ICMERE2019-PI-000

Effect of Rotation and Changing Vertical Positions for Conjugate Pure Mixed Convection in Square Cavity with Two Heat Conducting Rotating Cylinders

Ehsanul Azim[1,a*], Md. Jahid Hasan Sagor[1,b] and A.B.M.Rafiqul Hasan[1,c]

[1] Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Bangladesh
[a*]ehsanulazim050BUET@gmail.com
[b] jahidhasansagor96@gmail.com
[c]badhonbin@gmail.com

Abstract- *A numerical analysis of conjugate pure mixed convection inside a differentially heated square cavity with two vertically positioned heat conductive rotating cylinders has been performed using Finite Element Method. This study has been done assuming both top and bottom wall as insulated and the left vertical wall as isothermally heated and right vertical wall as isothermally cooled..The study has been analyzed for Richardson Number,Ri=1 and the range of Reynolds Number(Re) has been considered* $1 \le Re \le 10000$*.Besides,the speed ratio(Ω) of the cylinders has been considered* $\Omega = 1$*. The working fluid in the cavity is air (Prandtl number, Pr=0.70484). The results of this study have been investigated in terms of streamlines, isotherms, average Nusselt number .The results show that the directions of rotating cylinders and Reynolds number have contribution upon the average Nusselt number, flow pattern and isotherms and accordingly for counterclockwise-counterclockwise rotation of both cylinders best heat transfer phenomena have been observed. For this reason, the condition for best heat transfer phenomena in case of counterclockwise-counterclockwise rotation has been tested from variation of distance between to cylinders. These results have been shown in terms of average Nusselt number and Reynolds number*.

Keywords: Conjugate Pure Mixed Convection, Rotating Circular Cylinder, Enclosure, Vertical cylinder Locations, Finite Element Method.

1. INTRODUCTION

Recently, semiconductor industry has flourished greatly due to minimization of circuit design with the development of efficient cooling system. Air cooling is widely used due to easy maintenance and reliability. It is important to consider the effect of conduction heat transfer within a solid body when analyzing a mixed convection problem. Many authors have investigated the numerical study of mixed convection in various shaped cavities with rotating cylinders and those have been reported in literature.

Liao and Lin [1] numerically investigated natural and mixed convection within domains with stationary and rotating cylinder. The heat transfer quantities were found for $10^4 \le Ra \le 10^6$. Chatterjee, Gupta and Mondal [2] studied numerically mixed convective transport of Cu-H_2O in a differentially heated and lid-driven square cavity with a rotating cylinder. Their results show that heat transfer greatly depends on the rotational speed of the cylinder, mixed convective strength and nanoparticle concentration. Khanafer and Aithal [3] investigated the numerical study of mixed convection in a differentially heated cavity with two rotating cylinders for various pertinent parameters such as Richardson number, Reynolds number, non-dimensional rotation speed of the cylinder, and the location of the cylinders. Their study demonstrated that average Nusselt number depends on direction of rotational speed of cylinders which eventually affect flow pattern; isotherms . Selimefendigil and Oztop [4] numerically researched using finite element method mixed convection of ferrofluid lid driven cavity in the presence of two rotating cylinders. Their research showed that variations of Reynolds number magnetic dipole strength affect flow patterns and thermal transport within the cavity. The results of their research disclose the fact of heat transfer enhancement within the cavity depending on cylinder angular velocities, ratio of angular velocities and diameter ratios. Mixed convection heat transfer in a lid-driven cavity with a rotating cylinder for various parameters such as Richardson number, the non-dimensional angular velocity of the cylinder, and the direction of rotation was investigated by Khanafer,Aithal and Vafai [5]. Their results revealed that angular velocity of the cylinder (magnitude and direction) have effect on average Nusselt number. Moreover, their study intanced greater heat transfer enhancement with a rotating cylinder compared with a stationary cylinder.

The present study has been first validated on mixed convection in a square cavity in the presence of two rotating cylinders positioned horizontally for Richardson number, Ri=0.01,1,5,10 and for Reynolds number, Re=100 keeping equal distance from side walls and from top and bottom wall where the wall of the cylinders was kept insulated. Further investigation has been done on conjugate mixed convection in the same cavity but the cylinders have been positioned vertically for $Ri=1$ and $1\leq Re\leq 10000$ keeping equal distance from side walls and from top and bottom wall for different orientations. Then for the best heat transfer rate from them the distances have been changed to compare between them for that orientation.

2. MATHEMATICAL MODEL

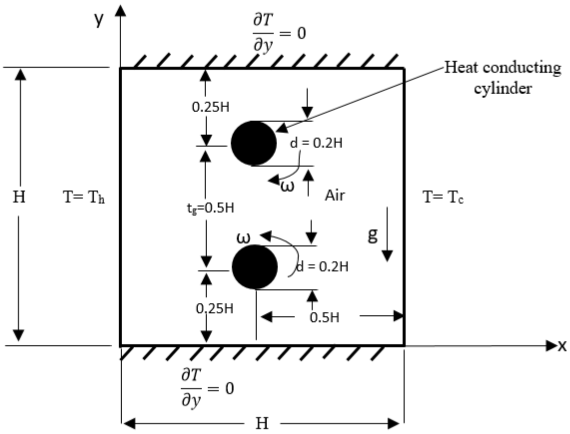

Fig.1: Schematic diagram of the square cavity with two rotating circular cylinders in CCW-CW orientation.

The schematic description of the present study has been shown in Fig.1.The flow inside the cavity has been assumed two-dimensional, steady, laminar. Two rotating circular cylinders of radius r_0 (= d/2=0.1H) are placed within the cavity as depicted in Fig. 1. No slip boundary condition is imposed on all cavity walls and both cylinders are rotating.The left wall has been kept at a high temperature T_h and the right wall has been kept at a low temperature T_c .The working fluid in the cavity is air (Prandtl number, $Pr=0.70484$) and the thermo-physical properties of the working fluid are taken to be constant except for the density variation, which is assumed to vary linearly with temperature according to the Boussinesq approximation. Mixed convection phenomena inside the domain follow the mass, momentum and energy conservation equations that read, in its dimensionless form

$$\frac{\partial U}{\partial X}+\frac{\partial V}{\partial Y}=0 \tag{1}$$

$$U\frac{\partial U}{\partial X}+V\frac{\partial U}{\partial Y}=-\frac{\partial P}{\partial X}+\frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2}+\frac{\partial^2 U}{\partial Y^2}\right) \tag{2}$$

$$U\frac{\partial V}{\partial X}+V\frac{\partial V}{\partial Y}=-\frac{\partial P}{\partial Y}+\frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2}+\frac{\partial^2 V}{\partial Y^2}\right)+Ri\Theta \tag{3}$$

$$U\frac{\partial \Theta}{\partial X}+V\frac{\partial \Theta}{\partial Y}=\frac{1}{PrRe}\left(\frac{\partial^2 \Theta}{\partial X^2}+\frac{\partial^2 \Theta}{\partial Y^2}\right) \tag{4}$$

For heat conducting cylinders, the energy equation is:

$$\frac{\partial^2 \Theta}{\partial X^2}+\frac{\partial^2 \Theta}{\partial Y^2}=0 \tag{5}$$

The following scales are used to get the above non-dimensional governing equations since it is convenient to solve the above equations in non-dimensional form:

$$X=\frac{x}{H},Y=\frac{y}{H},U=\frac{u}{u_0},V=\frac{v}{u_0},\Theta=\frac{T-T_C}{T_H-T_C},$$

$$P=\frac{p}{\rho u_0^{\ 2}},u_0=r_0\omega,Re=\frac{u_0H}{\nu}, \tag{6}$$

$$Gr=\frac{g\beta(T_H-T_C)H^3}{\nu^2},Pr=\frac{\nu}{\alpha}$$

Where, T_H is the highest temperature and T_C is the lowest temperature. The boundary conditions for the present study are presented in Table I

TABLE I: BOUNDARY CONDITIONS OF THE PRESENT MODEL IN NON-DIMENSIONAL FORM

Boundary wall(s)	Thermal field	Flow field
Left vertical wall	$\Theta = 1$ (isothermal)	$U=V=0$
Right vertical wall	$\Theta = 0$ (cold wall)	
Top and bottom walls	$\partial\Theta/\partial X = 0$ (insulated)	
Rotating cylinder		For clockwise rotation(CW): $U=\frac{y-y_c}{\sqrt{(x-x_c)^2+(y-y_c)^2}}$, $V=-\frac{x-x_c}{\sqrt{(x-x_c)^2+(y-y_c)^2}}$ For Counterclockwise rotation(CCW): $U=-\frac{y-y_c}{\sqrt{(x-x_c)^2+(y-y_c)^2}}$, $V=\frac{x-x_c}{\sqrt{(x-x_c)^2+(y-y_c)^2}}$

At the solid–fluid vertical interfaces of the block:

$$\left(\frac{\partial\Theta}{\partial X}\right)_{fluid}=K\left(\frac{\partial\Theta}{\partial X}\right)_{solid}$$

At the solid–fluid horizontal interfaces of the block:

$$\left(\frac{\partial\Theta}{\partial Y}\right)_{fluid}=K\left(\frac{\partial\Theta}{\partial Y}\right)_{solid}$$

Here, $K = k_s/k_f$ =12624.244942 is the thermal conductivity ratio of the solid wall and the fluid where k_s=395(for copper) and k_f=0.031289(for air).Further, dimensionless heat flux is calculated by computing the average Nusselt number along the heated vertical wall of the cavity as

$$Nu = -\int_0^1 \left.\frac{\partial \Theta}{\partial X}\right|_{X=0} dY \qquad (7)$$

3. SIMULATION PROCEDURE

Galerkin finite element formulation has been used to solve the non- dimensional governing equations. The application of this technique is well described by Taylor and Hood [6] and Dechaumphai[7]. To validate the developed model, simulations have been carried out to compare the mixed convection problem in a differentially heated cavity in the presence of rotating cylinders as reported by Khanafer, Aithal and Vafai [5]. The summary of the comparison has been shown in Fig.2 .It has been observed that present result and reported result have closest agreement.

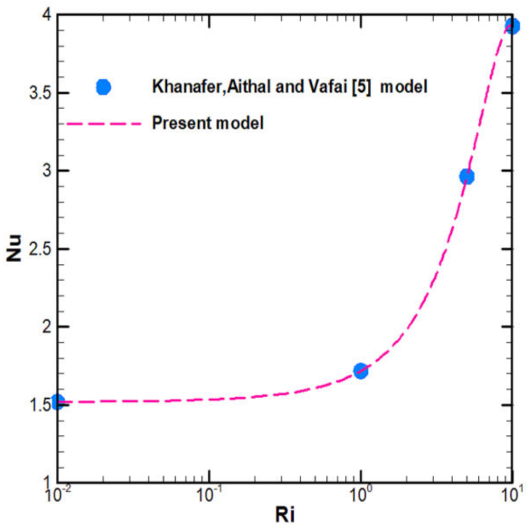

Fig.2: Comparison between the present result and the result of Khanafer,Aithal and Vafai [5] in terms of average Nusselt number for for various Richardson numbers (Re=100) for rotational speed of ω=10 . Left cylinder rotating clockwise and right cylinder rotating counterclockwise

4. RESULTS AND DISCUSSIONS

In the current numerical work, the following ranges of the dimensionless groups are considered: The working fluid is air with Prandtl number (Pr)= 0.70484, the value of the Richardson number is 1 while the range of Re is between 0 and 10000.The physical parameters of the present system are:location of the cylinders(t_g=0.25H,0.5H,0.75H); speed ratios of the cylinders taken as 0.1,0.5,1,2,10;the direction of cylinders: CCW-CCW,CW-CW,CW-CCW,CCW-CW. The numerical results have been used to explain the effect of several parameters at a small fraction of the possible situations by simplifying the configuration. The presentations of the results have been started with the streamline and isotherm patterns in the cavity. Representative distributions of Nu at the heated wall in the cavity have also been presented.

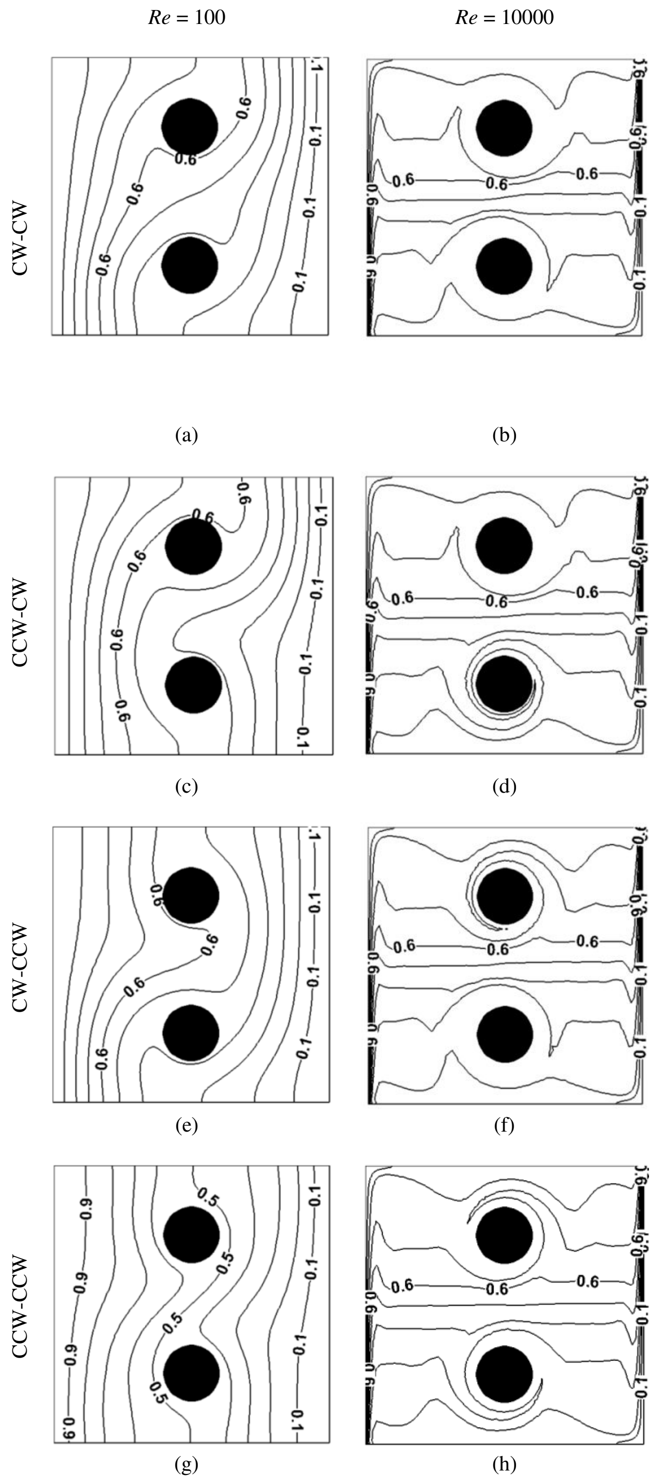

Fig.3: Isotherm plots for different orientations of rotating cylinders inside a square cavity.

This section presents selective results for the contour plots of streamlines, isotherms inside the square cavity having two vertically placed rotating cylinders. Isotherm plots are used to analyze thermal field whereas streamline plots are used to visualize flow field inside the square cavity. Fig. 3and 4, show the isotherms and streamlines variation for Ri=1 and inner rotating circular cylinder location (t_g=0.5H) when the

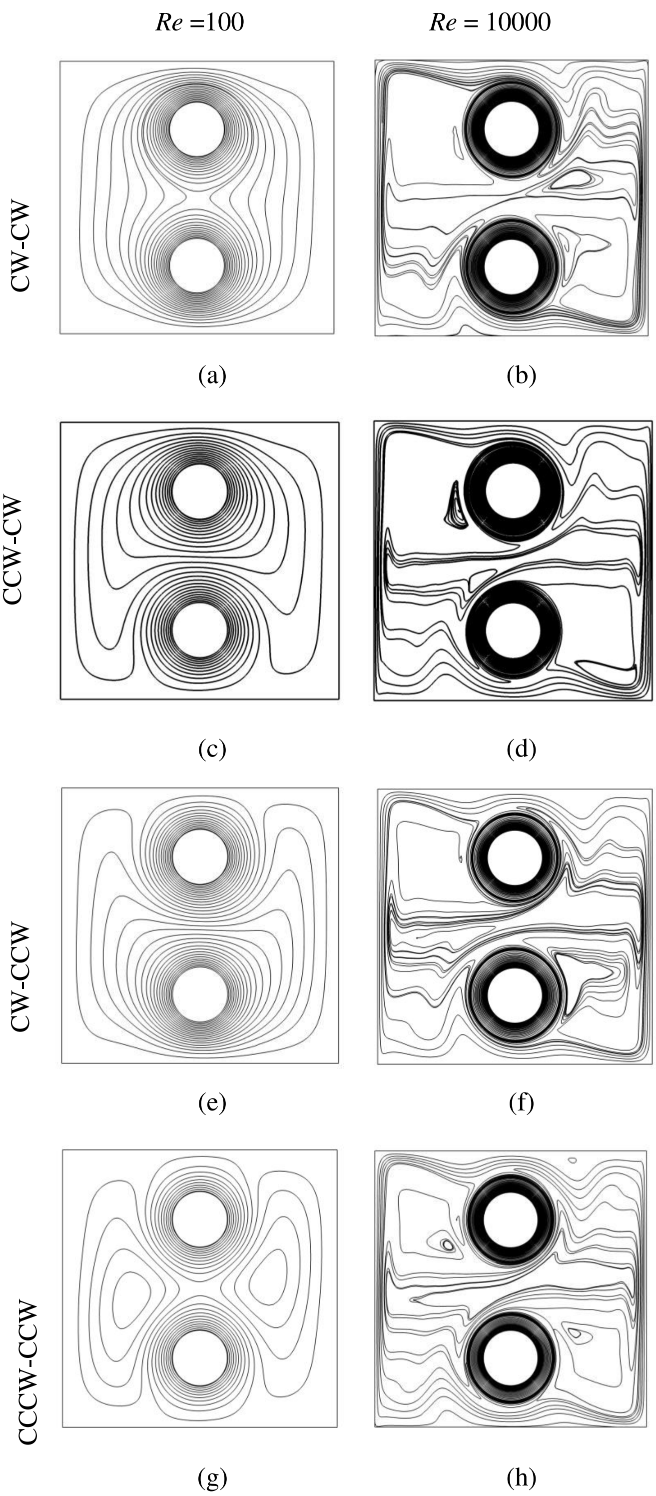

Fig.4: Streamline plots for different orientations of rotating cylinders inside a square cavity.

solid–fluid thermal conductivity ratio (K) is 12624.244942, Ω=1 and Re=100 and Re= 10000 respectively as a representative case. It is observed that, with the increase of Re, the isotherm contour greatly changes. For all four types of different orientations it is observed that for low Reynolds number (here in the Fig.3 lowest Reynolds number is 100) parallel lines (compared with vertical axis passing through the centre of cylinder) are observed in the two edges of the square cavity and ripples are generated after some distance of the edges of the cavity around the cylinders intersecting the insulated walls. After some distances from edges increasing the ripple portions increase heat transfer rate.

For all four cases over the cylinder slightly oval shaped lines are observed which suggest that higher rate of heat transfer over those surfaces than the edges .For clockwise-clockwise orientation, it is observed that with the increase of Reynolds number from 100 to 10000s the parallel lines become distorted lines. Generally the isotherm plots approximately parallel and symmetrical adjacent to the left hot side wall of the enclosure indicate that the conduction heat transfer has become the dominant mode of energy transport in the square enclosure. The reason of domination of distorted lines is due to small role of angular rotational velocity of the rotating solid circular cylinder, when the Reynolds number is small. From the other hand, the isotherm lines become more thicker and occupy almost the whole enclosure, when the Reynolds number is increased from 100 to 10000.This leads to increase the heat transfer effect by the rotating solid circular cylinder, and as a result more heat can be transferred from the rotating solid circular cylinder and dissipated uniformly inside the square enclosure. These distorted lines indicate that heat transfer in that zone is convection mode of heat transfer. In the present developed model, the rotating circular cylinder is heat conducting cylinder.

So, it is natural that the number of those distorted lines are around cylinder are higher than edges of vertical walls of square cavity. In this case this number rises when Reynolds number increases in case of 10000. Again, the increase of Reynolds numbers indicate that the effect of viscosity decreases and it ensures enhancement of fluid mixing of fluid. Therefore heat transfer increases .This same scenario is observed in case of other three orientations. It is from Fig.3 that for counterclockwise-counterclockwise orientation the number of distorted lines is higher than other three orientations. So, it can be said that for counterclockwise-counterclockwise orientation with the increase of Reynolds number heat transfer increases.

Fig. 4 illustrates flow field via streamline plot inside the differentially heated square cavity. Generally it can be said that, when the air inside the square enclosure touches the conductive rotating circular cylinder, it becomes hotter and lighter due to high thermal conductivity effect of the inner cylinder. The air is lifted and tries to move around the conductive rotating circular cylinder until it contacts the adiabatic top wall of the enclosure. The hot air near the adiabatic top wall becomes colder and then moves adjacent to the right cold side wall which changing its direction of flow leading to produce a rotating identical vortices around the conductive rotating circular cylinder. When Ri= 1, the heat transfer mode in the square enclosure is occurred by the combined mechanisms of natural and forced convection (i.e., the mixed convection is dominant).In this situation, the buoyancy forces effect balances the effect of the rotating circular cylinder. Observing the figures it can be said that the direction of rotation of the cylinders plays an important role in controlling the streamline distribution around them the flow pattern is strongly influenced by Reynolds number. Here, for Re=100 no minor vortices is observed. All we see are major vortices. In case of CW-CW this major vortices are symmetrically distributed for both cylinders. In case of CCW-CW major vortices are more in number for upward cylinder which rotates clockwise. Again in case of CW-CCW these vortices are more in number for downward cylinder which rotates clockwise. For CCW-CCW, the major vortices size decrease, the introduction of minor vortices appears due to dominance of forced convection. It indicates that CCW-CCW has been affected both by forced and natural convection while the other three are mainly by natural convection. When, the Reynolds number increases from 100 to 10000 as shown in Fig. 4, the role of the angular rotational velocity of the rotating solid circular cylinder becomes clear dominant, since the Reynolds number is directly proportional to the angular rotational velocity (ω). Due to this reason, the circulation of flow vortices increases dramatically and becomes stronger, which leads to make the forced convection effect more dominant. It can be observed that, intensity of the vortices circulation becomes more concentrated comparing with the corresponding vortices when R=100 as shown in Fig. 4. With the increase of Reynolds number from 100 to 10000 the intensity of streamlines increases within the cavity and there is presence of minor vortices alongside the main vortices. These minor vortices are strong enough to reduce the thermal boundary layer thickness. If boundary layer thickness decreases, the heat transfer increases. Fig. 4 shows that the number of secondary vortices increases with the increase of Reynolds number. This increase of Reynolds number increases heat transfer. This is same for all four orientations according to Fig.4.It is observed that the secondary vortex is higher in case of counterclockwise-counterclockwise orientation it. So, it can be said that heat transfer rate increases higher in case of counterclockwise-counterclockwise orientation than other three orientations.

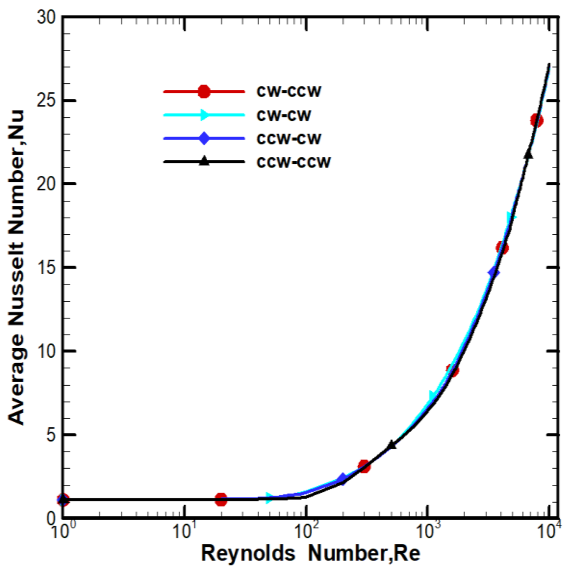

Fig.5: Variation of Nusselt number with Reynolds number for different orientation of rotating circular cylinder.

Fig.5 illustrates that average Nusselt number is independent of Reynolds number upto about Reynolds number 1000.After that with the increase of Reynolds number average Nusselt number increases maintain an exponential relationship. Increasing Nusselt number indicates increase of convection heat transfer. From Fig.5 it is observed that average Nusselt number is higher for counterclockwise-counterclockwise orientation. So, heat transfer rate is higher than other three orientations for this case. The same result has been from Fig.3 and Fig.4.

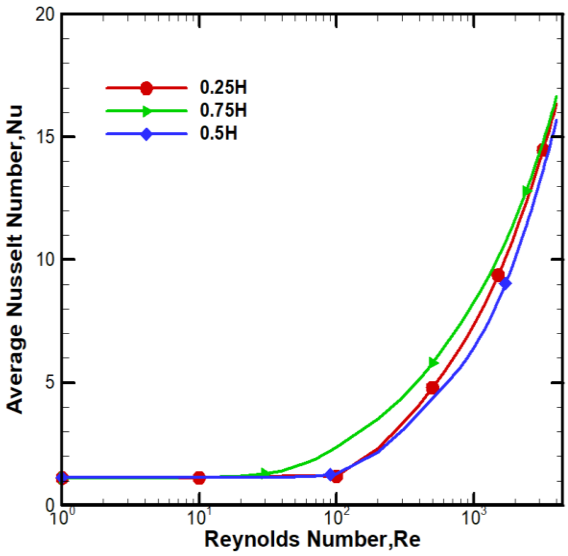

Fig.6: Variation of Nusselt number with Reynolds number for CCW-CCW orientation of different distances betewen rotating circular cylinders.

Fig.6 shows that if the distance between the cylinders is changed than the rate of heat transfer changes as average Nusselt number changes. Again, average Nusselt number increases with the increase of Reynolds number. In case of t_g= 0.75H the highest average Nusselt number has been found. For this reason, in this case heat transfer rate is higher than other three considered cases.

5.CONCLUSION

Some issues are addressed in the study, i.e., the influences of Re , Nu , locations of cylinders, and direction of cylinders on the heat transfer. Based on the results obtained in the present study, several conclusions can be summarized as follows:

(a) The direction of cylinders has a great role on heat transfer rate. For CCW-CCW we get the highest heat transfer rate.

(b)Increasing Re indicates heat transfer rate increases due to enhancement of forced convection.

(c)The location of both cylinders has a great role. For t_g=0.75H, CCW-CCW we get the highest heat transfer phenomena. So, Nu is also highest in this case.

6. ACKNOWLEDGEMENT

We would express our heartiest gratitude to Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET) for providing continuous encouragement throughout our research. We are really lucky to have such support for our computational resources provided by our department to continue our research.

7. REFERENCES

[1] C.-C. Liao and C.-A. Lin,"Mixed convection of a heated rotating cylinder in a square enclosure", *International Journal of Heat and Mass Transfer,*vol. 72 ,pp. 9–22 ,2014

[2] D. Chatterjee, S.K. Gupta and B. Mondal, "Mixed convective transport in a lid-driven cavity containing a nanofluids and a rotating circular cylinder at the center", *International Communications Heat and Mass Transfer* ,vol. 56, pp. 71–78, 2014

[3] Khalil Khanafer and S.M. Aithal, "Mixed convection heat transfer in a lid-driven cavity with a rotating circular cylinder", *International Communications Heat and Mass Transfer* ,vol.86 ,pp. 131–142,2017

[4] F. Selimefendigil and H. Oztop, "Mixed convection of ferrofluids in a lid driven cavity with two rotating cylinders", *Engineering Science and Technology, an International Journal* ,vol.18,no.3, pp. 439–451,2015

[5] Khalil Khanafer, S.M. Aithal and K.Vafai, "Mixed convection heat transfer in a differentially heated cavity with two rotating cylinders", *International Journal of Thermal Sciences*,vol.135,pp. 117-132,2019

[6] C. Taylor and P. Hood, "A numerical solution of the Navier–Stokes equations using finite element technique", *Computers and Fluids* ,vol.1,no.1,pp. 73–89,1973

[7] P. Dechaumphai, *Finite Element Method in Engineering,second edition*, Bangkok:Chulalongkorn University Press, 1999.

8.NOMENCLATURE

Symbol	Meaning	Unit
g	gravitational acceleration	ms^{-2}
Gr	Grashof number	dimensionless
H	Cavity side length	m
Nu	Average Nusselt Number	dimensionless
p	Pressure	Nm^{-2}
P	Pressure	dimensionless
Pr	Prandtl Number	dimensionless
r_o	Radius Of Cylinder	m
Re	Reynolds Number	dimensionless
Ri	Richardson Number	dimensionless
T	Temperature	K
u,v	velocity components	ms^{-1}
U,V	velocity components	dimensionless
x,y	Cartesian coordinates	m
X,Y	Cartesian coordinates	dimensionless
CW	Clockwise	dimensionless
CCW	Counterclockwise	dimensionless
α	thermal diffusivity	m^2s^{-1}
β	thermal expansion coefficient	K^{-1}
ρ	density of the fluid	kgm^{-3}
Ω	angular speed of the cylinder	dimensionless
Θ	temperature	dimensionless
ν	kinematic viscosity	m^2s^{-1}
f	fluid	dimensionless
s	solid	dimensionless