A NUMERICAL INVESTIGATION ON CONVECTION HEAT TRANSFER OVER ROTATING CYLINDER

Ashok Kumar[1], Sheetal dewangan[2], Ganpat lal rakesh[3], chunnu lal dewangan[4]
[1,2,3]Asst. Professor, Mechanical Department, CCET, Bhilai, Chhattisgarh, India,
Email - [1]ashoknitt14@gmail.com, [2]sdewangan1982@gmail.com, [3]ganpat123@gmail.com, [4]chunnu.dewangan@gmail.com,

Abstract: This numerical investigation devoted to convective heat transfer occurs on rotating circular cylinder in transient laminar flow regimes. The finite volume method technique with implicit pressure based model used and Navier-Stroke equation as governing equation. The numerical investigation predict the effect of spin rate ($0 \le a \le 2.5$) of circular cylinder and Prandtl number ($0.7 \le Pr \le 500$) of flowing fluid at Reynolds number 100. The cylinder wall consider as isothermic wall with uniform spin rate. The increasing Prandtl number increases average nusselt number with also reduce the thermal boundary layer thickness and increasing of spin rate decreases the average nusselt number. The high pining of cylinder lowering the average Nusselt number and dcreases heat transfer rate from rotating cylinder.

Key Words: convection, spinning, Prandtl number, Nusselt number, rotating cylinder.

1. INTRODUCTION:

Numerical investigation of two dimensional incompressible fluid flows over rotating circular cylinder is a greatest area of interest for researcher. Fluid flow over circular rotating cylinder is great important of design of aerodynamic system, fluid machinery and sea shore structures. The many researcher describes the phenomena of boundary layer separation on cylinder surface at various spinning rate but very less literature available of characteristics heat transfer rate on based on Prandtl number.

The some relevant literature areFornberg [1], has worked on steady state flow across circular cylinder up to Re=300. In this paper describe the behavior of wake formation on downstream of cylinder also show the characteristics stable and unstable vortex shedding appear on downstream of cylinder. Lienhard [2], investigate the various type of flow regime appears downstream of circular cylinder. The Re is below 5 the flow attached, the increasing of Re the detached flow at vortex street the stable and at Re = 49 point as critical Reynolds number and further increase the Re unstable vortex street appears at laminar flow regime, the Re more than 300 the fully turbulent flow vortex street. Ingham et al [3], has done the numerical investigation on flow over cylinder at steady state uniform flow regime. The results for hydrodynamic coefficient are present for low Reynolds number Re = 5 to 20 with angular velocity $0 < a < 3$. Ingham et al [4], also present another paper for Re = 60 and 100 at $0 < a < 1$. In this paper describes the behavior of drag and lift coefficient characteristics varies with Reynolds number and angular velocity of cylinder. Mittal et al [5], Investigate the vortex street appears at downstream of cylinder with angular velocity range had taken 0 to 5 for Re =200 at steady state flow regime. Hodnett et al [6], has done the numerical modeling on heat transfer from a circular cylinder at low Reynolds number. This problem calculation was able to temperature distribution and nusselt number for small and large value of time. Kang [7], has done the numerical simulation based on immersed boundary condition at Reynolds number 100 with various angular velocity of circular cylinder. This paper describes the flow characteristics and mechanism under the uniform shear on cylinder surface.Rajani et al [8], focused on the different laminar flow regime for two dimensional and three dimensional analyses with implicit pressure based finite volume method had adopted. They worked on very low Reynolds number and describes the behavior of laminar flow regime, the also concluded Re = 49 is a critical Reynolds number.

Bijjam et al [9], investigate the heat transfer over circular cylinder at unsteady state at Re = 100 and 150. They describe the hydrodynamic behavior of fluid with effect of nusselt number which governs the heat transfer take placed on cylinder surface. Bharti et al [10], has done the convective heat transfer across the circular cylinder at Reynolds number varies from 10 to 45 with Prandtl number varies 0.7 to 400. In this paper describes the effect of Re, Pr, thermal boundary condition and temperature distribution around the circular cylinder. The convective heat transfer calculated based on two different assumption, constant wall temperature and uniform surface heat flux. The uniform surface heat flux shows higher value of heat transfer as compare to constant wall temperature. Buyruk [11], has done the experiment study of heat transfer from circular cylinder at high Reynolds number. This experiment through shows the significance variation of local nusselt number and local pressure coefficient around the cylinder surface. Sharma et al [12], investigated the heat transfer from rotating cylinder on steady state laminar flow regime. In this study refer the low Reynolds number Re = 1 to 35, Prandtl number varies from 0.7 to 100 and angular velocity of cylinder range 0 to 5. This paper concluded that the increasing Reynolds number and Prandtl number heat transfer rate increase but with increasing the angular velocity the heat transfer rate reduce. Golani et al [13], has investigated the heat transfer across circular cylinder at transient laminar flow regime. In this paper shows the increasing Reynolds number the heat transfer rate also increased with constant Prandtl number (0.7). Hear also describes the behavior of hydrodynamic coefficient with increasing Reynolds number.

Nusselt number and heat transfer from cylinder is associated with the variation of Reynolds number, Prandtl number and angular velocity. The above discussion, we have seen the significance behavior of hydrodynamic forces, wake formation and instability of flow regime. This investigation is devoted to heat transfer from rotating cylinder and characteristics of local nusselt number on cylinder surface with variation of Prandtl number and angular velocity for Re = 100.

2. NUMERICAL MODELING:

2.1 Governing Equation

The Continuity equation, Navier-Stroke Momentum Equation and Energy Equation are used as dimensionless governing equations to solve the two dimensional grid domain for finite volume pressure based implicit method. Two dimensional, unsteady, incompressible and Newtonian fluids with constant fluid properties can be expressed by following equations:

Continuity Equation,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad (1)$$

Momentum Equation,

$$\frac{\partial u}{\partial t}+\frac{\partial (uu)}{\partial x}+\frac{\partial (uv)}{\partial y}=-\frac{\partial p}{\partial x}+\frac{1}{\text{Re}}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad (2a)$$

$$\frac{\partial v}{\partial t}+\frac{\partial (uv)}{\partial x}+\frac{\partial (vv)}{\partial y}=-\frac{\partial p}{\partial y}+\frac{1}{\text{Re}}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \quad (2b)$$

Equation 2a is corresponding to x-axis and 2b is crresponing y-axis direction.

Energy Equation,

$$\frac{\partial T}{\partial t}+\frac{\partial (uT)}{\partial x}+\frac{\partial (vT)}{\partial y}=\frac{1}{\text{RePr}}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right) \quad (3)$$

Where u and v are velocity component along the x and y directions of a Cartesian coordinate system respectively, t is the time, p is the pressure, Re (= $\rho U_\infty d/\mu$) is the Reynolds number, Pr (= $\mu C_p/k$) is the Prandtl number, T is the temperature, d is the diameter of cylinder, k is the thermal conductivity of fluid, C_p is the heat capacity of fluid, ρ is the density of fluid, μ is the dynamic viscosity of fluid and U_∞ is the free stream velocity of fluid.

Lift and Drag coefficients are expressed as following:

$$C_L=C_{LP}+C_{LV}=\frac{2F_L}{\rho U_\infty^2 d} \quad (4a)$$

$$C_D=C_{DP}+C_{DV}=\frac{2F_D}{\rho U_\infty^2 d} \quad (4b)$$

The lift and drag coefficients are combine effect of viscous and pressure coefficient. where C_{LV} and C_{LP} are viscous lift and pressure lift coefficient respectively and C_{DV} and C_{DP} are viscous drag and pressure drag coefficient respectively. Subscript s used for cylinder surface.

$$C_{LV}=\frac{2}{\text{Re}}\int_0^{2\pi}(\omega)_S\cos\theta d\theta \quad (5a)$$

$$C_{LP}=\frac{1}{2}\int_0^{2\pi}(\text{P})_S\sin\theta d\theta \quad (5b)$$

$$C_{DV}=\frac{2}{\text{Re}}\int_0^{2\pi}(\omega)_S\sin\theta d\theta \quad (5c)$$

$$C_{DP}=\frac{1}{2}\int_0^{2\pi}(\text{P})_S\cos\theta d\theta \quad (5d)$$

The surface average nusselt number can be evaluated as follows:

$$\overline{Nu}=\frac{1}{2\pi}\int_0^{2\pi}Nu_\theta d\theta \quad (6)$$

Where Nu_θ (=$h_\theta d/k$) local Nusselt number on cylinder surface and θ is angular position on cylinder surface.

2.2 Boundary Condition

Flow over in rotating cylinder the boundary condition should be taken for computational domain as follows:

At inlet:
The uniform flow boundary condition used as, $u=U_\infty$, v=0, and $T=T_\infty$

On cylinder surface:
No slip condition assume with u=0, v=0, and $T=T_W$

At outlet:
Default boundary condition used as, $\partial u/\partial x=0$, $\partial v/\partial y=0$ and $\partial T/\partial x=0$

where T_∞ is the free stream fluid temperature and T_W is the cylinder surface temperature.

2.3 Computational Grid

The grid domain used for these simulations is structured O-type meshing which has done on Gambit 6.3. The inlet and outlet boundary of domain is 100D (D= diameter of cylinder) times away from cylinder surface and cylinder rotating in counter clockwise direction. The meshing domain carried out near the cylinder wall grids are very fine and along the away from cylinder, grid became coarse. The fine grids are much efficient the capture of fluid behavior around the cylinder and produce results by visualization of flow pattern. The boundary and domain dimension is illustrated on fig-1.

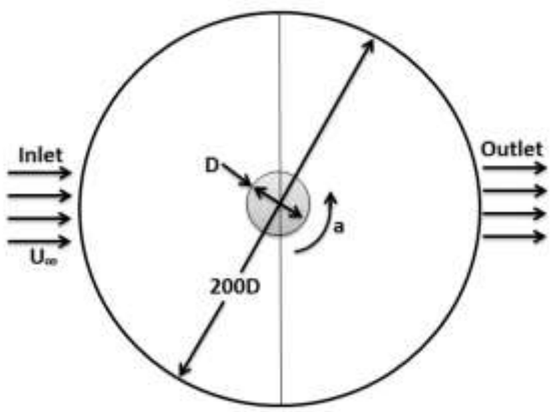

Fig-1: Boundary condition

2.4 Numerical Simulation

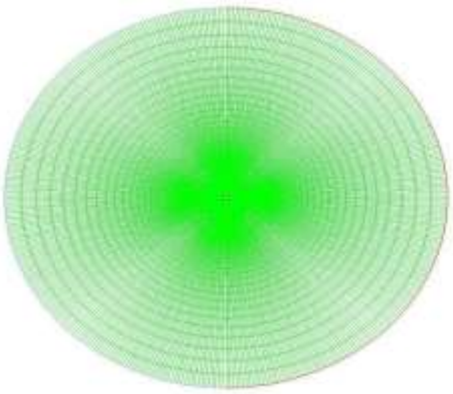

Fig-2a: O-Type grid meshing

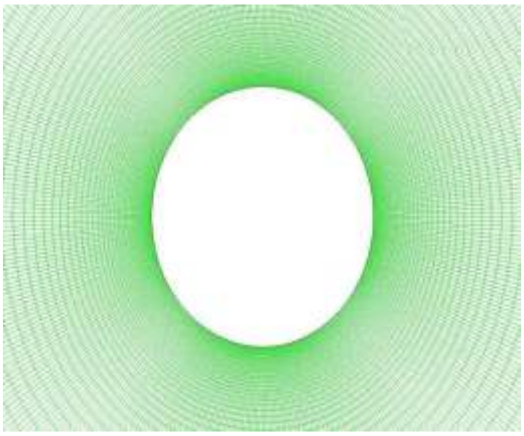

Fig-2b: Close view of grid aroung cylinder

This numerical simulation is carried out by the commercial simulation software Ansys 13. This solution based second order differential equation on finite volume method with implicit pressure modeland Navier-Stroke Equation used as governing equation and solved in irritation manner. These simulation work has done on for Re = 100 with various Prandtl number (Pr = 0.7 to 500) and angular velocity (a = 0 to 2.5). Evaluate the nusselt number the temperature difference taken as very small (ΔT = 1℃), and thermo physical property change of fluid is negligible and result should be accurate. The cylinder surface is considered as not slip condition with counter clock-wise rotation with uniform dimensionless spinning rate.The working fluid consider as air and the free stream velocity for Re = 100 is 1 m/s and the viscosity coefficient air is 0.001 Ns/m^2.

2.5 Result Validation

Two dimensional numerical simulation has done for incompressible, viscous, laminar flow regime at Re = 100 and Pr = 0.7 with stationery circular cylinder. R Golani et al [13] has done the numerical study flow across cylinder at range of Re = 50 to 180 with Pr = 0.7. This study based of finite volume method and concluded that the drag coefficient and nusselt number at Re = 100 is 1.3063 and 5.0866 respectively. S Bijjam et al [9] worked on heat transfer across the cylinder at Re = 100 and 150 with Pr = 0.7. This paper describes the characteristics of hydrodynamic behavior and effect of Reynolds number on heat transfer rate at Re = 100 nusselt number is calculated as 6.324. N Mahir et al [14] numerical study on heat transfer across the cylinder at Re = 100 and 200. This paper focused on the temperature distribution around the cylinder and heat transport visualization in tandem arrangement of cylinder and result concluded for drag coefficient and nusselt number at Re = 100 is 1.368 and 5.179 respectively. B N Rajani et al [8], Md. M Rahman et al [15] and S Tuann et al [16] has worked on unsteady state flow across the circular cylinder at low and moderate Reynolds number. These papers describe the hydrodynamics behavior of flow regime and vortex street formation on downstream of cylinder. Their results are compared in table 1 and show the excellent agreement with present work.

Table-1: Validation of numerical method at Re = 100 and Pr = 0.7 (a = 0)

Source	Cd	Nu
Present Work	1.286	5.0619
R Golani et al [13]	1.3063	5.0866
B N Rajani et al [8]	1.3353	--
S Bijjam et al [9]	--	6.324
N Mahir et al [14]	1.368	5.179
Md. M Rahman et al [15]	1.245	--
S Tuann et al [16]	1.221	--

3. RESULT AND DISCUSSION:

3.1 Flow Pattern

The vorticity contour examined for two dimensional incompressible fluid flows over a heated circular rotating cylinder with various dimensionless angular velocity of cylinder at Re=100 as show in fig-3. The visualization of vorticity contour on laminar flow regimes, the increasing of angular velocity of cylinder (counter clockwise direction) with flow pattern displaced upward on downstream of cylinder. Fig-4 show the stream line flow across the circular cylinder, the vortex shedding appears on downstream of cylinder and behavior of wake formation is influence by the angular velocity of cylinder. At the low angular velocity of cylinder or cylinder is stationary the two stagnant point are located on cylinder surface. The increasing the angular velocity the stagnant point is shifted with along the rotational direction and at high angular velocity they merge and form new single stagnation point. The measure of vertex shedding magnitude is used as Shrouhal number. The many researcher work on the study of characteristics of wake formation and vortex-street appears on the cylinder. The high Shrouhal number is show the strong magnitude of cylinder vibration and fluid structure and machine element are failed at Re = 100 the Shrouhal number is to be estimated St = 0.1580.

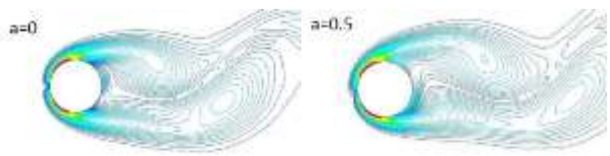

Fig-3: Instantaneous vorticity contour at different spin rate

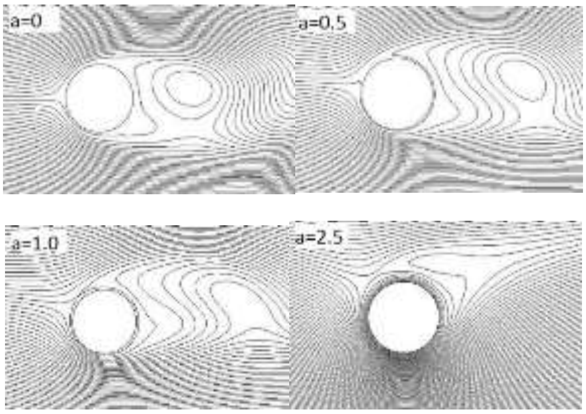

Fig-4: Instantaneous stream line contour at different spin rate

The hydrodynamic forces, darg and lift force are come to view due to combine effect of pressure force and viscus force on cylinder surface. The significant force acted on rotating cylinder surface and geneartion of lift due to magnus effect. Fig-5 show the time history for lift coefficient for laminar flow regime with range of angular speed a= 0 to 2.5 at Re = 100. The cyclic fluctuation of drag coefficient indicate the vortex sheadding apperas on downstream of cyinder and unstable laminar flow regime. The increase of angular speed with drag coefficient is decreased. The angular speed incresese the strong magnus effect show and lift coefficient is increases. At high angular speed of the cylinder fluctuation is reduced and downstream of cylinder flow regime became stable.

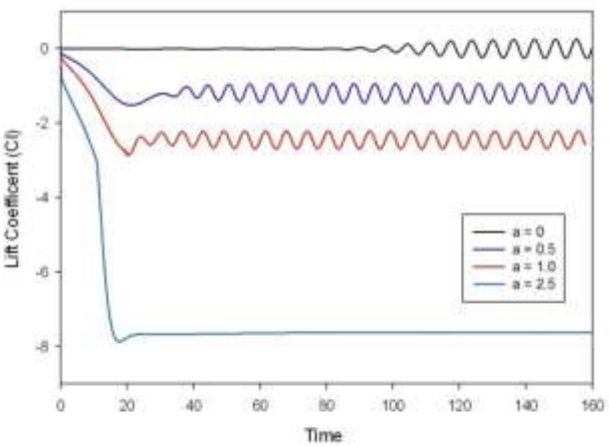

Fig-5:Graph between Lift coefficient and Time history at different spin rate

Fig-6 show the pressure coefficent distrubution on cylinder surface with various angular speed at Re = 100. The low rotating speed or stationary cylinder the pressure fluctition is high and varies with angular position of cylinder surface. The seperation point the pressure coefficient is zero and the negative pressure coeffcient indicate the wake formation on down stream of cylinder surface. Increasing the spinning rate of cylinder the magnitude of pressure coefficient also invreases and strong vaccume pressure created on down stream of cylinder surface. The high spinning rate merge to two

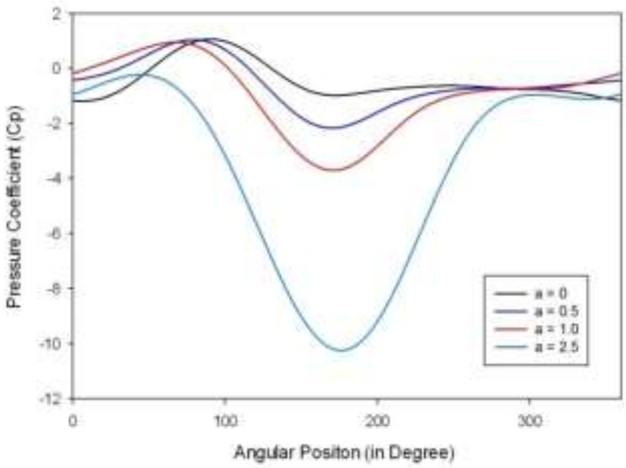

Fig-6: Graph between pressure coefficient and Angular position at different spin rate

stagbnation point and produce single stationation point and move outward at higher radial position from cylinder surface with increse cylinder spinning rate.

3.2 Isotherm Pattern

The instantaneous isotherm pattern for different Prandtl number ($0.7 \leq Pr \leq 500$) with range of angular speed a = 0 to 2.5 at Re = 100 show in fig-7. The isotherm represented the temperature distribution aroung the cylinder, thermal boundary thickness and characteristics of heat transport from heated cylinder surface to flowing fluid. The visualization of isotherm the high rotating speed displaced the vortex street and high Prandtl number easily disappear the temperature vortex street away from cylinder. The high Prandtl number shows the high thermal inertia with low thermal conductivity of fluid. At the low Prandtl number, the large region of temperature distribution around the cylinder but increases the Prandtl number the temperature region is shrink and only near thecylinder surface temperature distribution show at high rotating speed. The low Prandtl number is show thick thermal boundary and increasing the Prandtl number thermal boundary layer is become thinner.

3.3 Effect of Prandtl Number

The numerical investigation of heat transfer across the rotating cylinder is carried out under the constant wall temperature of cylinder surface with very small temperature difference between the cylinder surface and free stream fluid. The small temperature difference gives the good result due the change in thermo-physical properties is negligible. Fig-8 shows the local nusselt number around the cylinder surface at different Prandtl number. The variation of nusselt number around the cylinder surface shows the significant effect of heat transport from cylinder. Heat transfer influenced by the Nusselt number, the up stream of cylinder is attached flow but down stream is detached due to vortex street produce. The detached flow regime lower the average Nusselt number. The fig-7 show the isotherm pattern flow for temperature distribution around the cylinder surface. The high nusselt number produce high rate of heat transfer, werethe separation is occurs the nusselt number is drop and heat transfer rate reduced. At high rotating speed of cylinder and high Prandtl number the nusselt number is almost uniform around the cylinder surface and very low nusselt number indicate the fully separation is occurs and flow become swirling with single point stagnation point. Fig-9 shows the average nusselt number for various Prandtl number with range of rotation speed a = 0 to 2.5 at Re = 100. The increasing of Prandtl number increase the nusselt number with the increases of rotating speed reduced the nusselt number.

This numerical investigation done on transient laminar flow regime at Re = 100 with large range of Prandtl number (Pr= 0.7 to 500) and rotating speed range (a = 0 to 2.5). The graph ploted between average Nusselt number and spinning rate can we used to calculate the intermediate time average nusselt number for flow across the cylinder within the range of $0.7 \leq Pr \leq 500$ and $0 \leq a \leq 2.5$ at Re = 100.

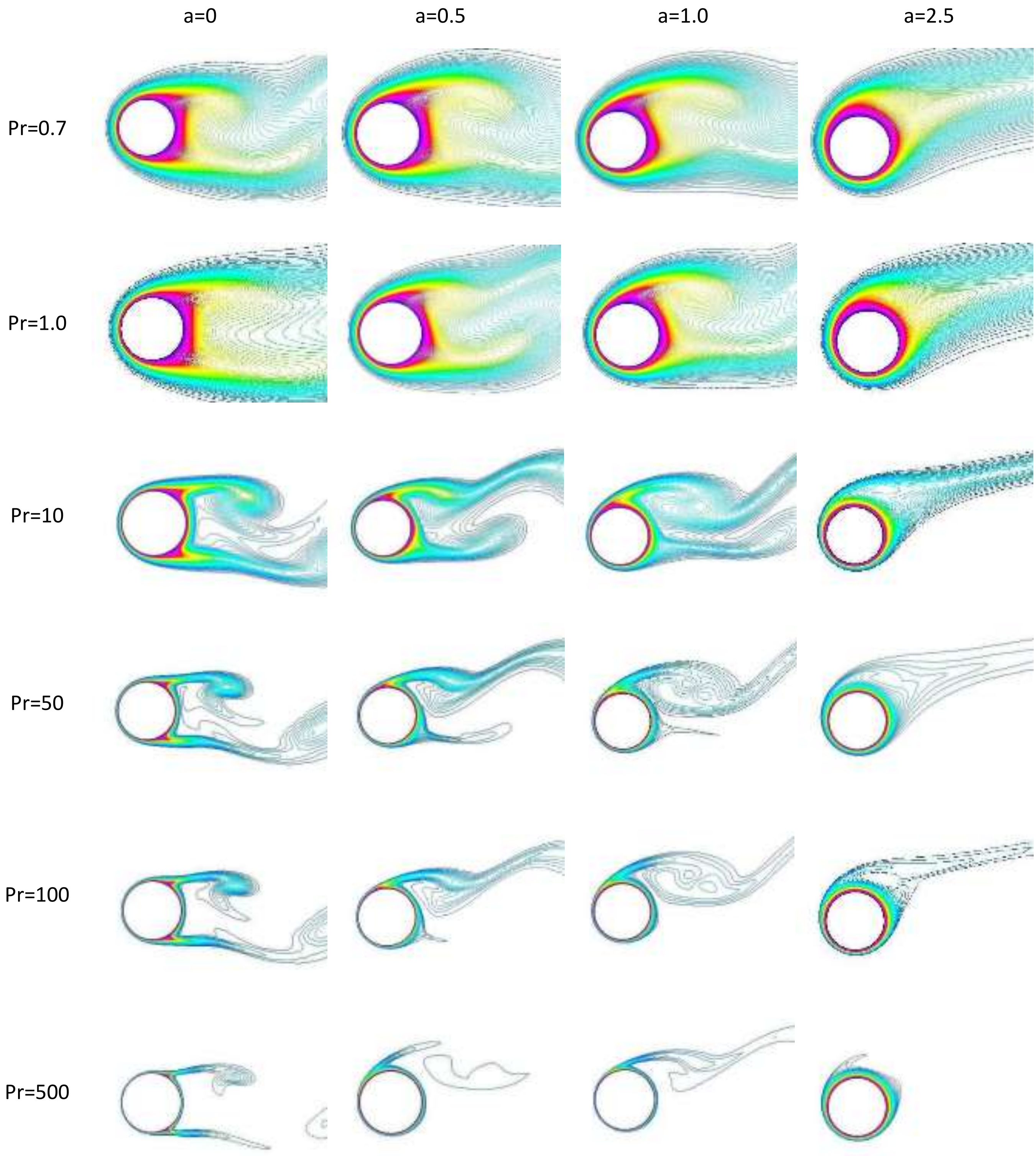

Fig-7: Instantaneous Isotherm contour at different spin rate and Prandtl number

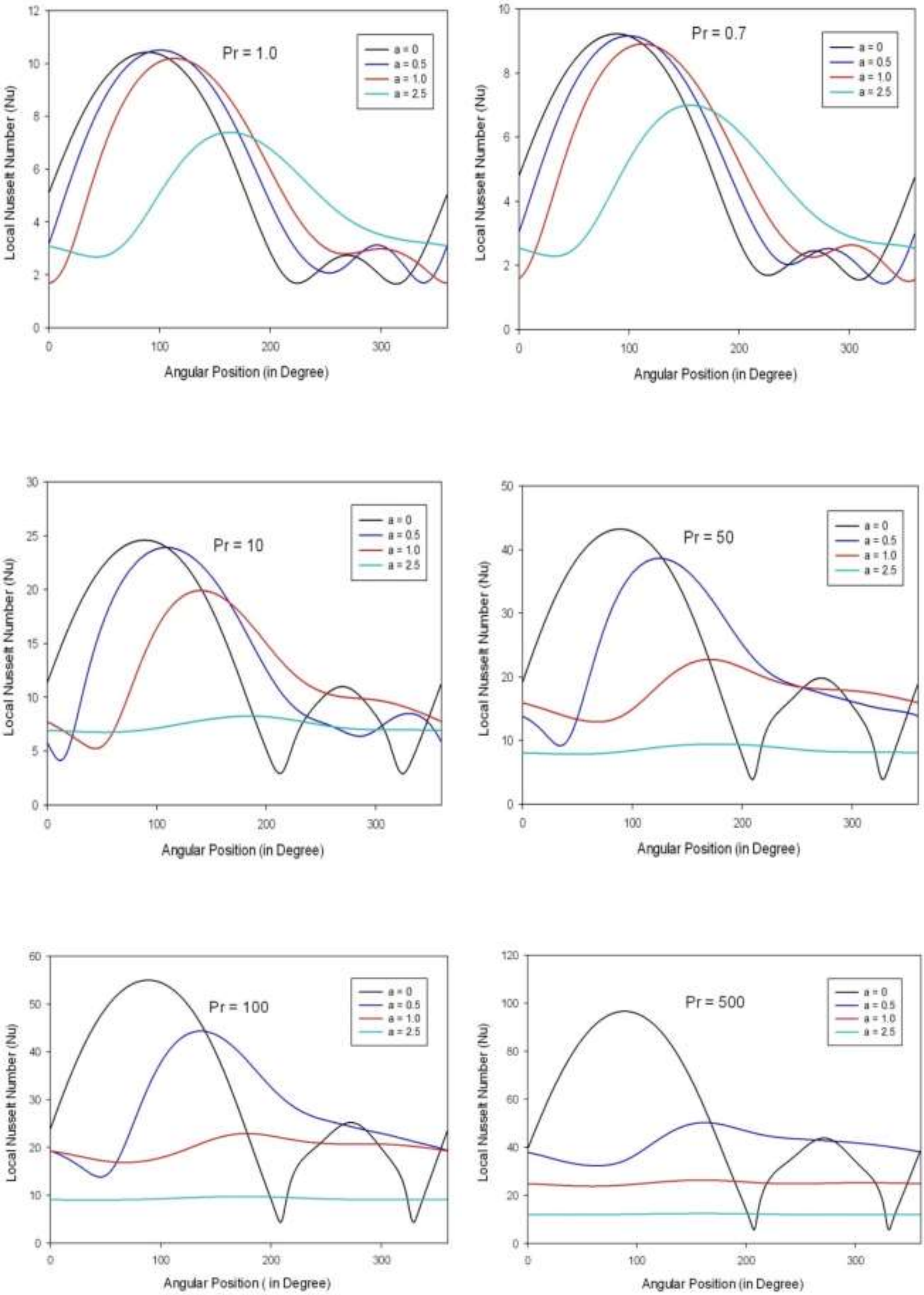

Fig-8:Graph between Local Nusselt number and Angular position at different spin rate

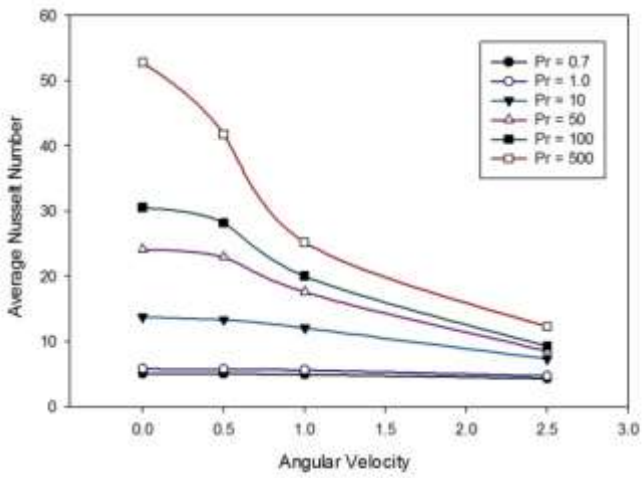

Fig-9:Graph between Average Nusselt number and Angular position at different spin rate

4. CONCLUSION:

Laminar unsteady convective heat transfer across the rotating cylinder numerically investigated using ANSYS 13, for several Prandtl number (Pr = 0.7 to 500) and range of rotating speed (a = 0 to 2.5) at Re = 100. The solution is based on finite volume method with pressure based implicit technique used. The continuity, momentum and energy equation are used as governing equation for solved the numerical code. The present result is compared with available literature and observed to good agreement. The flow pattern and isotherm pattern signifies the local nusselt number around the cylinder surface and separation is reduced the local nusselt number. The high Prandtl number increases the nusselt number but high rotation speed is reduced the nusselt number. Improve the heat transfer rate from the cylinder surface is only possible the attached flow and spinning rate of cylinder should be low.

REFERENCES:

1. Fornberg B. A numerical study of steady viscous flow past a circular cylinder. J Fluid Mech 1980; 98:819-55.
2. Lienhard JH. Synopsis of lift, drag, and vortex frequency data for rigid circular cylinders. Technical Extension Service 1966. Bulletin 300.
3. Ingham B D and Tang T. A Numerical Investigation into the Steady Flow Past a Rotating Circular Cylinder at Low and Intermediate Reynolds Numbers. J Computational Physics. vol.87, No.1,1990.
4. Ingham B D and Tang T. On study flow past a rotating circular cylinder at Reynolds number 60 and 100. J Computes & Fluid. Vol.19, No.2, pp. 217-230, 1991.
5. Mittal S, Raghuvanshi A. Control of vortex shedding behind circular cylinder for flow at low Reynolds numbers. Int J Numer Meth Fluids 2001; 35:421-47.
6. Hodnett PF, Rose DM. Unsteady heat transfer from a circular cylinder in a low Reynolds number flow. Journal of Applied Mathematics and Physics 1974; 25:179-88.
7. Kang S. Laminar flow over a steadily rotating circular cylinder under the influence of uniform shear. J Physics Fluid Vol. 18, 047106, 2006.
8. Rajani BN, Kandasamy A, Majumdar S. Numerical simulation of laminar flow past a circular cylinder. Applied Mathematical Modelling 2009; 33:1228-47.
9. Bijjam S, Dhiman AK, Srikanth S. Unsteady laminar flow and heat transfer across a circular cylinder confined in a channel. 4thInt Conference on Fluid Mechanics and Fluid Power 2010.
10. Bharti RP, Chhabra R, Eswaran V. A numerical study of the steady forced convection heat transfer from an unconfined circular cylinder. Heat Mass Transfer 2007; 43:639-48.
11. E Buyruk. Heat transfer and flow structures around circular cylinders in cross flow. Tr. J Engg. & Environmental Science. 23, pp. 299-315, 1999.
12. Sharma V, DhimanA K. Heat Transfer from a rotating circular cylinder in the steady flow regimes: effect of Prandtl number. Thermal Science Vol. 16, No.1, pp. 79-91, 2012.
13. Golani R, Dhiman AK. Fluid flow and heat transfer across a circular cylinder in the unsteady flow regime. The Int J Engg and Science 2014; 3:08-19.
14. Mahir N, Altac Z. Numerical investigation of convective heat transfer in unsteady flow past two cylinders in tandem arrangements. Int J Heat Fluid Flow 2008; 29:1309-18.
15. Rahman MM, Karim MM, Alim MA. Numerical investigation of unsteady flow past a circular cylinder using 2-D finite volume method. Journal of Naval Architecture and Marine Engineering 2007; 4:27-42.
16. Tuann SY, Olson MD. Numerical studies of the flow around a circular cylinder by a finite element method. Computers and Fluid 1978; 6:219-40.
17. Meneghini JR. Numerical simulation of bluff body flow control using a discrete vortex method. Thesis Report 1993. University of London, UK.
18. Mittal S, Kumar V, Raghuvanshi A. Unsteady incompressible flow past two cylinders in tandem and staggered arrangements. Int J Numer Meth Fluid 1997; 25:1315-44.
19. Abdel-Raouf AM, Galal M, Khalil EE. Heat transfer past multiple tube banks: A numerical investigation. 10th AIAA/ASME Joint Thermophysics and Heat Transfer Conference 2010.