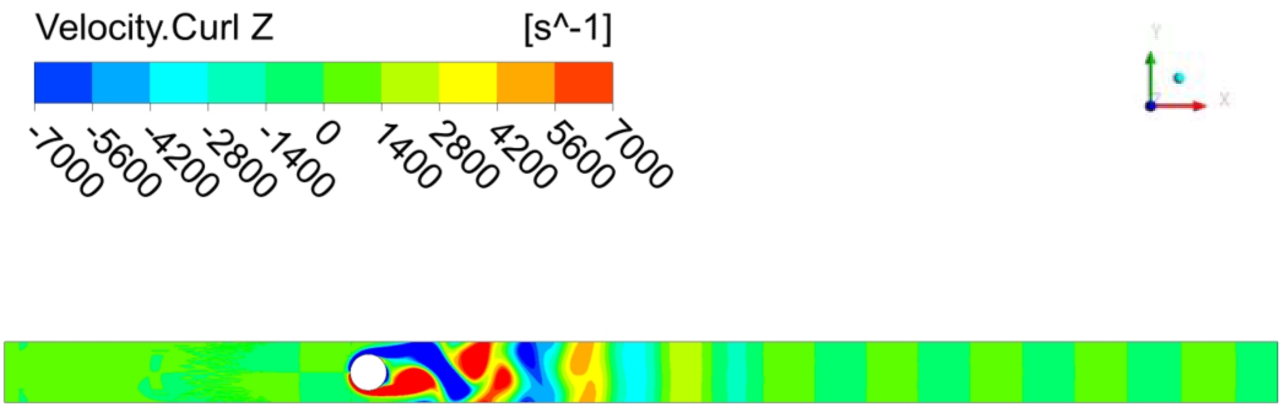

Figure 3. Results of velocity curl for Reynolds number equal to 100 with *w** = 1.66.

As shown in Table 3 and Fig. 4, the Strouhal number increases significantly as the distance between the pins decreases.

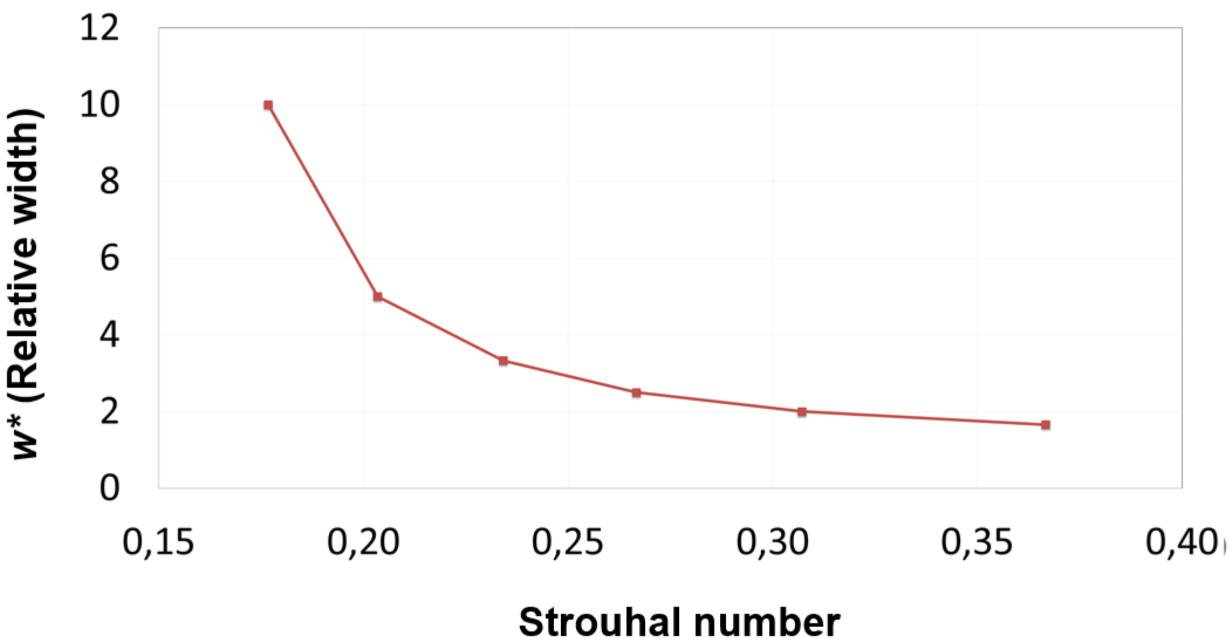

Figure 4. Variation of Strouhal number with relative width, *w**.

Next, no-slip conditions were set at the channel's walls which account for wall confinement as utilized in the experimental studies of Zhang et al. (2017). The corresponding drag coefficient and Strouhal number results for w^* = W/d = 5 and h^* = H/d = 1.6 are then presented in Table 4.

Table 4. Reynolds effect on the drag coefficient results for the no-slip conditions case, with w^* = 5 and h^* = 1.6.

Re	C_d	*St*
39	4.047	-
58	3.315	-

Vortex shedding does not occur for the cases presented in Table 4, in agreement with recent experiments (Zhang et al., 2017). The vorticity contours and velocity streamlines for one of these cases (Re = 39) are shown in Figures 5 and 6. Two fixed and symmetrical vortices are formed behind the cylinder and vortex shedding does not occur, as expected for this Reynolds number, on the basis of the theory on flows past a cylinder.

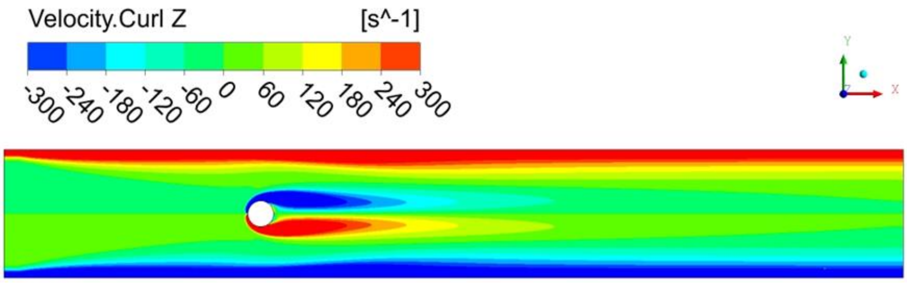

Figure 5. Results of velocity curl for Reynolds number equal to 39 (Table 2) with w^* = 5 and h^* = 1.6.

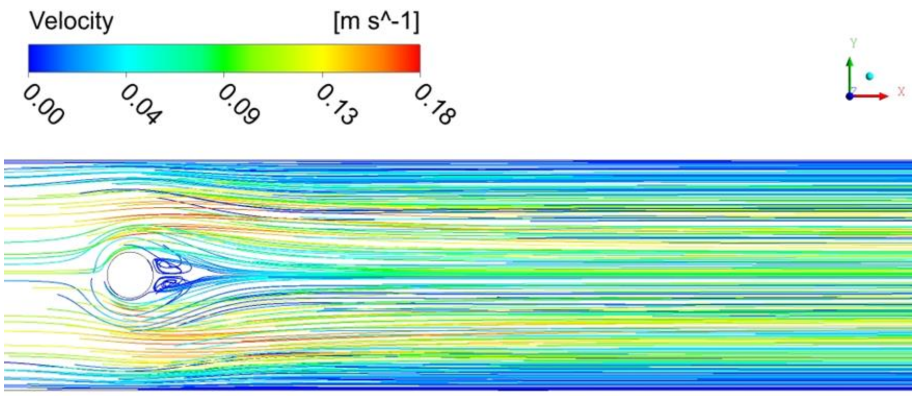

Figure 6. Results of velocity streamlines for Reynolds number equal to 39 (Table 2) with w^* = 5 and h^* = 1.6.

An additional case at higher Reynolds number was analyzed in order to capture the vortex shedding phenomenon as observed by Zhang *et al.* (2017). One of the cases in Zhang *et al.* (2017) was selected, in which vortex shedding occurred, when the Reynolds number was equal to 498, width-to-diameter ratio equal to 4, and height-to-diameter ratio equal to 2. For this case, the shear-stress transport turbulence model was utilized as the laminar flow model could not capture the Strouhal number once the regular oscillating behavior was not obtained. The laminar flow model is commonly used to represent flows in microchannels with unblocked cross sections, in light of the fairly low values of Reynolds numbers encountered in such situations. However, a detailed analysis on the critical Reynolds number for the onset of vortex shedding from a confined pin fin in a microchannel is lacking in the open literature. Therefore, the shear-stress transport turbulence model was selected to account for possible effects that might influence the present flow, since this model covers a wide range of Reynolds numbers and gives good overall results at a substantially lower computational cost compared to DNS (direct numerical simulation) and LES (large eddy simulation) models.

The flows in both the XY and XZ mid planes are visualized below, where the XY plane is transversal to the vertical walls and the XZ plane is parallel to the vertical walls. Figures 7 and 8 show the vorticity distribution obtained using the laminar flow model.

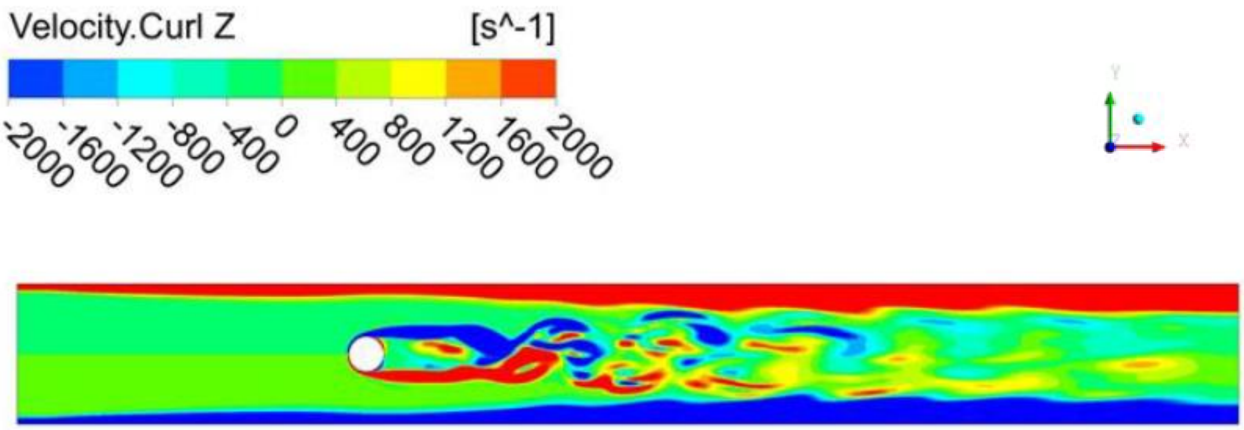

Figure 7: Results of velocity curl Z using laminar flow model for Re=498, $w^* = 4$ and $h^* = 2$, in a view of the XY plane.

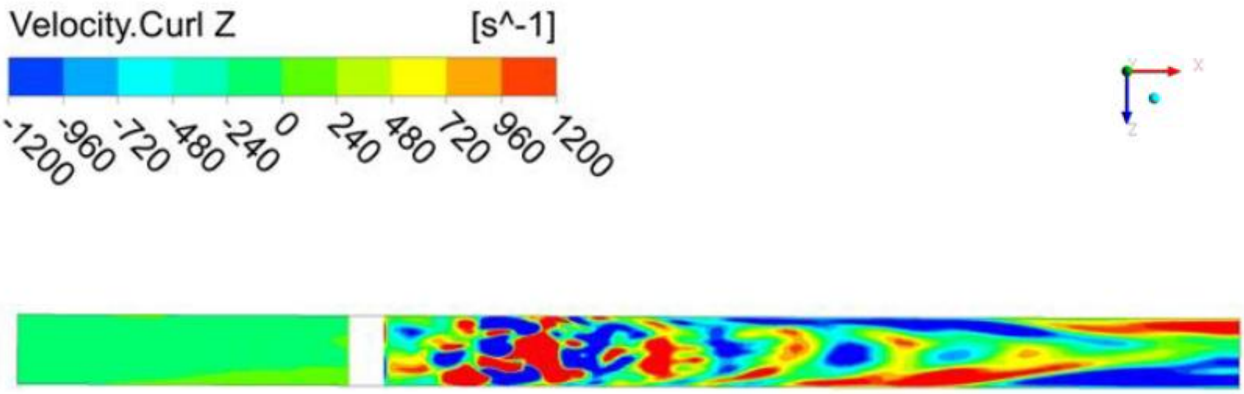

Figure 8. Results of velocity curl Z using laminar flow model for Re=498, $w^* = 4$ and $h^* = 2$, in a view of the XZ plane.

Figures 9 and 10 show the results of velocity streamlines and pressure for the same case of Re=498, $w^* = 4$ and $h^* = 2$, calculated using the laminar flow model.

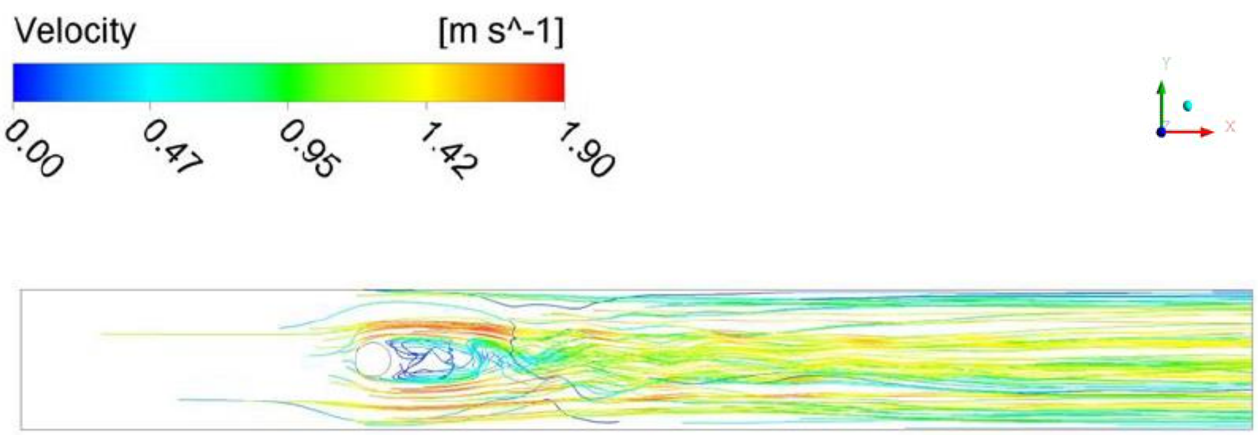

Figure 9. Results of velocity streamlines using the laminar flow model for Re=498, $w^* = 4$ and $h^* = 2$, in a view of the XY plane.

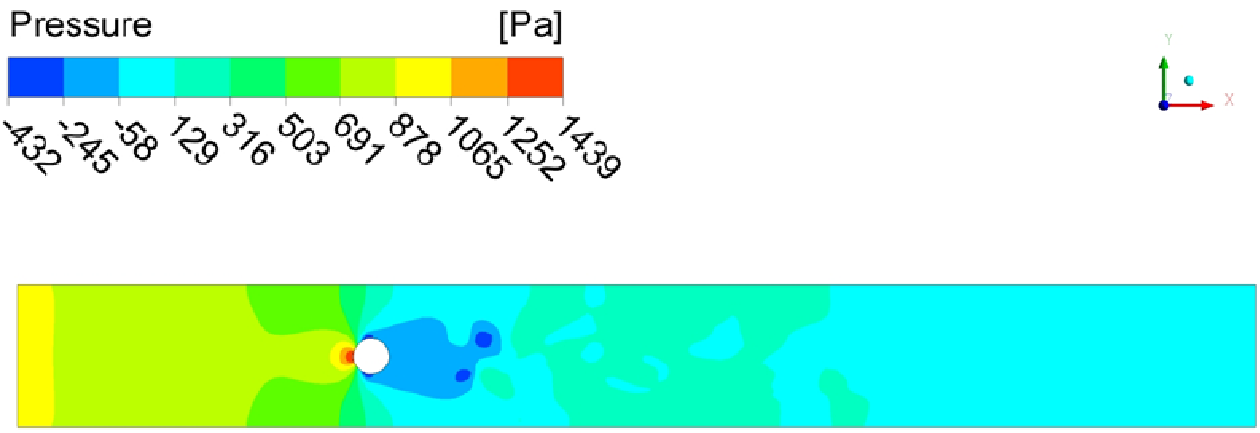

Figure 10. Results of pressure field using laminar flow model for Re=498, *w** = 4 and *h** = 2, in a view of the XY plane.

Figures 7 to 10, show that the two separated shear layers are elongated due to confinement and roll further downstream and hence no specific vortex structure or periodic phenomena can be identified. Switching to the shear-stress transport turbulence model results in slightly different vorticity distributions, as shown in Figures 11 and 12.

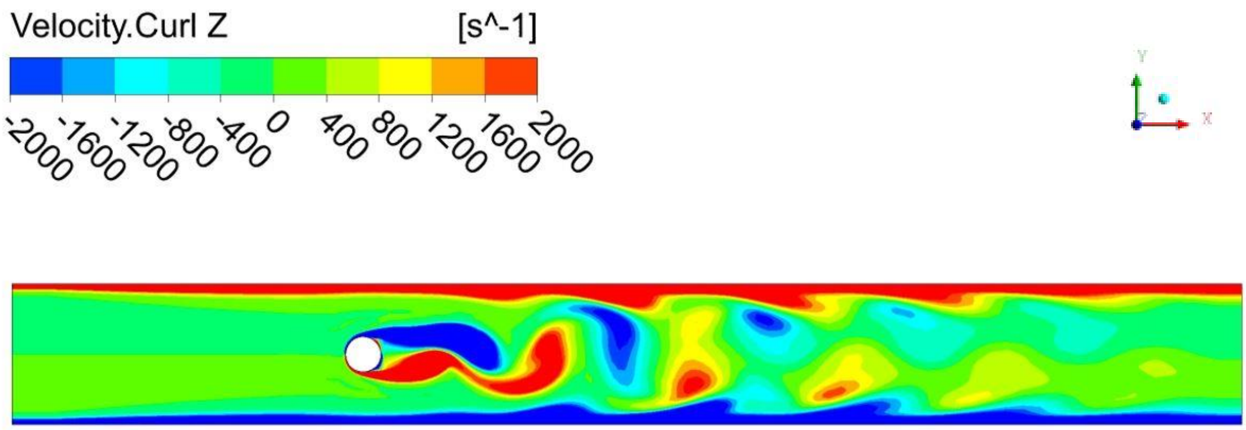

Figure 11. Results of velocity curl Z using SST model for Re=498, *w** = 4 and *h** = 2, in a view of the XY plane.

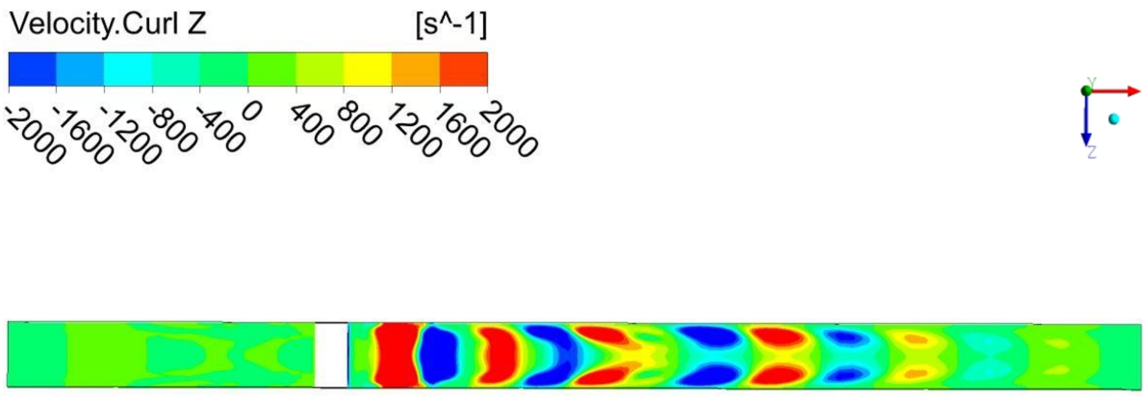

Figure 12. Results of velocity curl Z using SST model for Re=498, *w** = 4 and *h** = 2, in a view of the XZ plane.

Figures 13 and 14 show the results of velocity streamlines and pressure field for the shear-stress transport turbulence model, again with Re=498, w^* = 4 and h^* = 2.

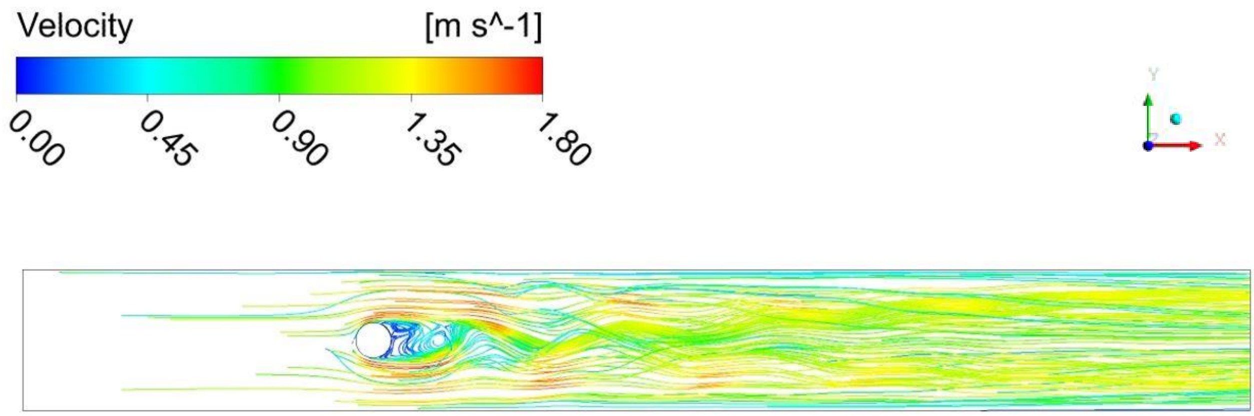

Figure 13. Results of velocity streamlines using the SST model for Re=498, w^* = 4 and h^* = 2, in a view of the XY plane.

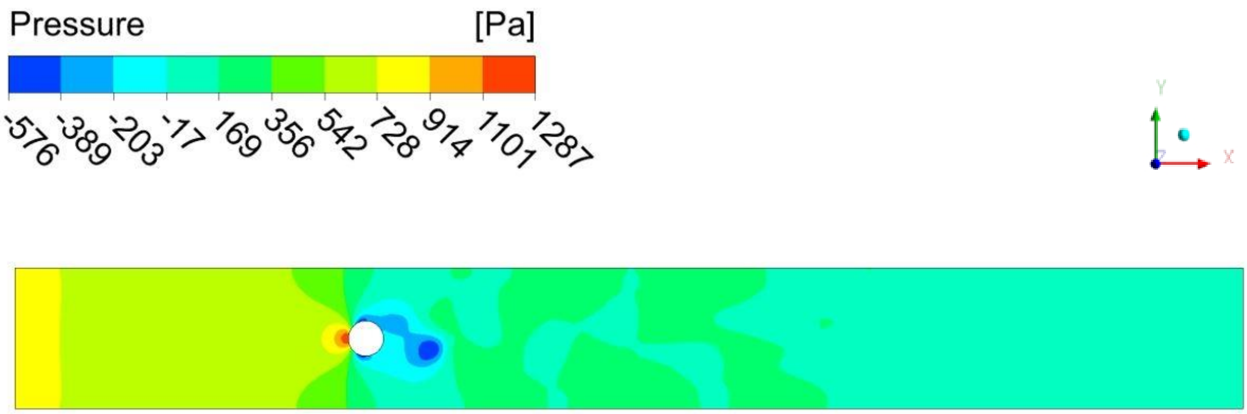

Figure 14. Results of pressure field using the SST model for Re=498, w^* = 4 and h^* = 2, in a view of the XY plane.

It can be clearly observed, from these images, the vortex shedding behind the cylinder, unlike the previous cases (Re=39), when just two fixed vortices were present.

Finally, Table 5 illustrates the excellent agreement between the numerical and experimental results for the Strouhal number in this particular case with vortex shedding. In addition, the computed drag coefficient for this case is C_d=1.618.

Table 5 – Results for the Strouhal number in the case with vortex shedding, Re=498, w^* = 4 and h^* = 2.

	Present work	*Zhang et al.* (2017)	Relative deviation (%)
Strouhal number	0.28	0.26	7.7

5. CONCLUSIONS

In this work, the flow over a cylindrical pin confined within a microchannel is analyzed, towards the understanding of the influence of the channel's stationary walls on the flow behavior, via the corresponding Strouhal number and drag coefficient values. It was observed that with an increase of the lateral confinement, the Strouhal number goes up significantly as well. Furthermore, the increase of microchannel bottom/up walls confinement acts as a buffer, favoring the formation of two fixed and symmetrical vortices behind the cylinder and leading to a suppression of the vortex shedding. It was noted that depending on the Reynolds number and the dimensions of the microchannel, a turbulence model should be prefered to capture the trends observed experimentaly. The present simulations further clarify the role of the confinement effect on vortex shedding, leading to a good agreement between numerical and experimental results and provide a first step towards the rational design of pin fin microsystems for biofuel synthesis applications.

6. ACKNOWLEDGEMENTS

The authors acknowledge the financial support of the sponsoring agencies CNPq, CAPES-INMETRO, and FAPERJ, from Brazil, and the UK Newton Fund (EP/M029573/1), as well as the technical support from ESSS, Brazil.

7. REFERENCES

Bardina, J. E., Coakley, T. J. and Huang, P. G., 1997. "Turbulence Modeling Validation, Testing and Development". AIAA Journal, pp. 97-2121.

Chakraborty, J., Verma, N. and Chhabra, R. P., 2004. "Wall effects in flow past a circular cylinder in a plane channel: a numerical study". Chemical Engineering and Processing, pp. 1529–1537.

Costa Junior, J. M., Naveira Cotta, C. P., Nunes, J., Tostado, C. P., 2015. "Design, fabrication and characterization of a microeactor for biodiesel synthesis". Heat Pipe Science and Technology, V.6(3-4), pp. 135-153.

Kanaris, N., Grigoriadis, D. and Kassinos S., 2011. "Three-dimensional flow around a circular cylinder confined in a plane channel". Physics of Fluids.

Pontes, P. C., Chen, K., Naveira Cotta, C.P., Costa Junior, J. M., Tostado, C.P., Quaresma, J. N. N., 2016. "Mass transfer simulation of biodiesel synthesis in microreactors". Computers & Chemical Eng. Journal, v. 93, pp36-51.

Rajani, B. N., Kandasamy, A. and Majumdar, S., 2009. "Numerical simulation of laminar flow past a circular cylinder". Applied Mathematical Modeling, pp. 1228-1247.

Renfer, A., Alfieri, F., Tiwari, M. K., Brunschwiler, T., Michel, B. and Poulikakos D., 2013a. "Computational modeling of vortex shedding in water cooling of 3D integrated electronics". International Journal of Heat and Fluid Flow, pp. 745-755.

Renfer, A., Tiwari, M. K., Meyer, F., Brunschwiler, T., Michel, B. and Poulikakos D., 2013b. "Vortex shedding from confined micropin arrays". Microfluid Nanofluid.

Renfer, A., Tiwari, M. K., Brunschwiler, T., Michel, B. and Poulikakos D., 2011. "Experimental investigation into vortex structure and pressure drop across microcavities in 3D integrated electronics". Experiments in Fluids, pp. 731-741.

Son, J. S. and Harranty, T. J., 1969. "Numerical solution for the flow around a cylinder at Reynolds numbers of 40, 200 and 500". Journal of Fluid Mechanics, pp. 369-386.

Zhang, S., Tiwari, M. K., Balabani, S., Naveira Cotta, C. P. and Cotta, R. M., 2017. "Residence time control in micromixers with vortex shedding". Iguazu Falls, Brazil: 9th World Conferences on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics - ExHFT-9.

8. RESPONSIBILITY NOTICE