ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

NUMERICAL STUDY OF HEAT DISSIPATION FOR FLOW OVER PERFORATED HOT CYLINDER WITH DIFFERENT HOLES ANGLE AT DIFFERENT MATERIALS

NAGHAM YASS KHUDAIR [1], AYAD MOHAMMED NATTAH [2], RAFEL H. HAMEED [3] and HAYDER K.RASHID [4]

[1]University of Babylon / College of Materials Engineering/Metallurgy Department. Babylon, Iraq.
Email: naghamalbakry@gmail.com
[2]University of Babylon / College of Materials Engineering/Metallurgy Department. Babylon, Iraq.
Email: ayad.natah@uobabylon.edu.iq
[3]University of Babylon / College of Engineering / Mechanical Engineering Department. Babylon, Iraq.
Email: eng.rafel.hekmat@uobabylon.edu.iq
[4]University of Babylon / College of Materials Engineering/ Ceramic Engineering Department, Babylon, Iraq.
Email: mat.hayder.k@uobabylon.edu.iq

Abstract

The operation of many industrial devices can be improved by increasing the disposal of generated heat. The perforated cylinder can be used in many engineering application as heating pipe and perforated rib. For this reason, the heat dissipation from perforated hot tubes improving by presented a new shape. There are mutually effects between pressure fluctuations, temperature and fluid flow velocity vectors for choosing a suitable turbulent mode that presented the heat perforated tube. Moreover, the shape of hot tube and its holes plays a great rule for improving and increase the heat loss to fluid that flow through it. The turbulent model can be used to illustrate the effect of dimensions and angle of holes for perforated cylinder. The K-ε turbulent model at high Reynolds number with induced the wall function that presented the effect of holes shape on improving fluid flow behavior at entrance and exit holes regions. The numerical analysis is used to investigate the improvement in thermal translated with annuals space of hollow cylinder affected by changing the arcs and angles of the side holes of the heat pipe at different Reynolds number. The ANSYS 16.1 software used to simulate the 2D geometry of fluid flow over and through perforated heat tubes. Building a simulation turbulent model by (Ansys-Fluent 16.1) using the circular cross section with symmetrical holes at angles (20^o, 30^o and 40^o) with high Reynolds number in real dimensions. The results show that the entrance arc helps flow to continue through the inside space of tubes taking high amount of Excess heat energy. Moreover, the type of materials has a good effect on improving Nusselt number that proportional to the material's conductivity.

Keywords: Perforated Cylinder, Heat Dissipation, Control Boundary Layer, Stream Lines.

NOMENCLATURE

$U\infty$ Free stream velocity (m/s)
T Temperature (K)
Re Reynolds number (-)
Nu_L Local Nusselt number (-)
Nu_v average Nusselt number (-)

Greek symbols

ΔP: pressure drop across the tube, Pa.

ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

μ: dynamic viscosity, kg/m s.
ρ : Flow density, kg/m^3.
α: Thermal diffusivity
ν: kinematic viscosity m^2/s

INTRODUCTION

Heat transfer and flow mechanics in the annulus spaces play an important role in many essential engineering applications. The combination of the two phenomena in closed spaces gives strong non-linear attitude [1]. Constants related to the heat transfer such as Nusselt, Reynolds and Prandtl numbers in addition to the case design are important parameters in heat transfer. On the other side, velocity and pressure are other variables that affect flow mechanism. Study all the preceding variables in an enclosed regenerator would be of interest for increasing the performance of the regenerator. Heat transfer and flow mechanisms in the annulus spaces have been the topic of many pioneer investigations due to its essential engineering applications. Such tools into enclosed spaces give strong non-linear attitude due to the efficient coupling between the flow and heat equations. The numerical simulation can be used to investigate the effect of perforated cylinder on enhancement the flow dissipation from hot cylinder. Where, Ziyan et al. [1] presented the numerically investigated of the fluid flow and heat transfer of a fully developed Taylor-Couette flow in concentric and eccentric annuli with a stationary adiabatic outer cylinder and a stationary or rotating isothermal inner cylinder. The effect of rotational speed and eccentricity on axial and tangential velocity distribution, pressure drop and forced convection heat transfer were conducted for different values of radii ratios, Reynolds number, Taylor number, and Prandtl number. The Couette-Poiseuille flow, in concentric and eccentric annuli with a rotating inner pipe, is examined numerically by Ameen et al. [2] presented the leverage of eccentricity on fluid flow in a medium in small range of (0.3-0.6) and large ratio of radii of (0.6-0.8). Moreover, they studied the influence of eccentricity on the flow structure, pressure and velocity at selected radii ratios. They pointed that the pressure drop decreases impressively with increasing eccentricity, but, the effect of eccentricity on the pressure drop is minimal at low value of radius ratios. The velocity distribution on the axial and tangential the effect of eccentricity is effective, in the small space of the eccentric annulus. A decrease in axial velocity corresponds to an increase the tangential velocity as the eccentricity increased. The heat transfer and Taylor-Couette flows in a horizontal annular gap between finite coaxial cylinders in the configuration of rotor-stator are elaborated numerically by Chaieb et al. [3]. Finite volume methods with numerical schemes of second order space and time accuracies were used. The heat transfer and flow characteristics which promote around a heated, rotating vertical cylinder enclosed within a stationary concentric cylinder were conducted numerically by Ball and Farouk [4]. A tall annulus, radius ratio and adiabatic horizontal end-plates were fixed. They examined the effect of buoyancy forces by heating the inner cylinder on the development of the Taylor vortex flow. They found that the formation of Taylor vortices in an isothermal flow was late until the rotational parameter $\delta = Gr/Re^2$ has a value below unity for any value of Reynolds number above its critical value. On the other hand, the Taylor cells first appear at the top of the annulus. Also, the rotational effects on the mixed convection for low fluids of Prandtl number enclosed between the annuli of concentric and eccentric cylinder on

The Seybold REPORT

ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

a horizontal position studied numerically by Lee [5]. He showed that the mean Nusselt number raise with excess Rayleigh number for both concentric and eccentric fixed inner cylinders. A considerable effect of the rotational Reynolds number at lower Prandtl numbers on the flow patterns was observed, as well as, the mean Nusselt number value was unchanged with respect to the rotational Reynolds number. Where, Nair et al. [6] studied the natural and mixed convection heat transfer in an annulus enclosed by concentric cylinders, the inner cylinder is either rotating at different angular speed or stationary or the outer cylinder is the enclosure. The factors that have been studied are rotational Reynolds number (Re), Also, they presented radial velocity component variation along a particular region in the domain. Large eddy simulations of the heated flow in an annular gap with interior cylinder rotation were elaborated by Schneider et al. [7] for Reynolds number 9000 and two values of the rotation number 0.21 and 0.858. Rectangular, closed and open trapezoidal were the selected geometry of the cavities, they used Renormalization group (RNG) k–ε model for Reynolds and Taylor numbers in the range of $5 \times 10^3 < Re < 6.5 \times 10^4$ and $160 < Ta < 1900$ respectively. In the open trapezoidal shape, β > 90∘, the more mixing happen and the temperature is more influenced via the fluid amidst the two cylinders. The flow and heat characteristics of fluid under turbulent flow through a tube of heat exchanger fitted with perforated hollow cylinder inspected by Nakhchi and Esfahani [9]. The effect of the index of perforated, diameter ratio of the hollow cylinder, and Reynolds number on the heat transfer enhancement and thermal performance were studied. Their results indicated that the resistance of flow can be minimized up to 86.2% with raising the perforated index. Also they studied the fluid mixing between the tube walls and the core region. Thus the rate of heat transfer to be best due to the demolition of the boundary layer that caused by perforated hollow cylinder. The Taylor-Couette flow with an impervious outer fixed cylinder and porous rotating inner one were predicted by Mochalin et al. [10]. The rotating porous cylinder is included into the computational domain and the flow of liquid inside of it is intent. They discussed the effect of the permeability of the rotating inner cylinder and the ratio of permeate-retentate onto the boundary of centrifugal stability and supercritical flow details. They found that the filtration velocity becomes dispense irregularly along the porous cylinder when its hydraulic resistance is lower than a definite value. After some critical threshold, the flow structure drastically changes, and spiral vortices appears, which crosses the porous rotating cylinder back and forth several times, providing multiple flow recirculation through the porous cylinder. A Taylor-Couette-Poiseuille flow in an annular channel of a slotted rotating inner cylinder, corresponding to a salient pole hydrogenerator studied numerically by Lancial et al.[11] to get better conception of flow and thermal phenomena in electrical machines. They performed to investigate all main flow regimes and to derive correlations in terms of the Nusselt number distribution on the side and face of the rotor pole. They showed that the Nusselt number is proportionate to the tangential Reynolds number to the power 1/7 in the inductive and pole faces trailing side, this relation was identical to the one encountered in classical Taylor-Couette-Poiseuille flows between two smooth and concentric cylinders. The conduct of engine oil flow of convective heat transfer through a concentric and an eccentric annulus created between inner heated rotating cylinder and outer cooled stationary cylinder were carried out numerically by Abed et al. [12].On the other hand, Ozkan et al. [13] analyzed experimentally the flow within the annular region of a shrouded cylinder and the control of the

ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

unsteady vertical flow generated by the inner cylinder. A splitter plate was fixed in the annular region and the angle of the plate was hanged with respect to the centerline of cylinder. The splitter plate angle range of $60^o \leq \theta \leq 180^o$ with an increment of 30^o. For high entrainment of fluid through the cylinder, the value of porosity selected was 0.7. They showed that the splitter plate located in the annular region of shrouded cylinders was efficient on decreasing the levels of turbulence just behind the base of cylinder, and the near wake of the perforated shroud. The control of unsteady vertical flow happened downstream a cylinder of circular cross section located in little water flow using concentrically located outer perforated cylinder were examined experimentally by Pinar et al. [14]. The effects of porosity and diameter ratio, Di/Do were illustrated, constant level of the water and free stream velocity were used. They observed that the inner circular cylinder was highly affected by the existence of surrounding outer perforated cylinders and the diameter ratios (Di/Do) does not have a significant influence on the flow structure for region between the inner and outer perforated cylinder unlike downstream of concentric cylinder. Chithrakumar et al. [15] presented experimentally the convective heat transfer from the inner fixed heated cylinder of a vertical concentric annulus with outer rotating cylinder for various range of rotational speed. They indicated that as Rotation parameter increase there was an obvious enhancement in heat transfer. Borujerdi and Nakhchi [16] studied experimentally the influence of depth of slot to the ratio of width, rotary motion and access velocity on Nusselt number and friction factor in an annular flow between two concentric cylinders with fine and slotted surface. The thermal fatigue of the inner and outer cylinder on the rotating machine investigated experimentally by. Shiina [17] obtained that the heat transfer coefficient independent on the axil flow rate but the speed of inner rotating cylinder has a strong influence on it. They proposed empirical correlation to compute the heat transfer coefficient for the gap between the inner and outer cylinder. Enhancement force convection into vertical annuli using porous media were performed numerically and experimentally by Yousif et al. [18]. A three- dimensional numerical simulation was conducted using ANSYS Fluent software, utilizing water for various range of Reynold number and extensive range to the heat flux of the wall. Their experimental work consistence an annular tube be composed of two concentric cylinders, three values of diameter of porous media and porosities were used to study the flow and heat characteristic of the annular tube. They specified that the value of Nusselt number increased with decreased porosity, the best Nusselt number gated at lower value of porosity was used. Experimental and numerical studies were conducted by Chithrakumar et al. [19] to comprehend the convective heat transfer in a vertical annular gap created between a heated stationary inner cylinder and a rotating cold outer cylinder. Experimental work was done at the values of aspect ratio and radius ratio remains constant and large values of rotational parameter between 0 and 526 at different heat loads. The numerical study was carried out on the same parameters that were used in the experimental side. The effect of rotating outer cooled cylinder on heat transfer and fluid flow for a horizontal annulus filled with air were carried out experimentally and numerically by Khalaf et al. [20], the inner cylinder was subjected to a uniform heat flux and kept fixed. Reynold's number, Rayleigh number, and constant radius ratio and aspect ratio were studied on their work. They showed that the average heat transfer coefficient was affected by increasing Rayleigh number more

ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

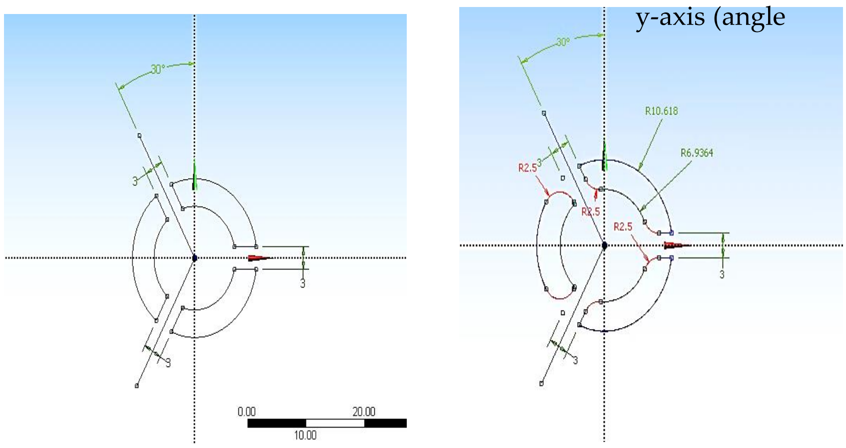
than rotational Reynolds number. As well, they deduced empirical correlations of the average Nusselt number for free and mixed convection.

The aim of the present study is to increase the transfer rate from hot tubes by introduced new shape. The new shape represented by introduced hollow perforated hot tube in front of flow. The tube shape under investigation presented innovate perforated hole dimensions which is improved by adding new arc and investigate it. The directly simulated flow and heat translated through perforated heat tube based on the K-ε turbulent model at high Reynolds number with induced the wall function. The energy equation has also been dealt with to see the improvement in getting rid of excess thermal energy due to the new shape. However, the cavity has a wide effect on flow type due to shear layer oscillation [21] and [22]. The open cavity shape will controlled to prevent this problem for obstruction of flow at entrance or exit regions with hollow cylinder will be shown in elaborated articles.

Geometry Building

The model of 2D is drawn using of the ANSYS16.1 software and with the specified dimensions. The flow is come from left to right at different Reynolds number. Various angles of heat perforated tubes cross-setions were tested and a circle tube cross section was selected for case simulation as shown in figure (1). As shown in this figure part a represent the circle tube section had Sharp edge hole. While, part b represent the circle tube section had Curved edge perforation at symmetrical angle 30°. The hole angles will changed from 20° until 40° and symmetrical with x=axis. Each hole has diameter 3 mm and the outer radius of cylinder 10.618 with inner radius equal to 6.9364 mm. The edge radius is 2.5 mm. The properties of the two materials that used in the numerical investigation, where are density, specific heat and thermal conductivity. The table (1) shows the materials properties that used in ANSYS16.1 software.

a



b

Figure 1: Model in 2D represent the section of perforated cylinder where (a) Sharp edge hole (b) Curved edge perforation at angle 30°

ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

Table 1: Illustrates The Materials Properties

Table 3. 1

Materials	Density (ρ) kg/m³	Specific heat (Cp) J/kg.k	Thermal conductivity (K) w/m.k
Aluminum	2719	871	202
Copper	8978	381	387.6

Meshing

In order to obtain a high accuracy numerical solution of flow over circular cylinders with holes. The mesh must be satisfied the result required from simulation depends on the effect of the turbulent model. However, it must be divided into very small parts called elements and they are surrounded by nodes. The above partition process is called mesh. Make mesh with high accuracy to get good and perfect results. The problem focuses on the open cavity that occurs especially at entrance as well as exit regions of perforated holes. Moreover, the annuals hot tube where, the may the major heat transfer happen between the hot tube surface and circulated flow over it. The mesh of hot tubes with symmetrical hole and fluid flow street can be shown in figure (2).

After completing the work of the shape of the contact regions and surfaces named of the solids, the mesh is applies and adjusts the program options as follows:

1) Total number of the mesh elements (21312) and the total number of nodes (10946).
2) Uniform free stream are applied at the inlet boundary from left and exit is treated right at atmospheric pressure.

The numerical solution is presented by using ANSYS 16.1 program software solved with (VOF) model. The Volume of Fluid Model (VOF) can be used to simulate the heating fluids by solving set of momentum equations and tracking energy equation for fluids throughout the domain.

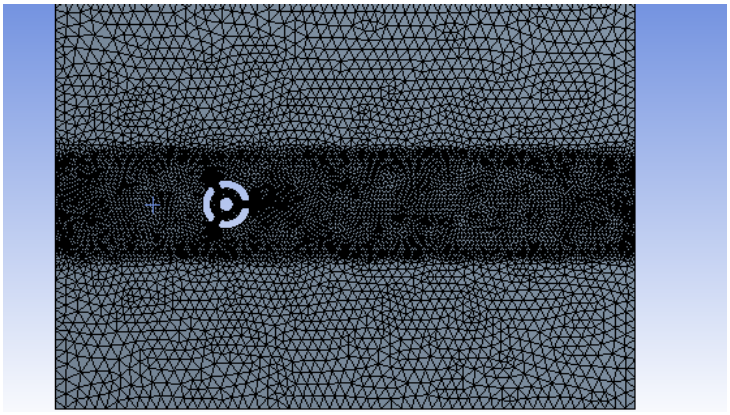


Figure 2: Mesh generation of flow over and inside hot perforated cylinder and the wake street

The Seybold REPORT
ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

Governing Equations

In order to model the turbulent flow field throughout and over hot perforated cylinder, averaged Navier-Stokes equations have been used depending on VOF model with turbulent model that called The K-ε at high Reynolds number with induced the wall function. The energy equation has also been dealt with to see the improvement in getting rid of excess thermal energy due to the new shape with controlling heat transfer. The flow at entrance region of perforated will induced circulation and vortex due to pressure fluctuation due to generate open cavity. There for some equation that deals with open cavity will introduce with solutions [23].

Contiuity equation

$$\nabla.\rho U = 0 \qquad \ldots (1)$$

Momentum equation

$$\rho U.\nabla U = -\nabla P + \mu \nabla^2 U \qquad \ldots (2)$$

Energy equation

$$\nabla.\rho UCpT = \nabla.(\alpha \nabla T) \qquad \ldots (3)$$

Modified K-ε Model

By assumption that the Reynolds number is high enough, the k-ε turbulence model can be used with coupling of pressure gradient and heat effect on the enhancement of wall function treatment. Moreover, it is important that the turbulence is in equilibrium in boundary layers, taking the effect of entrance and exit regions of perforated holes. This model used to treatment the swirl effect due to obstruction at the sharped and curved end illustrated its effect on heat transfer rate.

$$\frac{\partial(\rho k)}{\partial t} + \nabla.(\rho u k) = \left(\mu + \frac{\mu_T}{\sigma_k}\right)\Delta k + G - \rho\varepsilon \qquad \ldots (4)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla.(\rho u \varepsilon) = \left(\mu + \frac{\mu_T}{\sigma_\varepsilon}\right)\Delta \varepsilon + C\varepsilon 1 \frac{\varepsilon}{k} G - C\varepsilon_2 \rho \frac{\varepsilon^2}{k} \qquad \ldots (5)$$

Where k is the turbulence kinetic energy, is the dissipation rate and "G" denotes the production of turbulence kinetic energy. featuers above tubes will be presented as follow The assumption is the same for all parts in this work and can be summerized as below:

1) Water flow is steady acrossing hot pipe.
2) water velocity is uniform at inlet side before the hot pipe, moreove its turbulent flow deals with open cavity when flow across outer surface of perforated holes at hot pipe.
3) The heat flux will applied at the pipe surface only.
4) The perforated hole have equal cross sectional area with diameter equal 3mm.

The Seybold Report
ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

5) The surfaces of all inner location were specified as no-slip and with high y+ wall treatment at vicinity wall modeling.

Since the flow have been obstructed by sharped edge the swirl will perfomed and the turbulent boundary layer separation are indispensable. Consequently, the K- ε must be introduced with enhancement of wall effect [26].

The boundary flow conditions represented by water flow velocity and the geometry characteristics of each situation under consideration of new angle of holes. Moreover, the mutual influence with thermal effects that provided by heat flux located along the inner and outer surface of perforated cylinder are simulated numerically by the software ANSYS 16.1.

The Reynolds number can be give by

$$Re = \frac{\rho V D_h}{\mu} \quad \ldots(6)$$

Where D_h is haydrolic diameter and μ is dynamic viscosity,.

The Nusselt Number can be given by

$$Nu = \frac{h\, D_h}{K} \quad \ldots(7)$$

And the heat trasfer coefficient can be given by

$$h = \frac{1}{A_s}\int^{A_s} \frac{Q}{T_w - T_a} dA \quad \ldots(8)$$

Where T_a represent the water flow temperature in vinicity region to the hot perfotayed tubes surfaces and A_s is local tubes surfces area and T_w is the averaged inner surface tubes wall temperature. Prandtl number is a dimensionless quantity that puts the viscosity of a fluid in correlation with the thermal conductivity.

$$Pr=\frac{\nu}{\alpha} \quad \ldots(9)$$

RESULTS AND DISSCUSIONS

Validation

The result of velocity contour for flow over rotating cylinder is used for validation the results figure (4). This situation compared with the results of A. Tafuni et al [24] figure (3). As shown in these figures, the agreement is very well. The present work depends on the k-e model at high Reynolds number. As shown in this figure the separation will occur slower and symmetry from top and bottom cylinder wall in the assumption of high Reynolds number.

The Seybold REPORT
ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

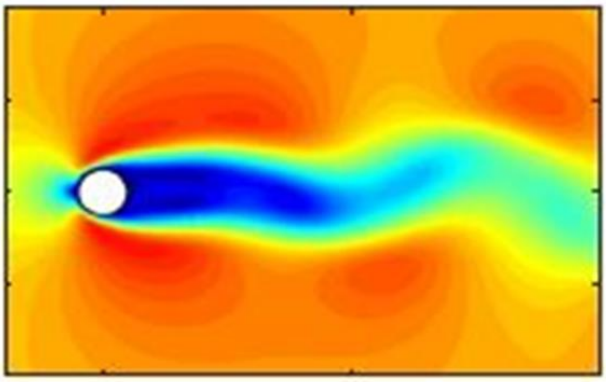


Figure 3: velocity contour over rotating cylinder A. Tafuni et al [24]

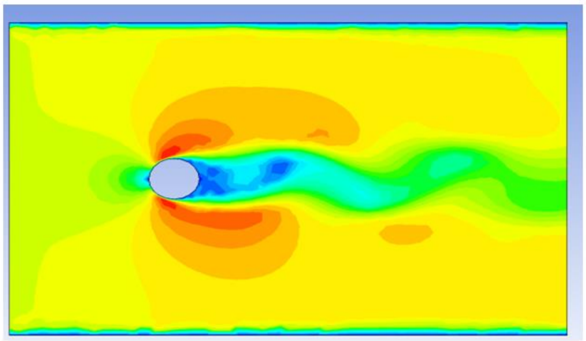


Figure 4: velocity contour over rotating cylinder present work

The numerical solutions and simulation is carried out for several situations to obtain the best condition in terms of heat transfer from the perforated cylinder to the flow. However, the perforated tube represents the hot part that needed to be cooler. In this manuscript the fluid push through the annulus region of hot cylinder. At first the geometry of perforate holes consist of sharp leading and sharp trailing edge as shown in figure (1) part a. The re-circulation region and vortex near the trailing edge of the open cavity observed in all perforated holes as shown in figure (5).Also, some place seems to become stronger. The vortex wills oscillation the shear layer due to pressure and velocity fluctuation through it. This vortex will operate as an obstruction of flow to be inlet to annulus hot hollow cylinder. Furthermore, the increasing in fluid velocity increases the fluid energy casing a stronger vortex as shown in figure (5). The area of open cavity is important factor in dealing and controlled the flow properties [25]. The sharp leading and trailing punching edge will produce a special case of temperature, pressure and velocity distribution over and in front of hot cylinder as shown in figure (6). The sharp edges give more disturbances and low heat and mass translated through the perforated. Consequently, the process of getting rid of excess heat will worsen with the increase in the speed of fluid flow due to the formation of vortices that impede the entry of water into the hot cylinder, in addition to the expenditure of more energy.

ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

Re=8131.5 Re=13552.5 Re=18973.5

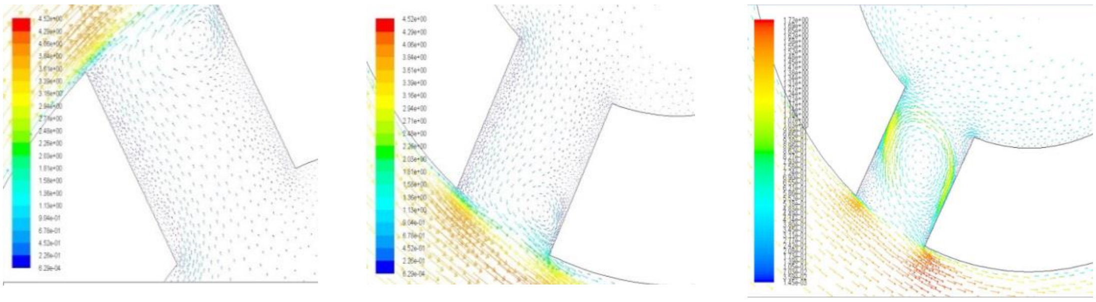


Figure 5: Model in 2D represent the flow over perforated cylinder with sharped edge

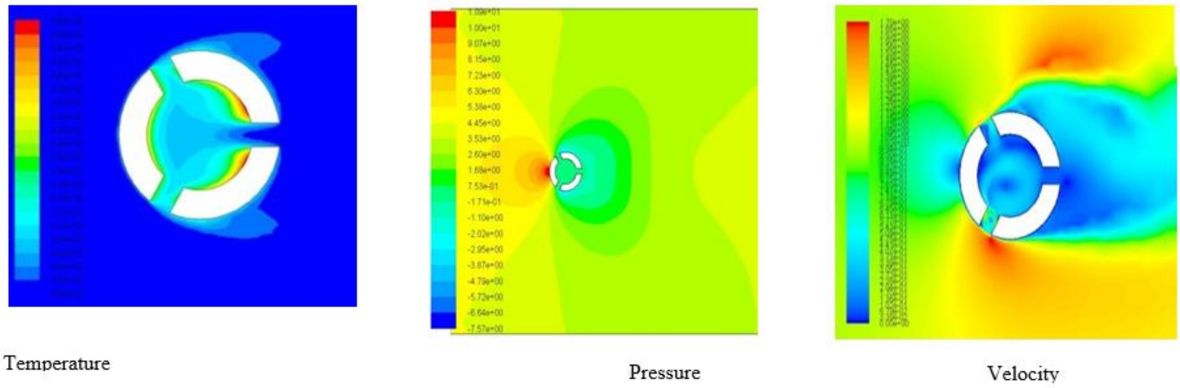


Figure 6: Model in 2D represent the flow over perforated hot cylinder where with sharped edges and its effect on temperature, pressure and velocity counter distribution

For improving the heat and mass transfer through the hot perforated tube, the dimension of holes were changed as well as its angle compare with flow direction. Where, the weakness of the flow through the holes in the hot cylinder was addressed by improving the entry areas to this cylinder by placing arcs with appropriate dimensions (Radius=2.5 mm) in addition to keeping the sharp edges to enhance the pressure differences leading in opposite side of flow at entry of the fluid within the confined area inside the cylinder see figure (2). Figure (7) represent the stream lines in 2D represent the flow over perforated cylinder at angles 20, 30 degree. While, figure (8) refer to pressure distribution in 2D represent the flow over perforated cylinder at angles 20, 30 degree. From the previous two figures, its clearly notice the great improvement in the process of entering the cold fluid through the side holes of the hot cylinder, which allows the process of heat exchange to be happened between the fluid and the hot solid surface from the inside. If look carefully at the flow from left to right and with a rotational movement over the hot cylinder to reach the leading edge of hole, the new arc will allows the fluid to enter from the left side with respect to the hole, and the sharp part on the right side hinders the part of fluid from continuing to advance around the outside part of the cylinder. This effect is very clear shown with pressure distribution shown in figure (8). For allowing for fluid to flow easily, the locations of the brackets were changed oppositely at the end of the hole leading to the inside

ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

of the cylinder. The process of rotating inside the cylinder increases the time of heat exchange between the outer surfaces of the cylinder with the fluid inside it to reach the ideal state. The process of rotating the fluid inside the cylinder requires a large power machine, and therefore it will fail in the task required of it if the amount of fluid entering from the holes is small. The temperature distribution and its effect with the new shape of leading and trailing edge of holes can be shown in figure (9). In order to confirm the large changes resulting from the unique shapes that were chosen as holes see figures (10) and (11).As shown in these figures, it is possible to note the change in the direction and velocity amounts of the internal flow of the hot perforated cylinder. Moreover, the flow behavior around the outer surface of the cylinder, and the change is tracked within the course of the fluid outside and away from this cylinder. The temperature of the surface of the cylinder has a great influence on the properties of flow, especially the viscosity, which has a major role in changing pressure, speed, and the occurrence of vortices. However, it was observed that the effect of temperature decreases as the speed increases this mean increases Reynolds number. The results also showed the significant effect of the angles of the holes on the amount of fluid entering the hot cylinder, which indicates that the greater the angle of inclination of the holes will improved the heat transfer process.

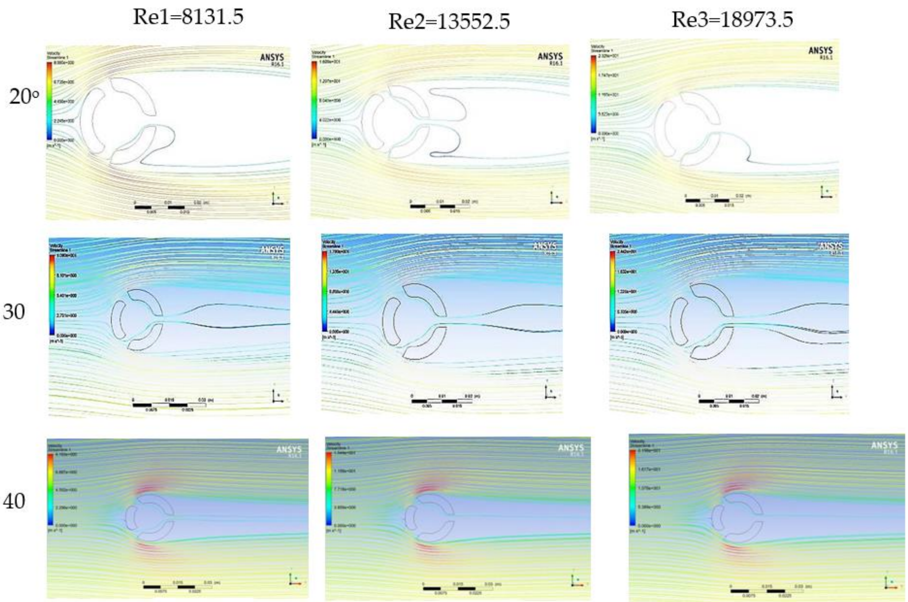


Figure 7: stream lines in 2D represent the flow over perforated cylinder at angles 20, 30, and 40 degree

The Seybold REPORT
ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

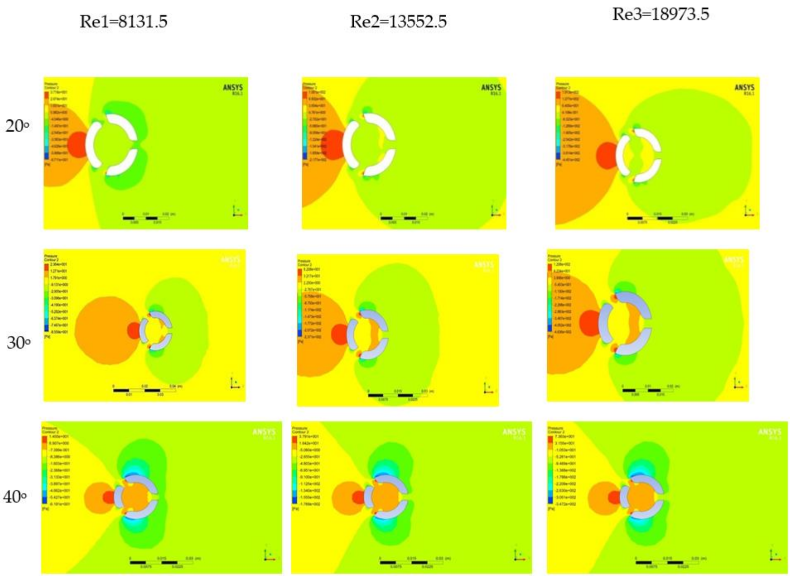


Figure 8 : Pressure distribution in 2D represent the flow over perforated cylinder at angles 20,30 and 40 degree

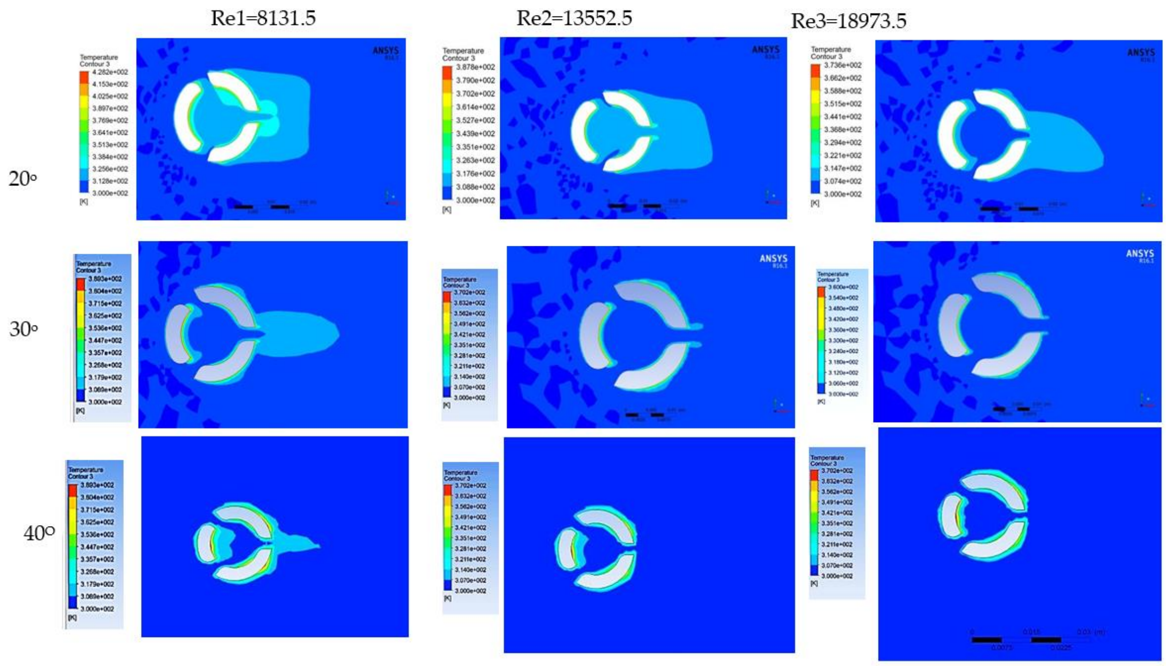


Figure 9: Temperature distribution in 2D represent the flow over perforated cylinder at angles 20,30 and 40 degree

ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

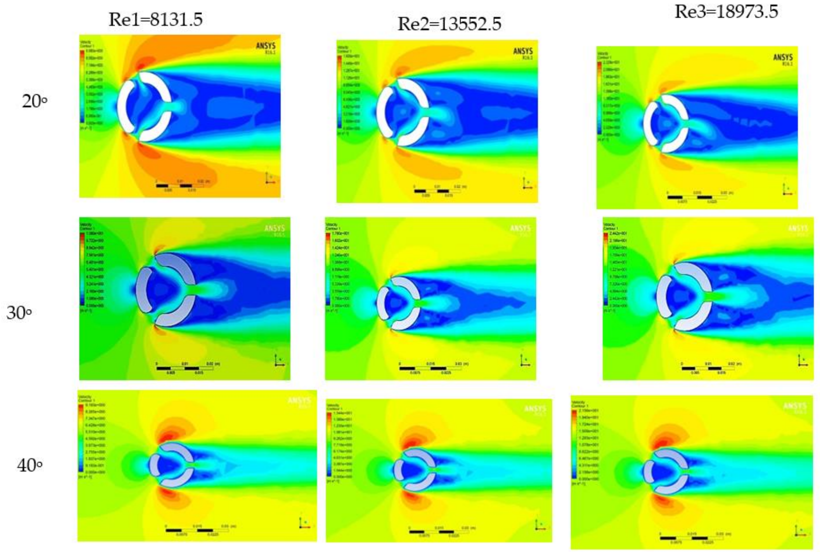


Figure 10: Velocity contour in 2D represent the flow over perforated cylinder at angles 20,30 and 40 degree

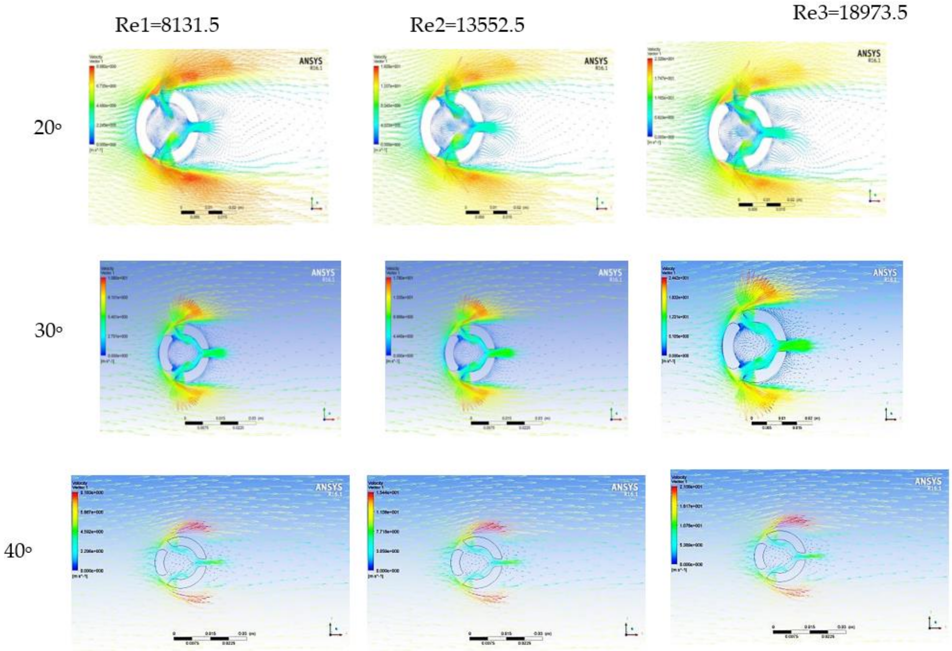


Figure 11: Velocity vector in 2D represent the flow over perforated cylinder at angles 20,30 and 40 degree

From the above figures the maximum heat transfer rate occurs at angle of 40^{O} of perforated hole. The numerical results for the effect of the flow characteristics inside hot perforated tube

ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

on heat transfer rate at angle of 40^{O} of perforated hole can be seen in figure 12 and 13. Where, these figures carried out the relation between location and Nu ratio. The location represented by angle starting from middle point and increased with positive value to the right and negative value to left. As shown in these figures the Nu number increase as the Re number increase. Since, the increasing in fluid velocity making the flow ability to draw the hot fluid with it and enter a colder fluid, generating a larger temperature difference, which increases the heat exchange process, producing a greater value of Nu number. Moreover, the figure (12) represents the hot cylinder for Aluminum materials compared with figure (13) for Copper materials in terms of heat transfer rate represented by Nu number. The comparison between these figures, the heat transfer rate is very clearly bigger with copper hot perforated tube and more effective compare with that of the aluminum.

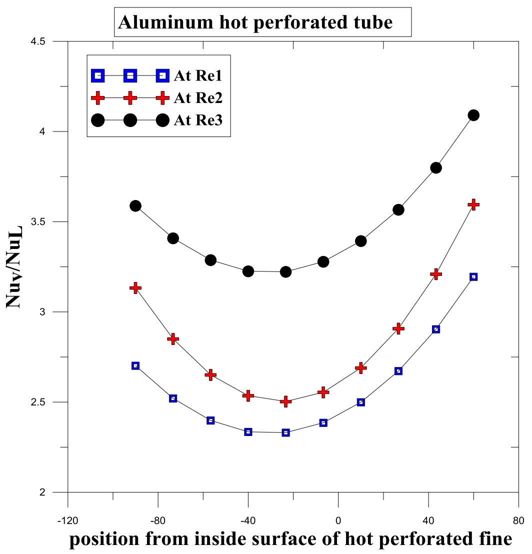


Figure 12: Relation between Nu and positions of inside Aluminum hot cylinder

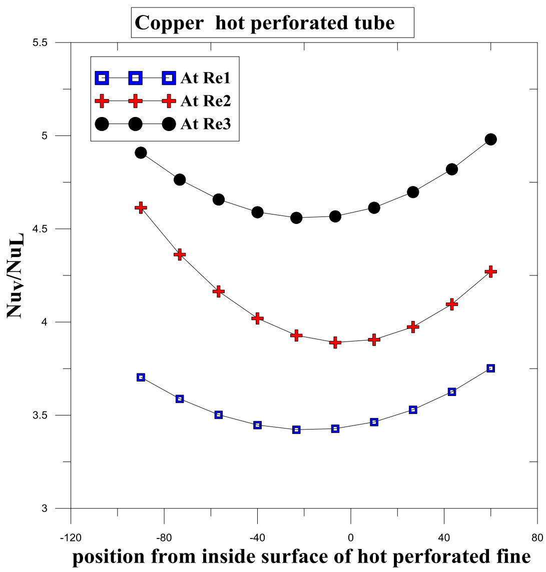


Figure 13: Relation between Nu and positions of inside Copper hot cylinder

CONCLUSIONS

Numerical calculations of the improvement in heat transfer from a perforated cylinder showed many distinctive results. These results are related to a number of factors, including the shape of the hot body, the properties of the fluid used, and the mathematical funds required to achieve the optimal simulation to show the changes that occurred and the update provided.

ISSN 1533-9211

DOI 10.17605/OSF.IO/RP8FV

The conclusions can be summarized as below:

1. The K- ε model for turbulent flow at high Reynolds number is more accurate for with heat dissipation.
2. The streamline distribution gives a sufficient view for effect of shape on pressure and velocity fluctuation.
3. The temperature has wide effect on fluid property, but this effect reduces as Reynolds number increase.
4. The angle of perforated holes has distinguished advanced for improving heat transfer rate.
5. The open cavity phenomenon can be generating types of vortices of the flow which is a kind of obstruction to the flow through it.
6. The dimension of narrow paths plays a major rule for controlling of flow type.
7. The heat transfer rate increase with increase the value of conductivity of materials.

References

1) H. Abou-Ziyan, R. Ameen, K. Elsayed, Fluid flow and convection heat transfer in concentric and eccentric cylindrical annuli of different radii ratios for Taylor-Couette-Poiseuille flow, Advances in Mechanical Engineering 13 (8) (2021) 1-22.

2) R. Ameen, K. Elsayed, A. Helali, H. Abou-Ziyan, Effect of eccentricity on couette-poiseuille flow in stationary and rotating annular space of different radii ratios, 17th International Conference on Aerospace Sciences & Aviation Technology (ASAT-17-109-PP) (2017).

3) I. Chaieb, T. Boufendi, X. Nicolas, Taylor-Couette flow with mixed convection heat transfer and variable properties in a horizontal annular pipe, Thermal Science, VINČA Institute of Nuclear Sciences (2021).

4) K. S. Ball, B. Farouk, on the development of taylor vortices in a vertical annulus with a heated rotating inner cylinder, International Journal for Numerical Methods in Fluids (7) (1987) 857-867.

5) T. S. Lee, Laminar fluid convection between concentric and eccentric heated horizontal rotating cylinders for low-prandtl-number fluids, International Journal for Numerical Methods in Fluids (14) (1992) 1037-1062.

6) Y. Nair, V. Shakkottai, B. Prasad, Mixed convection heat transfer in an annulus with rotating inner cylinder, Proceedings of the ASME 2016 Heat Transfer Summer Conference.

7) M. Schneider, B.A. Younis, B. Weigand, Large-eddy simulations of flow and heat transfer in heated concentric annulus with inner cylinder rotation, International Journal of Heat and Mass Transfer 114 (2017) 1248–1262.

8) A. Nouri-Borujerdi, M.E. Nakhchi, Prediction of local shear stress and heat transfer between internal rotating cylinder and longitudinal cavities on stationary cylinder with various shapes, International Journal of Thermal Sciences 138 (2019) 512–520.

9) M.E. Nakhchi, J.A. Esfahani, Numerical investigation of heat transfer enhancement inside heat exchanger tubes fitted with perforated hollow cylinders, International Journal of Thermal Sciences 147 (2020) 106153.

10) L. Mochalin, J. Cai, S. E, V. Brazhenko, D. Wang, Numerical study of the flow through an annular gap with filtration by a rotating porous cylinder, Engineering applications of computational fluid mechanics 16 (1) (2022) 469–483.

ISSN 1533-9211
DOI 10.17605/OSF.IO/RP8FV

11) N. Lancial, F. Torriano, F. Beaubert, S. Harmand, G. Rolland, Taylor-Couette-Poiseuille flow and heat transfer in an annular channel with a slotted rotor, International Journal of Thermal Sciences 112 (2017) 92-103.

12) W.M. Abed, A. Al-Damook, W. H. Khalil, Convective heat transfer in an annulus of concentric and eccentric cylinders with an Inner rotating cylinder, International Journal of Heat and Technology 39 (1) (2021) 61-72.

13) G. Ozkan, T. Durhasan, E. Pinar, A. Yenicun, H. Akilli, B. Sahin, Control of the flow in the annular region of a shrouded cylinder with splitter plate, EPJ Web of Conferences 143, 02088 (2017).

14) E. Pinar, T. Durhasan, G. Ozkan, M. M. Aksoy, H. Akilli, B. Sahin, The effects of perforated cylinders on the vortex shedding on the suppression of a circular cylinder, EPJ Web of Conferences 143, 02094 (2017).

15) V. Chithrakumar, G. Venugopal, M. Rajkumar, Thermal behaviour of heated stationary inner cylinder of a concentric vertical annulus formed with rotating outer cylinder, IOP Conf. Series: Materials Science and Engineering 1114 (2021) 012050.

16) A. Nouri-Borujerdi, M. E. Nakhchi, Friction factor and Nusselt number in annular flows with smooth and slotted surface, Heat and Mass Transfer (2018).

17) K. Shiina, S. Nakamura, Y. Mizushina, Heat transfer characteristics of fluid flow in an annulus with an inner rotating cylinder having a labyrinth structure, Heat Transfer-Japanese Research 25 (2) (1996).

18) M. Yousif, S. Shehab, H. Jaffal, Effect of porous media on the performance characteristics of a concentric vertical annular tube, Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 75 (2) (2020) 94-112.

19) V. K. Chithrakumar, G. Venugopal, M. R. Rajkumar, Convection in vertical annular gap formed by stationary heated inner cylinder and rotating unheated outer cylinder, Heat and Mass Transfer (2019).

20) S. Khalaf, A. Mohammed, Q. Slaiman, Heat transfer characteristics in a horizontal annulus enclosure with uniformly heated stationary inner cylinder and rotating outer cylinder, Journal of Mechanical Engineering Research and Developments 43 (6) (2020) 39-49.

21) Abraham J. Meganathan, An Experimental Study of Low Speed Open Cavity Flows, Thesis, University of Tennessee, Knoxville,2000.

22) Omar LADJEDEL, ADJLOUT Lahouari, Tayeb YAHIAOUI and IMINE Omar , Heat Transfer on Two Grooved Cylinders In a Tandem Arrangement, Proceedings of the ASME 2014 International Mechanical Engineering Congress and Exposition IMECE2014,November 14-20, 2014, Montreal, Quebec, Canada.

23) Sinisˇa Bikic´ , Marko Đurd--evic´ , Masˇa Bukurov and Slobodan Tasˇin, Comparison of single-hole and multi-hole orifice energy consumption, Advances in Mechanical Engineering 2022, Vol. 14(1) 1–13._ The Author(s) 2022.DOI: 10.1177/16878140221075461.

24) A. Tafuni, J.M. Dominguez, R. Vacondio, A.J.C. Crespo, A versatile algorithm for the treatment of open boundary conditions in Smoothed particle hydrodynamics GPU models, Comput. Methods Appl. Mech. Engrg. 342 (2018) 604–624.

25) Jian Zhang, Qingshan Yang and Qiusheng Li , Simulation of Pressure–Velocity Correlations by Green's Function Based on Reynolds Stress Model, Academic Editor: Mikhail Sheremet, Symmetry 2022, 14, 1352. https://doi.org/10.3390/sym14071352.