International Journal of Current Engineering and Technology
ISSN 2277 - 4106

Available at http://inpressco.com/category/ijcet

Research Article

Computational Investigation of Suction in Taylor-Couette Pump

Sampreeti Jena[a*] and Pandaba Patro[b]

[a] National Institute of Technology, Rourkela-769008
[b] Indian Institute of Technology, Kharagpur – 721302

Accepted 2 Jan. 2013, Available online 1March 2013, **Vol.3, No.1 (March 2013)**

Abstract

In the present paper, numerical simulation of a couette pump is performed to study the flow visualization and suction rate for different rotor configurations. Taylor vortices can be formed by subjecting a liquid to a shear stress within an annular space between a rotating cylinder (with blades or without blades) and a stationary bottom surface. The results are substantiated by FLUENT simulations for better understanding on the flow pattern of the fluid. The suction rate is higher in case of cylindrical rotor without blades.

***Keywords**: Taylor-Couette, Couette pump, rotor blades, Turbulence, CFD*

1. Introduction

Suction is the flow of a fluid into a partial vacuum, or region of low pressure. The pressure gradient between this region and the ambient pressure will propel matter toward the low pressure area. In fluid dynamics, the Taylor-Couette flow comprises a layer of liquid entrapped within the surfaces of two rotating cylinders. A Taylor-vortex reactor consists of two concentrical cylinders, where in most cases, the outer cylinder is static (stator) and the inner rotates (rotor). In a pump this flow mechanism is employed to compress the fluid between the rotating cylinders and transfer it from the inlet to outlet. In this project work, the objective has been to compare the mass flow rates of different rotor configurations (Number and profile of rotor blades).

In a Couette pump, the central shaft rotates with an angular velocity Ω and the outer wall is still. The cylindrical shaft may be replaced by a rotor with blades. In fluid dynamics, the Taylor–Couette flow consists of a viscous fluid confined in the gap between two rotating cylinders. For low angular velocities, measured by the Reynolds number, the flow is steady and purely azimuthal. This basic state is known as circular Couette flow, after Maurice Marie Alfred Couette who used this experimental device as a means to measure viscosity. Sir Geoffrey Ingram Taylor studied the stability of the Couette flow in a ground-breaking paper which has been a cornerstone in the development of hydrodynamic stability theory (G. I. Taylor *et al*, 1923). Taylor showed that when the angular velocity of the inner cylinder is increased above a certain threshold, Couette flow becomes unstable and a secondary steady state characterized by axisymmetric toroidal vortices, known as Taylor vortex flow, emerges. Subsequently increasing the angular speed of the cylinder the system undergoes a progression of instabilities which lead to states with greater spatio-temporal complexity, with the next state being called as wavy vortex flow. If the two cylinders rotate in opposite sense then spiral vortex flow arises. Beyond a certain Reynolds number there is the onset of turbulence. Circular Couette flow has wide applications ranging from desalination to magneto-hydrodynamics and also in viscosimetric analysis. Furthermore, when the liquid is allowed to flow in the annular space of two rotating cylinders along with the application of a pressure gradient then a flow called Taylor-Dean flow arises. Different flow regimes have been categorized over the years including twisted Taylor vortices, wavy outflow boundaries, etc. It has been a well researched and documented flow in fluid dynamics (C. D. Andereck *et al*, 1986).

Different flow patterns can be formed in the annular gap depending on the rotation speed and the resulting centrifugal forces (C. D. Andereck *et al*, 1986). The rotating flow is described by the dimensionless Taylor number T_a.

$T_a = r_i\omega(r_o - r_i)/V$

Taylor vortices are vortices formed in rotating Taylor–Couette flow when the Taylor number (T_a) of the flow exceeds a critical value. For flow in which $T_a > T_{ac}$, instabilities in the flow are not present, i.e. perturbations to the flow are damped out by viscous forces, and the flow is steady. But, as the T_a exceeds T_{ac}, axi-symmetric instabilities appear. The nature of these instabilities is that of an exchange of stabilities (rather than an over stability), and the result is not turbulence but rather a stable secondary flow pattern that emerges in which large toroidal vortices form in flow, stacked one on top of the

* Corresponding Author: Sampreeti Jena

other. These are the Taylor vortices. While the fluid mechanics of the original flow are unsteady when $T_a > T_{ac}$, the new flow, called Taylor–Couette flow, with the Taylor vortices present, is actually steady until the flow reaches a large Reynolds number, at which point the flow transitions to unsteady "wavy vortex" flow, presumably indicating the presence of non-axisymmetric instabilities.

One of the significances of Taylor–Couette flow is that the changes in flow regimes which eventually lead to turbulence. It is hoped that by studying these systems a more general understanding of transitions to turbulence will emerge (A. Y. Weisberg *et al*, 1997).

Without axial flow (Re_{axg}=0), the steady laminar flow in the annular space becomes unstable when the rotational speed of the inner cylinder is above a critical value. The other parameters (physical characteristics of the fluid and radii of the inner and outer cylinders), grouped in the Taylor number, also play a role in the instability. The first transition is characterized by the appearance of toroidal counter-rotating vortices, the vortex flow, which replace the original laminar Couette flow. In that case, the actual size of one vortex corresponds to the gap width. At higher rotating speeds, a second transition occurs where periodic azimuthally waves are superimposed on the vortices. Despite the number of published works, the understanding of the inherent complexities of the Taylor–Couette system is far from being complete (R. C. Giordano *et al*, 1998).

In earlier works (O. Richter *et al*, 2008), it has been shown that flow and mixing characteristics of a continuous flow Taylor-vortex device can be altered by modifications of the rotor geometry. In this study, we investigated novel rotors furnished with ribs and compared them with a conventional cylindrical rotor over a wide range of hydrodynamic conditions. The objective of this project was to determine the most ideal rotor blade configuration to generate maximum suction. The 2D, turbulent and steady state simulations are performed using commercial software Fluent 6.3.

The various configurations that have been considered are:-

Case 1: Rotor with straight standard number of blades (No. of blades=5).

Case 2: Rotor with both double curved sided blades (No. of blades=5).

Case 3: Rotor with single curved sided blades (No. of blades=5).

Case 4: Rotor with more number of blades (No. of blades=8).

Case 5: Rotor with larger hub diameter and shorter blade length (No. of blades=5)

Case 6: Rotor with single rotating solid cylinder

2. Governing equations

The continuity equation is

$$\nabla.\bar{U}=0 \tag{1}$$

The momentum conservation equation is

$$\nabla.(\rho\bar{U}\otimes\bar{U})=-\nabla p+\nabla.[\mu_{eff}(\nabla\bar{U}+(\nabla\bar{U})^T)] \tag{2}$$

The effective viscosity is defined by

$$\mu_{eff} = \mu_{lam} + \mu_{tur}$$
$$\mu_{tur} = \rho C_\mu k^2/\mathcal{E}$$
$$C_\mu = 0.09$$

The modified pressure p is expressed as

$$P = p + \tfrac{2}{3}\rho k + \rho r.\bar{g} \tag{3}$$

The standard k-Ɛ model is used to model the turbulence.

$$\nabla.(\rho\bar{U}k) = \nabla.[(\mu_{lam} + \mu_{tur}/\sigma_k)\nabla k]+ \overline{p} - \rho\mathcal{E} \tag{4}$$

$$\nabla.(\rho\bar{U}\mathcal{E})=\nabla.[(\mu_{lam} +\mu_{tur}/\sigma_k)\nabla\mathcal{E}]+(C_{\varepsilon1}\overline{p} - C_{\varepsilon2}P\mathcal{E})\mathcal{E}/k \tag{5}$$

The turbulent production due to shear is

$$\overline{p} = \mu_{tur}\nabla\bar{U}.(\nabla\bar{U}+(\nabla\bar{U})^T) \tag{6}$$

The closure coefficients are:

$C_{\varepsilon1}$=1.44
$C_{\varepsilon2}$=1.92
σ_k=1.0
σ_ε=1.3

Boundary conditions:

The inlet and outlet are pressure outlet with pressure = 101325 Pa

The outer cylinder and partition between inlet and outlet are defined as wall.

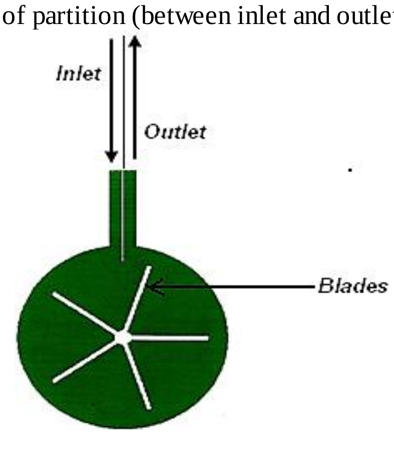
The rotating blades are represented as moving walls with angular velocity ω.

Numerical procedure

The 2D geometry for the Taylor Couette- flow was considered.

The geometry was created and discretized using GAMBIT.

The dimensions used are:
Hub diameter = 2m
Blade length =9m
Diameter of outer walls =26m
Width of inlet and outlet =1.4m
Thickness of partition (between inlet and outlet) =0.2m



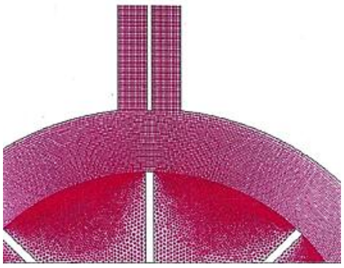


Fig.1 The computational domain and the grid structure used in the simulation

The commercial software FLUENT (version 6.3) was used in all the studies. A pressure based solver was used for solving the momentum equations. The second order upwind scheme was used to discretize the momentum equations as well as turbulence parameters. For the pressure velocity coupling, phase-coupled SIMPLE scheme was used. K- $\mathcal{E}$ Turbulence model was used with standard wall function. The density and viscosity of the working fluid were set at 1160 kg/m3 and 0.28kg/m-s respectively. At the inlet, the values of backflow turbulent intensity and back flow viscosity ratio were maintained at 2% and 10% respectively. At the outlet side, backflow turbulent intensity and viscosity were fixed at 5% and 10% respectively. The rotor edges were recognized as rotating walls and given an angular speed of 53 rad/sec. In the solution controls, both flow and turbulence conditions were activated. The mass flow rate (mass influx) at the inlet was found out. It is an indicator of the suction generating capacity of the corresponding configuration. The pressure difference created as a result of suction which drives the fluid into the pump cavity through the inlet is also measured by carrying out surface integral with respect to static pressure.

3. Results and discussions

Velocity profiles

The velocity contours are presented for different rotor configurations. The mass flow rates are found at outlet for each configuration. Angular velocity is maintained constant at 53 rad/sec.

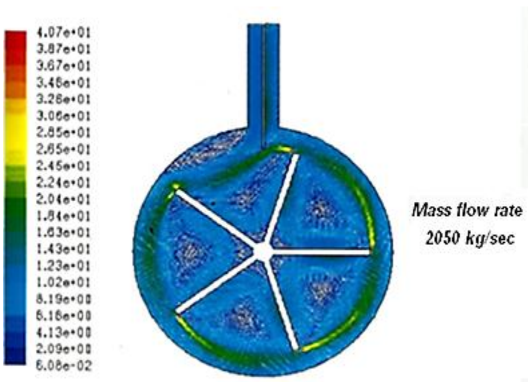


Fig 2. Rotor with 5 straight blades

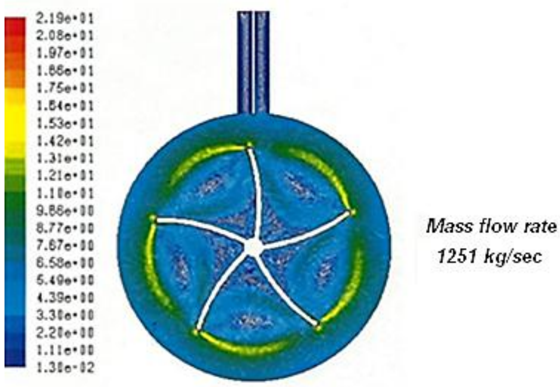


Fig 3. Rotor with 5 curved blades (Blade profile, $y=1.2^x$)

Fig 4. Rotor with 5 curved blades (Blade profile, $y=1.5^x$)

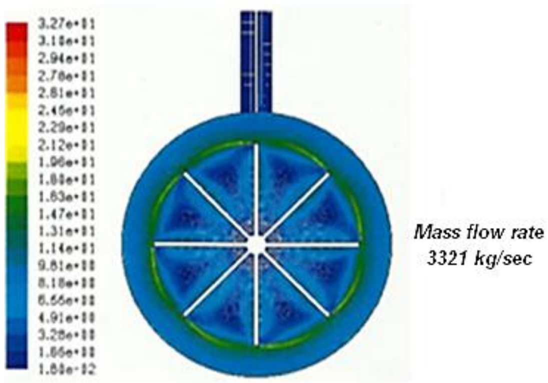


Fig. 5 Rotor with norrower and more number of blades

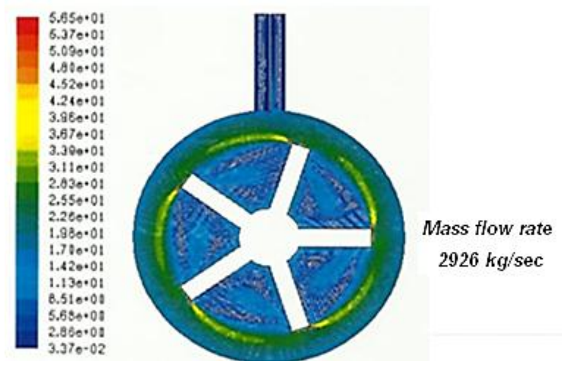


Fig. 6 Rotor with larger hub diameter diameter and shorter and thicker blades. Hub diameter increased from 1 unit to 3 units, Blade length is decreased 9 units to 7 units.

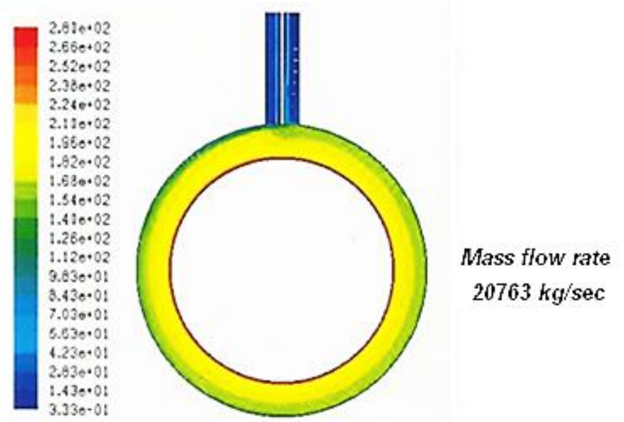


Fig. 7 Rotor with rotating solid cylinder without blades

It is seen from the above results that when the hub diameter is increased and the blades are made shorter (fig. 6), the suction rate increases. When the number of blades is raised and they are made narrow (fig. 5), the rise in mass flux rate is still higher. When the blades are made curved (fig. 3), the mass flow rate is drastically lower than that with straight blades (fig. 1). If the blades are curved only on a single side (fig. 4), the mass flux is further reduced. The suction effect produced is maximum if a single rotating cylinder is used, representing hypothetically, a rotor with infinite number of blades.

When the number of blades is increased, the energy imparted to the fluid per revolution of the rotor increases. This leads to higher inlet and outlet velocities and hence higher mass flux. In case of a rotating cylinder, the fluid particles adjoining the surface can easily follow the motion of the cylinder periphery. But in case of the rotor with blades, the fluid particles are unable to adhere to the rotating blade profile, leading to flow separation and eddy formation. This leads to decrease in energy as well as flow rate. In case of the rotor with curved blades, this problem further exacerbates and the low pressure vortex (eddy) formation results in a mass flow rate even lower than that of the rotor with straight blades.

Effects of the ribs are immobilisation and stabilisation of the toroidal vortices, and a shift of the onset of turbulence to higher Ta numbers. The analysis of the mixing characteristics, in particular, demonstrated the benefits of ribbed as compared to cylindrical rotors. The use of ribbed rotors enables the intensification of micro-mixing while at the same time macro-mixing is reduced considerably at Ta>130. We anticipate that Taylor-vortex reactors with ribbed rotors can be useful for process intensification in many applications.

Conclusions

Two dimensional, steady and turbulent simulation of Taylor- Couette flow was performed for different rotor blade geometry as well as for solid cylinder rotor. It was found from the simulations that the maximum suction or mass influx is generated in case of the solid rotating cylinder. Some of the concluding remarks are:

- Using straight blades yields better results than curved blades.
- Increasing the number of spokes attached to the hub produces more mass flux.
- Increasing the hub diameter while decreasing produces better suction.
- Increase in speed of rotation results in increase in mass flow rate as well as pressure difference across the inlet and outlet.

Nomenclature

p Pressure (Pa)
r Radius
Re Reynolds number
Ta Taylor number
U Velocity (m/s)

Greek Symbols

ε Turbulent energy dissipation rate (m^2/s^3)
k Turbulent kinetic energy (m^2/s^2)
ρ Density (kg/m^3)
μ Viscosity (kg/ms)
ω Angular velocity
σ fluid normal stress

Subscripts

i Inner
o Outer
T Transpose
tur Turbulent
lam Laminar

References

G. I. Taylor (1923), Stability of a Viscous Liquid contained between Two Rotating Cylinders, *Phil. Trans. Royal Society A*, vol. 223, pp. 289–343.

C. D. Andereck, S. S. Liu, and H. L. Swinney (1986), Flow regimes in a circular Couette system with independently rotating cylinders, *Journal of Fluid Mechanics*, vol. 164, pp. 155–183.

A. Y. Weisberg, I. G. Kevrekidis, and A. J. Smits (1997), Delaying Transition in Taylor–Couette Flow with Axial Motion of the Inner Cylinder, *Journal of Fluid Mechanics*, vol. 348, pp. 141–151.

R. C. Giordano, R. L. C. Giordano, D. M. F. Prazeres, and C. L. Cooney (1998), Analysis of a Taylor–Poiseuille vortex flow reactor – I: flow patterns and mass transfer characteristics, *Chem. Eng. Sci.*, vol. 20, pp. 3635–3652.

O. Richter, H. Hoffmann, and B. Kraushaar-Czarnetzki (2008), Effect of the rotor shape on the mixing characteristics of a continuous flow Taylor-vortex reactor, *Chem. Eng. Sci.*, vol. 63, pp. 3504–3513.