INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X

(Volume2, Issue4)

Available online at: www.Ijariit.com

Flow Past a Rotating Circular Grooved Cylinder

Ashish Kumar Saroj, XPLOCC Technologies, ashish.saroj27@gmail.com
Bharath Reddy, XPLOCC Technologies, chbharathreddy91@gmail.com
Gundu Jayadhar, XPLOCC Technologies, jayadhargundu1392@gmail.com
D.Gokul, Scientist SD, ISRO, gokul.d@shar.gov.in

ABSTRACT: CFD simulations of a two-dimensional steady state flow past a rotating circular grooved cylinder is analyzed in this study. Cylinder of diameter 0.1 m with 8 grooves of 0.01 m was examined at various Reynolds's number (0.1 to 50) and angular velocity (0 to 100 RPS). Incompressible Navier Stokes equation in Ansys Fluent 14.0 was used to examine the flow. The pressure and velocity contours for various Reynolds's number were generated. The result suggested that the flow remains attached to the surface of the cylinder up to the Reynolds number value of 4–5 and the flow pattern was independent of angular velocity at Reynolds's number 45-46 and the cylinder behaved like a stationary cylinder and above these Reynolds's numbers the flow is still two-dimensional, but no longer steady.

Keywords: Grooved cylinder, Angular velocity, Reynolds number, laminar, Incompressible fluid, Wake region.

I. Introduction

Investigations of incompressible flow past a rotating circular cylinder have been conducted by many researchers, based on theoretical, experimental and numerical approaches. Spinning cylinder flows are of great importance in aerodynamics and in design of engineering structures. Rotating cylinders are also devices well recognized for the control of boundary layer flows. There exist many research articles dedicated to the applications of rotating cylinders and in the implementation of other control techniques, such as blowing, suction and surface roughnes Bearman [1], Berger and Willie [2], Gad el Hak and Bushnell [3], Griffin and Hall [4], Sumer and Fredsoe [5] and Zdravkovich [6].The imposition of rotation is used to delay or suppress the propensity for vortex shedding thereby extending the steady flow regime. Indeed, the flow over a cylinder is influenced by a large number of parameters such as the nature of the far flow field (uniform, or shear, or extensional), confined or unconfined cylinder, type of fluid (compressible, incompressible, or non- Newtonian), stationary or rotating cylinder. Low Re-number spinning cylinder flows have been the subject of many studies. High Reynolds number flows over cylinders are complicated because of the coupled action of the shear layer instability and the early development of the fully turbulent attached boundary layer. Also, three dimensional effects are more pronounced compared to laminar flow regimes .In general, the drag coefficient is lower with a sphere with grooved surface than one with a smooth surface in the lower range of the transition region. The drag reduction achievable by controlling airflow over the surface of a structure not only prevents structural destruction, but also enhances the performance of the vehicle, considerably reducing energy consumption with reference to articles Yamagishi Y, Oki M [7, 8] and Lim HC, Lee SJ [9].

Studies (Smith, 1979) indicate that the circulation does not result from the common explanation of the air set into an opposing rotation by the friction of a no-slip wall, as this only occurs in a very thin boundary layer next to the surface. But this motion of the fluid in the boundary layer does affect the manner in which the flow separates from the cylinder. Boundary layer separation is moved back on the side of the cylinder that is moving with the fluid, and is moved forward on the side opposing the main stream. The wake then shifts to the side moving against the main stream causing the flow to be deflected on that side, and the resulting change in free stream flow creates a force on the spinning cylinder. The wake structure is quite similar to that observed for the non-rotating cylinder except for an upward deflection caused by the rotation of the cylinder. The tail of each vortex is engulfed by the one downstream of it. The vorticity is created on the surface of the cylinder because of the no-slip condition on velocity. It is transported to other locations in the flow via advection and diffusion. In steady flow, it is the diffusion mechanism is responsible for transporting the vorticity to the outer flow. Vortex shedding cannot occur if vorticity of sufficient strength is unable to diffuse to the flow outside the closed streamlines. The vorticity decreases as one move away from the cylinder along the solid line. Since the flow, close to the stagnation point, moves quite slowly vorticity may build up with time. A possible cause for this interesting behaviour of flow stability (or instability) is proposed. The strength of vorticity generated on the cylinder surface increases with the increase in spin rate. This is accompanied by an increase in the thickness of the region with closed streamlines around the cylinder.

II. Problem Description

The objective is to study incompressible steady flow of fluid (with uniform velocity U0) past a grooved cylinder of diameter 0.1m (infinitely long in z-direction) rotating with an angular velocity in the clockwise direction. The cylinder is placed in a domain whose outer boundary is a rectangle. The centre of the cylinder is located at the origin of the coordinate system. The free-stream flow is along the x axis. The Reynolds number, Re, is based on the diameter of the cylinder, free stream velocity and viscosity of the fluid.

Specification	Dimensions
Number of grooves	8(45^0 with axis of each other)
Dimension of cylinder	0.1m
Dimension of outer construction cylinder	0.12m
Dimension of grooves	
Bottom Base	0.01m
Height	0.01m
Top base	0.006m
Slant height	0.010444m

Table I. Features of grooves

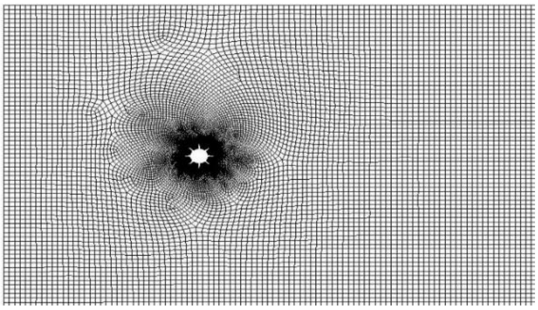

Figure 1. Geometry of the grooved cylinder

A. *Governing Equations*:

The governing equations used in the model are the continuity equations and the incompressible Navier-Stokes equations.

a. Conservation of mass $\Delta . u = 0$

b. Momentum Conservation of momentum

$$\frac{\partial}{\partial t}(\rho v) + div\,(\rho v v) + Q \;=\; 0$$

B. Boundary conditions

- Free-stream values are assigned for the velocity at the upstream boundary.
- At the upstream boundary a Dirichlet type boundary condition for the velocity and pressure specified that prescribes the value of the variable is constant at the boundary.
- On the upper and lower boundaries the component of stress vector along these boundaries and the velocity normal to them are assigned zero values.
- Pressure outlet boundary condition Dirichlet type boundary.
- Backflow is assumed to be to be normal to the boundary.

Conditions for cylinder**:**

- Moving wall with absolute rotational motion is assumed. No slip for shear condition is assumed.

The physically realistic boundary conditions for this flow:

- At the inlet plane: It is the uniform flow in x-direction $U_x = U_0$, $U_y = 0$.
- The top and bottom walls are assumed to be no slip boundaries.

III. Results and Discussion

In this study, the field equations have been solved using FLUENT. The unstructured quad cells of uniform and non-uniform grid spacing were generated. The two dimensional, laminar, segregated solvers was used to solve the incompressible flow on the collocated grid arrangement. Steady solvers have been used in this study. The power law scheme has been used to discretize the convective terms in the momentum equations. The semi-implicit method for the pressure linked equations (SIMPLE) scheme was used for solving the pressure–velocity coupling. FLUENT solves the system of algebraic equations using the Gauss–cell iterative method solver. Relative convergence criteria of 10−6 for the residuals of the continuity and x- and y-component of the momentum equations were prescribed in this work. Furthermore, a simulation was deemed to have converged. When the values of the global parameters had stabilized to at least four significant digits.

For the simplest case of the incompressible uniform flow of Newtonian fluids over an unconfined stationary cylinder, the flow undergoes several transitions with a gradual increase in the value of the Reynolds number. Thus, for instance, the flow remains attached to the surface of the cylinder up to about the Reynolds number value of 4–5.As the value of the Reynolds number is gradually increased, the occurrence of an adverse pressure gradient at some point on the surface of the cylinder results in the separation of flow from the surface there by leading to the formation of the so-called wake region. This region is characterized by the loss of symmetry of the flow field, although the flow is still steady and two-dimensional. The attached twin-vortices grow in size with further increase in the value of the Reynolds number and the flow continues to be symmetric about mid-plane. At about Re = 46–47, when the cylinder is at rest the wake becomes asymmetric and vortices begin to break off alternately from the upper and lower halves of the cylinder respectively thereby resulting in the so-called laminar vortex shedding regime. Under these conditions, the flow is still two-dimensional, but no longer steady and one must seek solutions to the time-dependent equations.

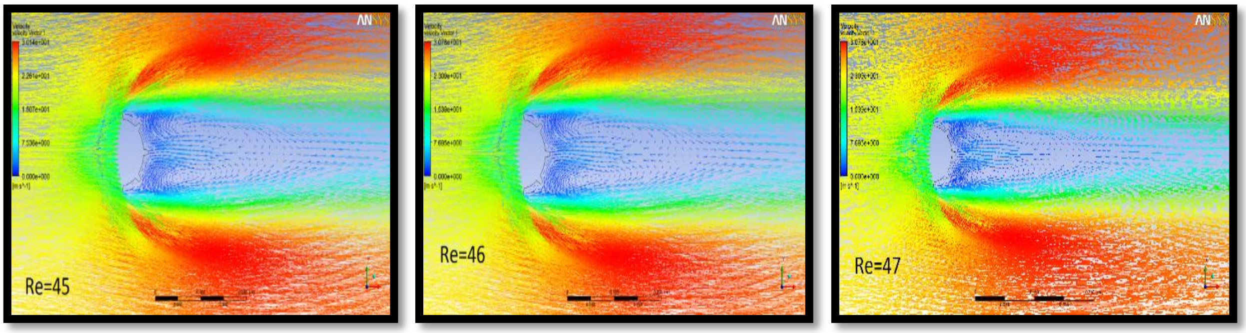

Figure 2 Velocity contours for various angular velocity ω = 0, 50 and 100 respectively

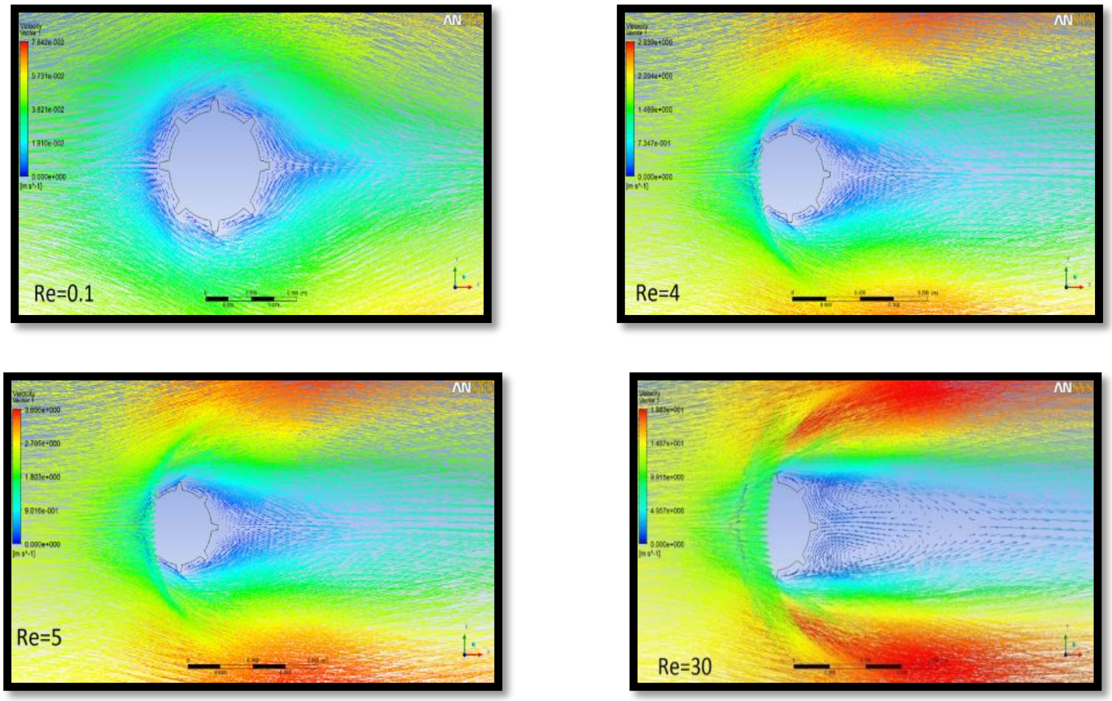

Fig. 3 Velocity contours at various Reynolds numbers (ω = 0)

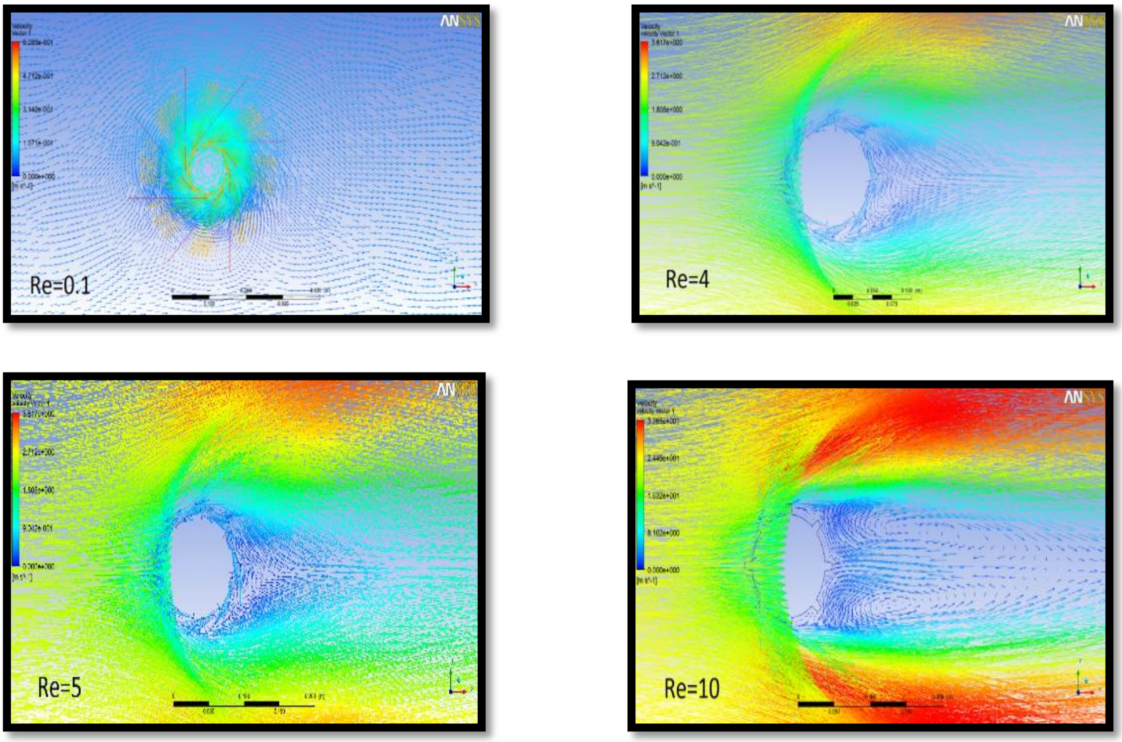

Fig 4 Velocity contours at various Reynolds numbers (ω = 100)

IV. Concluding Remarks

The mesh close to the cylinder is fine enough to resolve the boundary layer in agreement with results from other researchers. The Reynolds number is observed to influence the wake morphology more strongly near the vortex shedding suppression rotation rate. Although numerical solutions can be found for any value of Reynolds number greater than zero .Ideally some experimental measurements could give much more insight into these types of flows. Nevertheless, it appears from the simulations demonstrated up to now, that it is really difficult, if not impossible, to conduct accurate measurements at very high spin rates.

References

[1] B.M. Sumer, J. Fredsoe, Hydrodynamics around Cylindrical Structures, World Scientific, Singapore, 2006.
[2] C.H.K. Williamson, Vortex dynamics in the cylinder wake, Annu. Rev. Fluid Mech. 28 (1996) 477–539.
[3] Lim HC, Lee SJ (2002) flow control of circular cylinders with longitudinal grooved surfaces. AIAA J 40(10):2027-2036.
[4] M.M. Zdravkovich, Flow Around Circular Cylinders, vol. 1: Fundamentals, Oxford University Press, New York, 1997.
[5] M.M. Zdravkovich, Flow Around Circular Cylinders, vol. 2: Applications, Oxford University Press, New York, 2003.
[6] M. Coutanceau, J.R. Defaye, Circular cylinder wake configuration: a flow visualization survey, Appl. Mech. Rev. 44 (1991) 255–305.
[7] V.K. Patnana, R.P. Bharti, R.P. Chhabra, Two dimensional unsteady flow of power-law fluid over a cylinder, Chem. Eng. Sci. 64 (2009) 2978–2999.
[8] Yamagishi Y, Oki M (2005) Effect of The number of grooves on flow characteristics around a circular cylinder with triangular grooves. J Vis 8(1):57-64.
[9] Yamagishi Y, Oki M (2004) Effect of groove shape on flow characteristics around a circular cylinder with grooves. J Vis 7(3):209-216.