CFD Letters 13, Issue 11 (2021) 65-77

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Study on Magnetohydrodynamic Flow Past Two Circular Cylinders in Staggered Arrangement

Nur Marissa Kamarul Baharin[1], Mohd Azan Mohammed Sapardi[1,*], Nur Nadhirah Ab Razak[1], Ahmad Hussein Abdul Hamid[2], Syed Noh Syed Abu Bakar[1]

[1] Department of Mechanical Engineering, Kulliyyah of Engineering, International Islamic University Malaysia, Jalan Gombak, 53100, Selangor, Malaysia
[2] Fakulti Kejuruteraan Mekanikal, Universiti Teknologi Mara Malaysia, 40450 Shah Alam, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 14 August 2021
Received in revised form 11 October 2021
Accepted 12 October 2021
Available online 20 November 2021

Keywords:
Magnetohydrodynamic Flow; OpenFOAM; Heat Transfer; Hartmann number; Nusselt number; Reynolds number

ABSTRACT

The fusion reactor is anticipated to be a new source of clean energy. Magnetohydrodynamic flow in the fusion blanket is expected to cause the flow to be highly stable, causing the heat transfer to be poor. Passive vortex promoter such as bluff body is one of the methods found to be has a great potential in optimizing the heat transfer. In this study, two circular cylinders in a staggered arrangement are introduced to promote vortices to enhance heat convection from a heated wall using an electrically conducting fluid under a constant magnetic field. The effect of the Hartmann friction parameter and the height differential onto the Nusselt number were examined. Modified Navier—Stokes equations known as SM82 were used using OpenFOAM to simulate the confined, quasi-two-dimensional, incompressible and laminar MHD flow past the bluff bodies. It was found that the heat transfer is better when the height differential is small.

1. Introduction

The Magnetohydrodynamic (MHD) flow understanding is crucial in designing blankets for liquid-metal fusion reactors. The interaction between a strong and uniform magnetic field in an axial direction to the direction of the flow causes the flow to be highly stable. The modification of the flow pattern affects the heat transfer in the channel [1].

MHD in a fusion blanket occurs when a liquid metal flowing under a strong and uniform magnetic field used to confine the plasma in a fusion reactor induces a current. The induced current then interacts with the magnetic field, inducing a force in the opposite direction of the fluid motion, also known as the Lorentz force [2]. It creates a braking effect on the fluid’s motion, which can also be considered as an additional electromagnetic “drag” term that consequently interfere with the heat and mass transport mechanisms.

* *Corresponding author.*
E-mail address: azan@iium.edu.my (Mohd Azan Mohammed Sapardi)

https://doi.org/10.37934/cfdl.13.11.6577

In a study made by Singh and Gohil [3], they found that the transverse magnetic field acting on a flow suppresses heat transfer significantly. Umeda and Takahashi [4] did a numerical study on the MHD flow in a rectangular channel in the presence of a transverse uniform magnetic field. They concluded that magnetic field-induced flow laminarisation inhibits heat transfer. Many studies show an agreement that heat transfer is enhanced when a vortex promoter is introduced in a flow [5]. This is because the vortex shedding emerged from the promoter improves the mixing of the hot and cold fluid, increasing the efficiency of heat convection in the fluid.

The interaction between the electrically conducting fluid in motion with a strong and uniform magnetic field perpendicular to the direction induces Lorentz force. The induced force is acting towards the opposite direction of the motion of the fluid. It causes the flow tends to become quasi-two-dimensional. It is also very stable and damps all vortical structures parallel to the direction of the magnetic field, causing the MHD flow to be poor in heat transport [6,7].

Moreover, the presence of the magnetic field also causes a significant drop in pressure and laminarisation of the fluid flow [8]. Several other works found a similar observation in their numerical study investigating the flow structure of an MHD flow past a bluff body in a duct at high Hartmann number [6,9,10].

According to previous research, the magnetic field has an effect on the flow structure by stabilising the flow and lowering the heat efficiency. To introduce vortex dynamics into a flow, bluff bodies can be added to the channel [11]. Most studies have been conducted on circular cylinder bluff bodies and adding two circular cylinders as bluff bodies further increase the channel's unsteadiness [12]. Between two arrangements, it is discovered that the second cylinder behaves almost identically to an isolated cylinder in almost all cases when the arrangement is staggered. At the same time, very large amplitude oscillations are observed for the second cylinder in a tandem arrangement [12]. Previously, a study was conducted on the vortex induction of two cylinders in tandem arrangement. According to Borazjani and Sotiropoulos [13], when compared to an isolated cylinder, the tandem arrangement has larger motion amplitudes and a wider lock-in region, and above a threshold reduced velocity, large-amplitude motion is excited for the rear cylinder with amplitudes significantly greater than those of the front cylinder. In terms of unsteadiness, multiple cylinders is preferred because it allows for the observation of oscillations with significant amplitude.

Numerous studies on optimising heat transfer have been conducted using a bluff body as a vortex promoter. Multiple bluff bodies in the channel are expected to improve heat transfer. From the author's knowledge, no study has been done on the effect of two circular cylinder bluff bodies on the flow structure and heat transfer in an MHD flow. This study encompasses the effect of two circular cylinders in a staggered arrangement.

2. Methodology

2.1 Geometry Description

The system of interest under consideration is shown in Figure 1. The geometry is a rectangular channel confining two circular cylinders arranged in staggered. The walls of the channel and cylinders are assumed to be non-slip and electrically insulated in this problem. A uniform vertical magnetic field with a strength B is imposed perpendicular to the *x-y* plane. The bottom wall (parallel orientation to the magnetic field) is heated with a constant wall temperature θ_{hot} (fixed at value 1) whereas the top wall is set with $\theta = 0$. The heated wall with the interaction of the cylinders inside the channel is expected to contribute to heat transfer. A similar setup is used for the validation case but for a single-cylinder case where only an upstream cylinder is considered. The result is compared with the work of Mathupriya *et al.,* [14].

For this study, the blockage ratio, β, the height of the channel, *h*, and the gap ratio, *L/h,* are fixed at 0.3, 1, and 1.2, respectively. The blockage ratio is defined as the ratio of the upstream cylinder diameter, *D,* to the channel height, *h*. The two cylinders are set to have the same diameter, which is represented by *D*. The differential height ratio, L_y/h will be varied at L_y/h = 0.05, 0.1, 0.15, 0.2 and 0.25.

2.2 Governing Equations

Sommeria and Moreau [15] stated that at a high Hartmann number, the magnetic Reynolds number (which represents the ratio of induced and applied magnetic fields) is extremely small. The induced magnetic field is negligible in this case, and the resulting magnetic field is imposed only in the *z*-direction. The flow is quasi-two-dimensional under these conditions and consists of a core region. The velocity is invariant in the magnetic field direction, and a thin Hartmann layer at the wall perpendicular to the magnetic field [7]. Thus, they derived a two-dimensional model that governs this problem by averaging the flow quantities along the magnetic field direction.

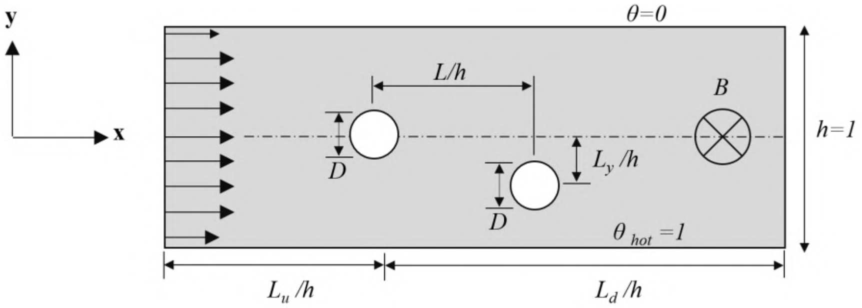


Fig. 1. Schematic drawing of the case under investigation

Schematic drawing of the case is shown in Figure 1, where it shows that the magnetic field, B is parallel to the cylinder axis and acts in a z-direction in the out-of-plane direction. h is the height of the channel, Lu/h is the upstream length ratio starting from inlet to centre of the upstream cylinder, Ld/h is the downstream length ratio, and D is the diameter of both the upstream and the downstream cylinders. The blockage ratio is $\beta = D/H$, and the gap ratio between the cylinders is L/h. The MHD flow inside the channel in Figure 1 is governed by Eq. (2) for a quasi-2D model. Sommeria and Moreau [15] proposed this model in 1982 [15,16], where the flow is assumed to be incompressible and laminar. By averaging the flow quantities in the core flow and Hartmann layers, this model approximates the MHD flow in a two-dimensional perspective. The non-dimensional magnetohydrodynamics equations of continuity (Eq. (1)), momentum (Eq. (2)) and energy equations (Eq. (3)) are reduced to

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} = -(u \cdot \nabla)u - \nabla p + \frac{1}{Re}\nabla^2 u - \frac{H}{Re}u \tag{2}$$

$$\frac{d\theta}{dt} + (u \cdot \nabla)\theta = \frac{1}{Pr\ Re}\nabla^2 \theta \tag{3}$$

where, u represents the velocity vector, p represents the kinematic pressure, and θ is the dimensionless temperature field $\theta = (\hat{\theta} - \hat{\theta}_{cold})/(\hat{\theta}_{hot-}\hat{\theta}_{cold})$. $\hat{\theta}$ represents dimensional temperature fields. H represents the friction term representing the Lorentz force effect on the flow, Re represents Reynolds number, and Pr is Prandtl number. The viscous dissipation and Joule heating terms are not considered in this case following previous studies [7,17,18].

Reynolds number is a dimensionless parameter that depends on the value of maximum inlet velocity, U, characteristic length (the channel inlet height), h, and kinematic viscosity, ν where it characterises the ratio of inertial forces to viscous forces. The dimensionless parameters of Reynolds number (Eq. (4)), Hartmann number (Eq. (5)), Peclet number (Eq. (6)) and Hartmann friction term (Eq. (7)) are defined as

$$Re = \frac{\rho U h}{\mu} = \frac{Uh}{\nu} \tag{4}$$

$$Ha = aB\sqrt{\frac{\sigma}{\rho \nu}} \tag{5}$$

$$Pe = RePr \tag{6}$$

$$H = n\frac{h^2}{a}\, Ha \tag{7}$$

where a, B, and σ are the half of out-of-plane channel height, the applied magnetic field, and the magnetic permeability of the liquid metal, respectively. The Prandtl number in Peclet number, *Pe*, is given by $Pr = \nu/\kappa_T$, where κ_T is the thermal diffusivity of the fluid, characterises the ratio of viscous to thermal diffusion in the fluid.

The local Nusselt number along the lower heated wall of the channel is given by Eq. (8) [19,20],

$$\mathrm{Nu_w}(x,t) = \frac{1}{\theta_f - \theta_w}\frac{\partial \theta}{\partial y}\Big|_{wall} \tag{8}$$

ϑ_f is the bulk fluid temperature, which is calculated using the temperature distribution and velocity as in Eq. (9),

$$\theta_f(x,t) = \frac{\int_0^h u\theta dy}{\int_0^h u dy} \tag{9}$$

A time-averaged Nusselt number is calculated for heat transfer through the heated wall by averaging the local Nusselt number over a few time steps and then integrating over the length of the heated bottom wall, L [7].

The Strouhal Number can be calculated using the frequency of vortex shedding as in Eq. (10),

$$St = \frac{fh}{u} \tag{10}$$

where f is the frequency of shedding.

To proceed with the analysis of the result, lift coefficient, C_L is also used for this study which defined as in Eq. (11),

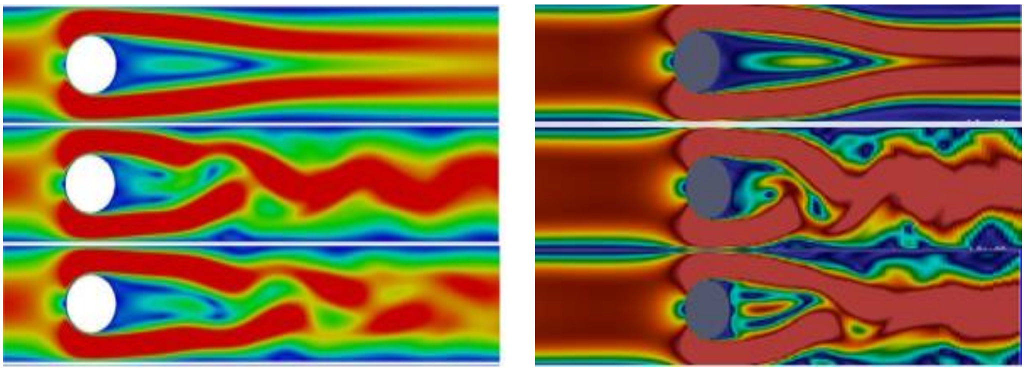
$$C_L = \frac{F_L}{\frac{1}{2}\rho u^2 h} \tag{11}$$

where F_L represents lift force.

The flow depicted in Figure 1 is studied for its flow behaviour. The blockage ratio, β is 0.3, Hartmann parameter, H = [0-200] and Re = [1-2000]. Since the quasi-two-dimensional condition established by Sommeria and Moreau [15] and Pothérat *et al.,* [21] is satisfied, the MHD flow in this research is assumed to be quasi-two-dimensional. A Prandtl number, Pr = 0.022 is used, which is a representative of Galinstan (*GaInSn*) liquid metal, for the heat transfer study.

2.3 Validation with the Previous Study

Validation for this project has been made using the previous study done by Mathupriya *et al.,* [14] on the flow past a circular cylinder. Figure 2 shows the contour of the magnitude of velocity generated in the study and in the current study at different times. Both show quite a resemblance to each other. The Strouhal number, *St*, in both their and current study agrees well, where *St* = 0.0333.



(a) (b)

Fig. 2. The velocity contour from (a) the study by Mathupriya *et al.,* [14] and (b) the present study at *Re* = 200

At the inlet, a fully developed velocity profile is imposed, while at the outlet, a constant reference pressure is imposed. The simulation-generated velocity profile at various H values is compared to the analytical Hartmann fully developed velocity profile. Based on Figure 3, at $H = 0$, the curve can be observed to be a quadratic profile, and as H increases, the profile becomes fuller and flattened. The inlet and outlet velocity of the channel seems to be in good agreement. Non-slip boundary conditions are imposed on the walls and the cylinders. The top wall non-dimensional temperature is set at $\theta = 0$, representing cold wall, and the bottom wall is $\theta_{hot} = 1$, while the cylinders are thermally insulated with zero temperature gradient surface.

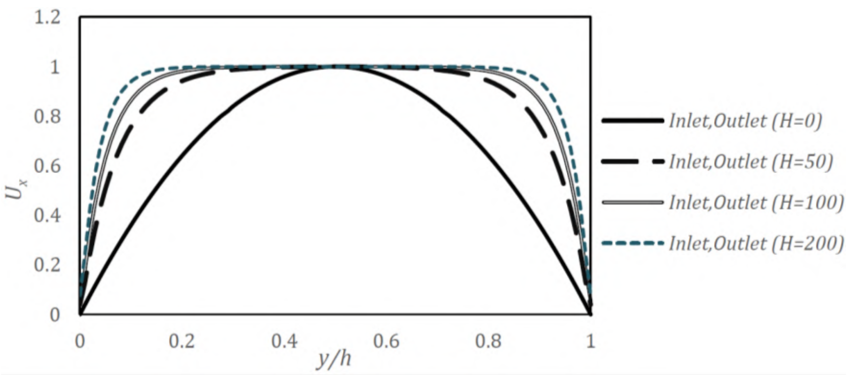


Fig. 3. Normalised velocity at the inlet and out of the channel at H = 0, 50, 100 and 200

3. Results

3.1 Mesh and Domain Dependency Study

This section discusses the grid dependency analysis and validations performed. The grid dependency study is based on the error percentage Strouhal number, *St*. The term is essential in analysing unsteady and oscillating flow problems. The number of elements is varied between 20,000-100,000. The desired error threshold was met using mesh M4 in Table 1, which is used hereafter. The deviation in the reading of *St* is very insignificant compared to M4 when the element is further increased.

Table 1
Percentage of error for *St* calculated with the different number of elements

Mesh	No. of Elements	*St*	Error, *St* (%)
M_1	20876	0.91743	2.11
M_2	26798	0.89847	0.27
M_3	35096	0.89606	0.36
M_4	52916	0.89286	0.09
M_5	74336	0.89366	-

Domain dependency study was done to ensure the domain size is adequately large so that the boundary condition does not affect the structure of the flow; hence, causing inaccurate results in simulation. In this study, the distance of the inlet boundary from the upstream cylinder varied. At the inlet, a fully developed Hartmann velocity profile is imposed; thus, the distance between the inlet boundary and the upstream cylinder needs to be far enough until the flow is independent to the upstream length. Figure 4 shows the *z*-vorticity contour for various upstream lengths for the same *Re*.

Table 2 shows the error of *St* and C_L readings with respect to the longer upstream length. It is found that as the upstream length goes beyond L_u/h = 2, the error is insignificant. Hence, the upstream length, L_u/h = 3, is chosen throughout this study.

(a)
(b)
(c)
(d)
(e)
(f)
(g)
(h)

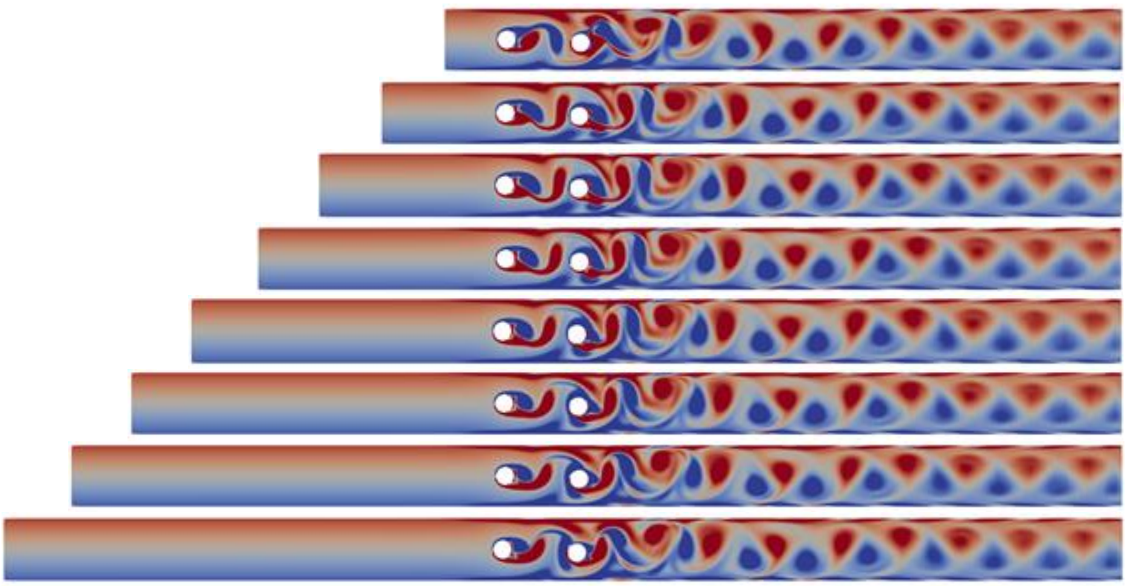


Fig. 4. Vorticity contour at upstream length, L_u/h = (a) 1, (b) 2, (c) 3, (d) 4, (e) 5, (f) 6 and (h) 8

Table 2
Percentage of error for Strouhal number and Lift coefficient at upstream, L_u/h = 1, 2, 3, 5, 6, 7 and 8

Length	*St*	Error *St* (%)	C_L	Error C_L (%)
1	0.9274	0.39	0.0020016	0.80%
2	0.9237	0.03	0.0021092	0.02%
3	0.9234	0.02	0.0021094	0.02%
4	0.9238	0.02	0.0021099	0.02%
5	0.9230	0.02	0.0021109	0.02%
6	0.9237	0.02	0.0021123	0.02%
7	0.9238	0.02	0.0021146	0.02%
8	0.9240	0.02	0.0021168	0.02%

3.2 The Effect of the Height Differential on the Critical Reynolds Number

This section discusses the effect of the height differential on the critical Reynolds number, Re_c, representing the threshold when the transitions from steady to unsteady flow occur. This study focuses on shifting the downstream cylinder closer to the hot wall, expecting that the shedding of the vorticity behind it will provide a better fluid mixing; hence, it improves the convection of heat from the hot wall to the fluid. The height differential, L_y/h, is varied with the value of 0.05, 0.1, 0.15, 0.20 and 0.25. Figure 5 shows the *z*-vorticity contour with various L_y/h, at *H* = 50 and *Re* = 800. It can be seen that as the height differential is getting bigger, the flow is vortex shedding getting less intense and eventually, the flow becomes steady at a bigger L_y/h.

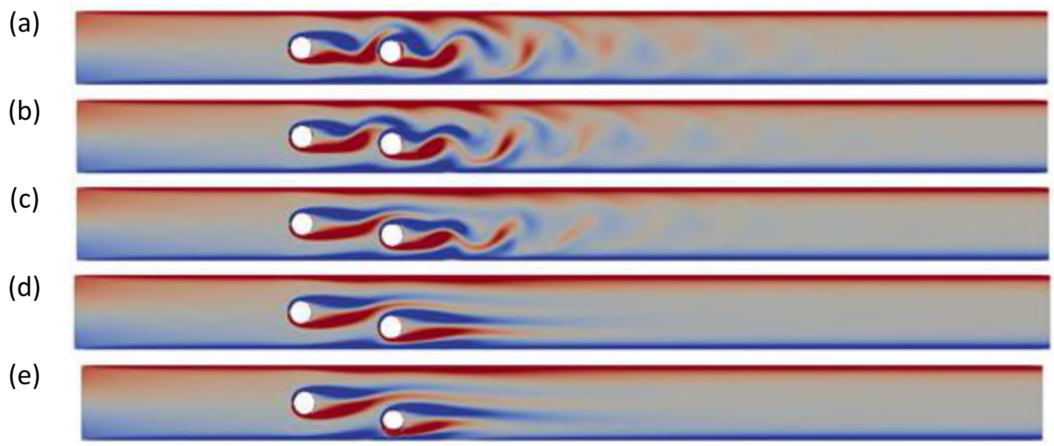


Fig. 5. The flow passing two cylinders in staggered arrangement at *H* = 50 and *Re* = 800 at L_y/h = (a) 0.05, (b) 0.1, (c) 0.15, (d) 0.2, and (e) 0.25

The critical Reynolds number is determined by finding the largest *Re* with a steady solution and the smallest *Re* with an unsteady solution. The parameter used as the criteria of unsteadiness in this study is the coefficient of lift force, C_L, on the downstream cylinder. Figure 6 shows the sample of C_L time-history at which the flow is (a) steady (L_y/h = 0.05, *Re* = 600) and (b) unsteady (L_y/h = 0.05, *Re* = 800).

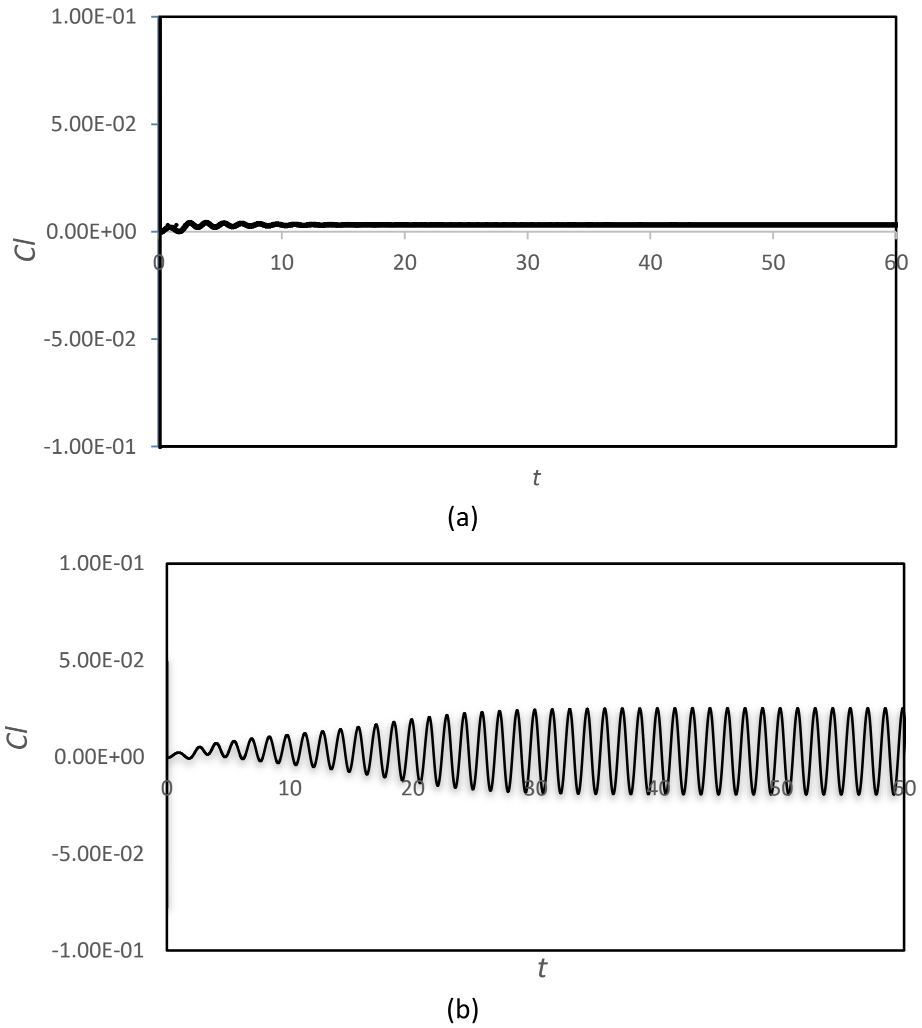


Fig. 6. The time-history of C_L for (a) a steady flow and (a) for an unsteady flow

The critical Reynolds number also can be determined qualitatively by observing the *z*-vorticity contour. Figure 7 shows the *z*-vorticity contour of (a) a steady (L_y/h = 0.05, *Re* = 600) and (b) an unsteady (L_y/h = 0.05, *Re* = 800) flows. This observation narrows down the predicted Re_c to be in between those two *Re*.

Figure 8 illustrates the value of Re_c at different L_y/h and *H*. Similar to the discussion in other studies on MHD; the critical Reynolds number is delayed as *H* is increased [9,10]. On the other hand, it is also can be seen that as the downstream cylinder getting closer to the bottom wall, the flow requires higher *Re* to become unsteady.

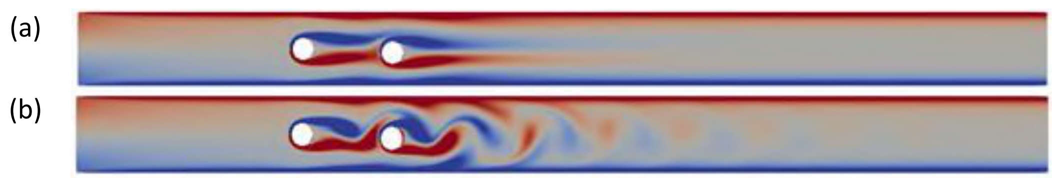


Fig. 7. The flow for a steady flow at (a) *Re* = 600, and (b) an unsteady flow at *Re* = 800

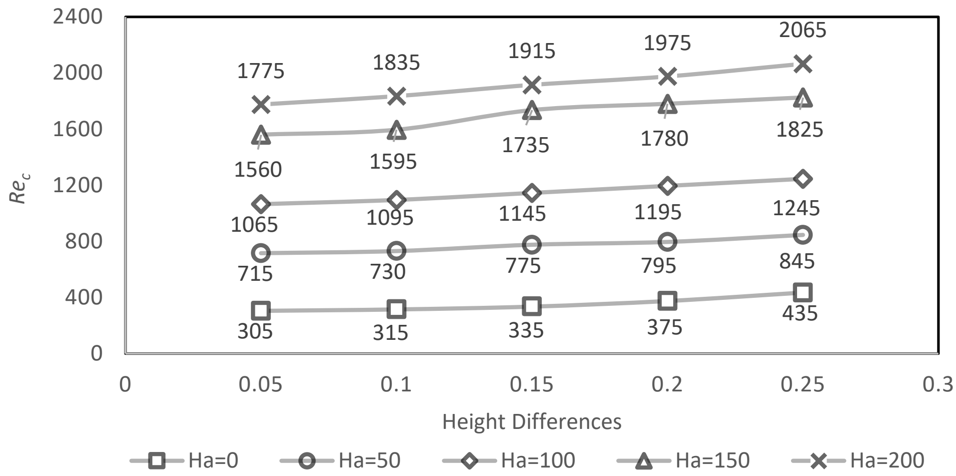


Fig. 8. The graph for the Re_c at different height Differential with different Hartmann friction parameter

3.3 The Effect on the Pressure Drop

Figure 9 shows the effect of *Re* and L_y/h on the pressure drop in the channel. The non-dimensional pressure drop $\left(\Delta p = {}^{\Delta\hat{p}}\!/_{\frac{1}{2}\rho U^2}\right)$ is inversely proportional to *Re*. Whereas, as the distance of the downstream cylinder is closer to the wall, the pressure drop is higher. This is because the blockage area increased across the channel; hence, causing the flow to be difficult to pass through the obstacles.

Figure 10 shows the effect of *H* on the pressure drop at various L_y/h at *Re* = 1000. As can be expected, the additional forcing term, the Hartmann friction term, represents the effect of Lorentz force on the flow causing the pressure drop to increase as *H* increases.

3.4 Heat transfer between Two Cylinders in a Channel

Figure 11 shows the temperature contour of a flow with L_y/h = 0.05, *Re* = 1200, *H* = 50. The addition of bluff bodies in the flow causes the flow to be able to become unsteady. This improves the fluid mixing in the fluid; hence, it contributes to the increase in Nusselt number, the ratio between the convective over conductive heat transfer. Since the main objective of the application is to transfer the heat from the hot wall into the fluid to be transferred to the outlet, it is of our interest to have high *Nu*.

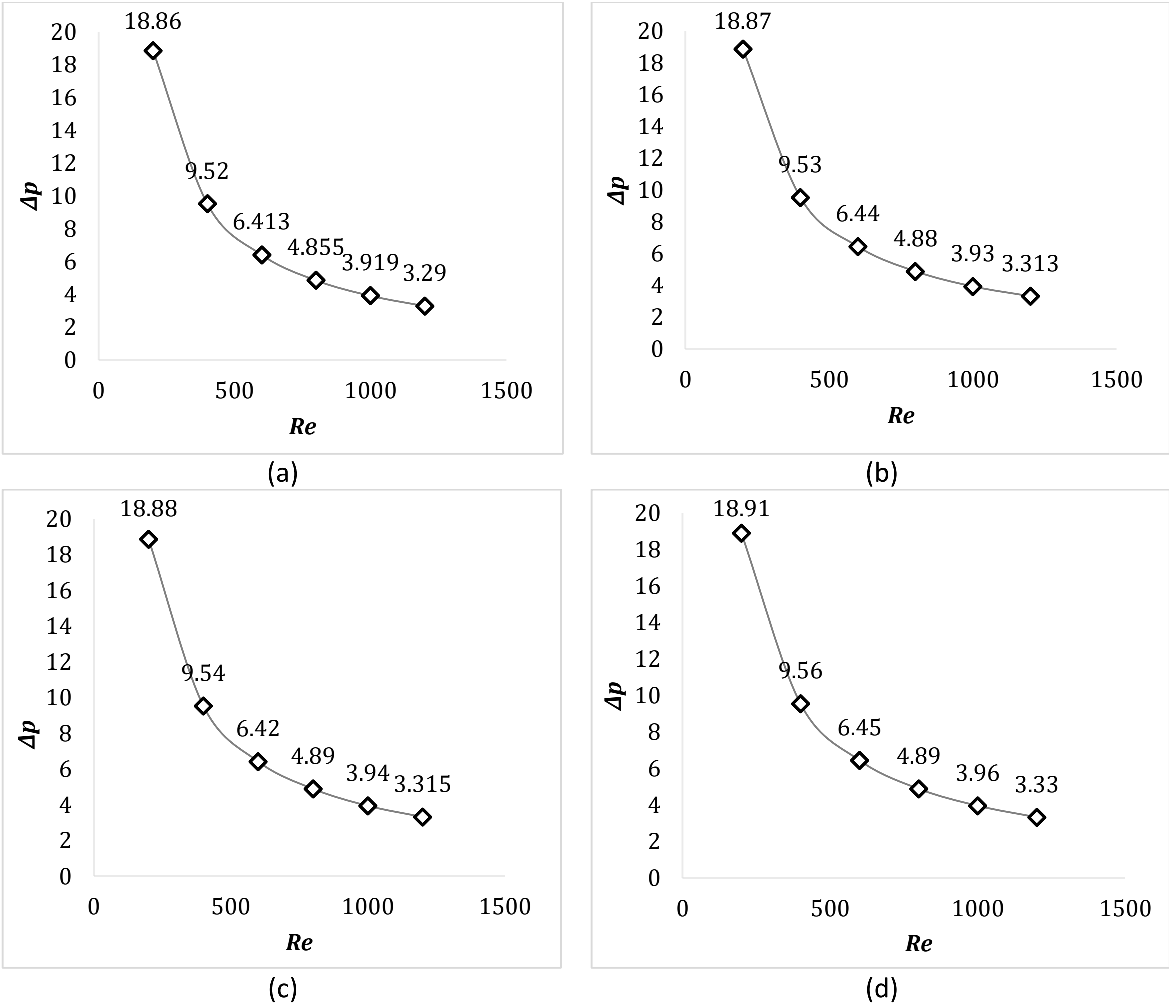


Fig. 9. The graph for the pressure drops with variation Reynolds Number at different height differential (a) L_y/h = 0.05, (b) 0.1, (c) 0.15 (d) 0.2

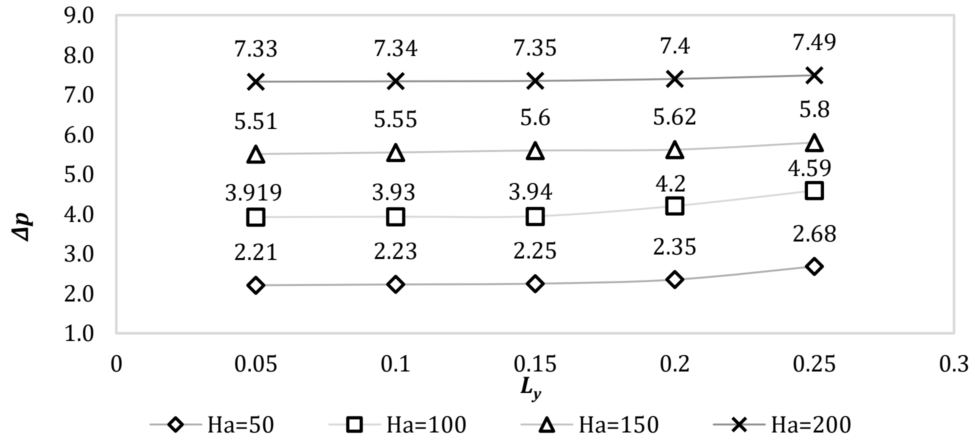


Fig. 10. The graph for the pressure drops penalty of H = 50, 100, 150 and 200 at different height differential for Re = 1000

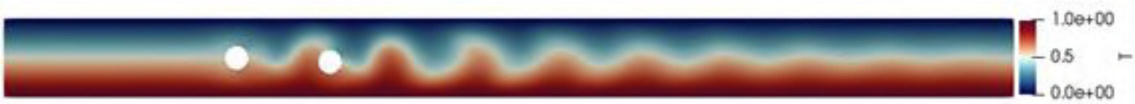


Fig. 11. The temperature gradient contour for L_y/h = 0.05, Re = 1200 at H = 50

Figure 12 illustrates the time-averaged Nusselt number as a function of *Re* for various L_y/h at *H* = 50. For all ranges of *Re* studied, the smaller the L_y/h, the larger the *Nu*. For all L_y/h, the value of *Nu* is independent of *Re* when *Re* is small. The value of *Nu* then observed to increase with the increase of *Re* starting approximately near the value of $Re \approx Re_c$; proving that the vortex shedding behind the cylinders contributes to enhancing the heat removal from the hot wall.

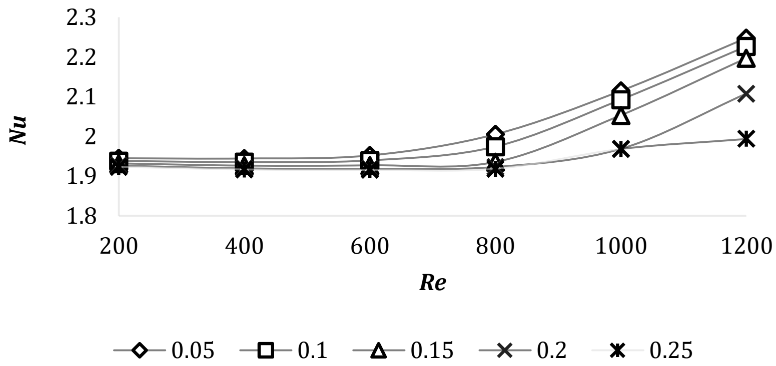


Fig. 12. Time-averaged Nusselt number plotted against Reynolds number for different height differential between two-cylinder, L_y = 0.05, 0.1, 0.15, 0.2 and 0.25 at *H* = 50

3.5 The Effect of Hartmann Friction Parameter on Heat Transfer Efficiency

Figure 13 shows the value of *Nu* as a function of *Re* at L_y/h = 0.05 and 0.01. It is evident that the higher the value of *H,* the lower the *Nu*. This agrees well with the finding of the previous studies, where the stability introduced by the constant magnetic field perpendicular to the flow inhibits the transfer of heat due to the poor fluid mixing [10,22]. Moreover, at the range of *Re* = [200-1200], *Nu* is always lower for L_y/h = 0.1 compared to those of L_y/h = 0.05.

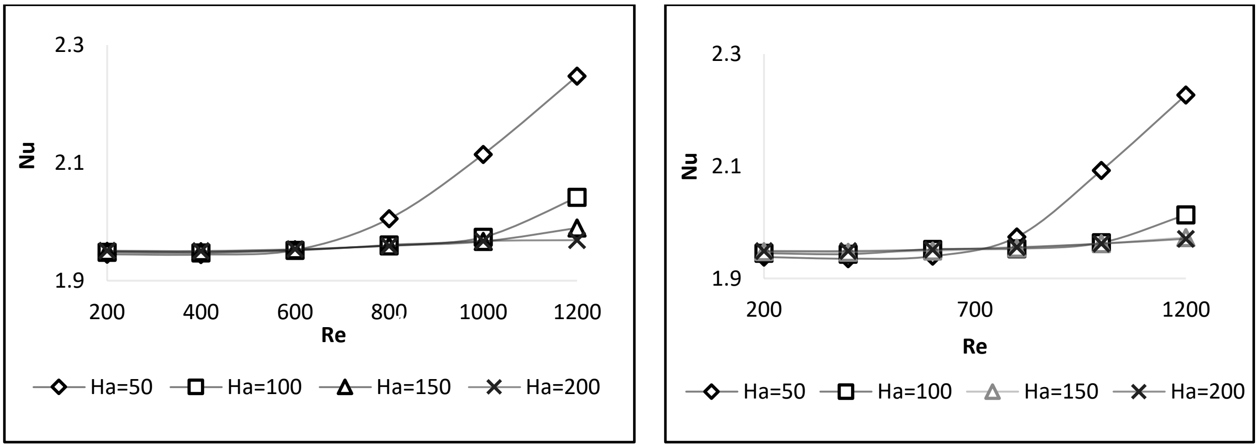


Fig. 13. Time-averaged Nusselt number plotted against Reynolds number for different Hartmann number at (a) L_y/h = 0.05 and (b) L_y/h = 0.1

4. Conclusions

In conclusion, this study numerically investigates the structure of the liquid metal flow pass two circular cylinders in staggered arrangement and how its effect on the heat transfer efficiency and the pressure drop by changing the location of the downstream cylinder vertically. The manipulated parameters studied in this paper are *Re*, *H*, and L_y/h. The simulations were done by using the modified Navier—Stokes equations to cater the effect of the uniform magnetic field perpendicular to the flow, known as SM82 quasi-two-dimensional model [15,21]. The numerical analysis found that the more inline the downstream cylinder with the upstream cylinder at the centreline of the channel, the better the convection of heat from the heated wall into the fluid. It can also be concluded that the pressure drop is lower if the downstream cylinder is closer to being in line with the upstream cylinder due to the smaller blockage in the channel.

Acknowledgement
This research was supported by IIUM Research Acculturation Grant Scheme (IRAGS180530054).

References

[1] Tassone, Alessandro, Gianfranco Caruso, Fabio Giannetti, and Alessandro Del Nevo. "MHD mixed convection flow in the WCLL: Heat transfer analysis and cooling system optimization." *Fusion Engineering and Design* 146 (2019): 809-813. https://doi.org/10.1016/j.fusengdes.2019.01.087
[2] Müller, Ulrich, and Leo Bühler. *Magnetofluiddynamics in channels and containers*. Springer Science & Business Media, 2013.
[3] Singh, Ranjit J., and Trushar B. Gohil. "The numerical analysis on the development of Lorentz force and its directional effect on the suppression of buoyancy-driven flow and heat transfer using OpenFOAM." *Computers & Fluids* 179 (2019): 476-489. https://doi.org/10.1016/j.compfluid.2018.11.017
[4] Umeda, Naotaka, and Minoru Takahashi. "Numerical analysis for heat transfer enhancement of a lithium flow under a transverse magnetic field." *Fusion Engineering and Design* 51 (2000): 899-907. https://doi.org/10.1016/S0920-3796(00)00424-5
[5] Aylı, Ece, and Özgür Bayer. "Optimization of Vortex Promoter Parameters to Enhance Heat Transfer Rate in Electronic Equipment." *Journal of Thermal Science and Engineering Applications* 12, no. 2 (2020): 021001. https://doi.org/10.1115/1.4043994
[6] Mück, B., C. Günther, U. Müller, and L. Bühler. "Three-dimensional MHD flows in rectangular ducts with internal obstacles." *Journal of Fluid Mechanics* 418 (2000): 265-295. https://doi.org/10.1017/S0022112000001300
[7] Hussam, Wisam K., Mark C. Thompson, and Gregory J. Sheard. "Dynamics and heat transfer in a quasi-two-dimensional MHD flow past a circular cylinder in a duct at high Hartmann number." *International Journal of Heat and Mass Transfer* 54, no. 5-6 (2011): 1091-1100. https://doi.org/10.1016/j.ijheatmasstransfer.2010.11.013
[8] Bhuyan, Pranjal Jyoti, and Kalyan S. Goswami. "Effect of magnetic field on MHD pressure drop inside a rectangular conducting duct." *IEEE Transactions on Plasma Science* 36, no. 4 (2008): 1955-1959. https://doi.org/10.1109/TPS.2008.927136
[9] Hussam, Wisam K., and Gregory J. Sheard. "Heat transfer in a high Hartmann number MHD duct flow with a circular cylinder placed near the heated side-wall." *International Journal of Heat and Mass Transfer* 67 (2013): 944-954. https://doi.org/10.1016/j.ijheatmasstransfer.2013.08.081
[10] Hamid, Ahmad HA, Wisam K. Hussam, Alban Pothérat, and Gregory J. Sheard. "Spatial evolution of a quasi-two-dimensional Kármán vortex street subjected to a strong uniform magnetic field." *Physics of Fluids* 27, no. 5 (2015): 053602. https://doi.org/10.1063/1.4919906
[11] Thompson, Mark C., Thomas Leweke, and Michel Provansal. "Kinematics and dynamics of sphere wake transition." *Journal of Fluids and Structures* 15, no. 3-4 (2001): 575-585. https://doi.org/10.1006/jfls.2000.0362
[12] Mittal, S., and Vinod Kumar. "Flow-induced oscillations of two cylinders in tandem and staggered arrangements." *Journal of Fluids and Structures* 15, no. 5 (2001): 717-736. https://doi.org/10.1006/jfls.2000.0376
[13] Borazjani, Iman, and Fotis Sotiropoulos. "Vortex-induced vibrations of two cylinders in tandem arrangement in the proximity-wake interference region." *Journal of Fluid Mechanics* 621 (2009): 321-364. https://doi.org/10.1017/S0022112008004850
[14] Mathupriya, P., L. Chan, H. Hasini, and A. Ooi. "Numerical study of flow characteristics around confined cylinder using openFOAM." *International Journal of Engineering and Technology* 7, no. 4 (2018): 617-623. https://doi.org/10.14419/ijet.v7i4.35.22925
[15] Sommeria, JoëL, and René Moreau. "Why, how, and when, MHD turbulence becomes two-dimensional." *Journal of Fluid Mechanics* 118 (1982): 507-518. https://doi.org/10.1017/S0022112082001177

[16] Dousset, Vincent, and Alban Pothérat. "Numerical simulations of a cylinder wake under a strong axial magnetic field." *Physics of Fluids* 20, no. 1 (2008): 017104. https://doi.org/10.1063/1.2831153

[17] Burr, Ulrich, L. Barleon, U. Müller, and A. Tsinober. "Turbulent transport of momentum and heat in magnetohydrodynamic rectangular duct flow with strong sidewall jets." *Journal of Fluid Mechanics* 406 (2000): 247-279. https://doi.org/10.1017/S0022112099007405

[18] Yoon, H. S., H. H. Chun, M. Y. Ha, and H. G. Lee. "A numerical study on the fluid flow and heat transfer around a circular cylinder in an aligned magnetic field." *International Journal of Heat and Mass Transfer* 47, no. 19-20 (2004): 4075-4087. https://doi.org/10.1016/j.ijheatmasstransfer.2004.05.015

[19] Umair, Siddique Mohammed, Abdulrahman Alrobaian, Sher Afghan Khan, Marthande Gnanagonda Kashinath, and Patil Rajesh. "Numerical Investigation of Critical Range for the Occurrence of Secondary Peaks in the Nusselt Distribution Curve." *CFD Letters* 10, no. 2 (2018): 1-17.

[20] Siddique, Umair, Emaad Ansari, Sher Afghan Khan, and Rajesh Patil. "Numerical Investigation of Semi-Empirical Relation Representing Standard Deviation in Nusselt Profile Due to Water Jet Impingement." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 65, no. 1 (2020): 42-53.

[21] Pothérat, Alban, Joël Sommeria, and René Moreau. "Effective boundary conditions for magnetohydrodynamic flows with thin Hartmann layers." *Physics of Fluids* 14, no. 1 (2002): 403-410. https://doi.org/10.1063/1.1423287

[22] Hussam, Wisam K. Mark C. Thompson, and Gregory J. Sheard. "Enhancing heat transfer in a high Hartmann number magnetohydrodynamic channel flow via torsional oscillation of a cylindrical obstacle." *Physics of Fluids* 24, no. 11 (2012): 113601. https://doi.org/10.1063/1.4767515