Proceedings of the International Conference on Heat Transfer and Fluid Flow
Prague, Czech Republic, August 11-12, 2014
Paper No. 11

Characterization of Laminar Flow and Power Consumption in a Stirred Tank by a Curved Blade Agitator

Amine Benmoussa, Mohamed Bouanini, Mebrouk Rebhi
University of Bechar, Energarid Laboratory
BP417 Route de Kenadsa, 08000 Bechar, Algeria
benmoussa.a@hotmail.fr; mbouanini@yahoo.fr; merebhi@yahoo.fr

Abstract **-** Many operations in process industries as chemical, biotechnological, pharmaceutical, petrochemical, and food processing that are performed in stirred tanks or in mechanically agitated vessels. Therefore determining the level of mixing and overall behaviour and performance of the mixing tanks are crucial from the product quality and process economics point of views. The most fundamental needs for the analysis of these processes from both a theoretical and industrial perspective is the knowledge of the hydrodynamic behaviour and the flow structure in such tanks. Depending on the purpose of the operation carried out in mixer, the best choice for geometry of the tank and agitator type can vary widely. Initially, a local and global study namely the velocity and power number on a typical agitation system agitated by a mobile-type two-blade straight (d/D=0.5) allowed us to test the reliability of the CFD, the result were compared with those of experimental literature, a very good concordance was observed. The stream function, the velocity profile, the velocity fields and power number are analyzed. It was shown that the hydrodynamics is modified by the curvature of the mobile which plays a key role.

Keywords**:** Agitated tanks, Curved blade agitator, Newtonian fluid, Laminar flow, CFD modelling, Finite volume method.

1. Introduction

The manufacture and development of materials very often involves agitation or mixing, either in industrial level or in laboratory (Nagata, S 1975). Mixing operations involving highly viscous Newtonian and Non-Newtonian fluids are widely used for various operations within a wide range of industries including the chemical, pharmaceutical, biotechnological, food process and petroleum industries.

The optimum design of a stirred tank for minimum capital and running costs depends on the desired production rate with a specified product's properties and it is achieved by, for example, a correct choice of tank and impeller geometry, rotational speed and location, of fluid addition and subtraction. A detailed knowledge of power and velocity distribution of the stirred tank configurations is therefore required (Ameur et al 2011).

The fine knowledge of the hydrodynamics structures of flows in the agitated vessel permits to understand and to fear the phenomena of transfer that develops and possibly of their mutual interactions. It also permits to improve the performances of the mobile of agitation set at work, by the amelioration of the geometric conditions, and optimal operations insure simultaneously the improvement of the quality of mixture and the economy of energy (Bouanini et al., 2008).

In general, stirred vessels have been evaluated over the years through experimental investigation for a number of different impellers, vessel geometries, and fluid rheology. Such an approach is usually costly and sometimes is not an easy task. With computational fluid dynamics (CFD), we can examine various parameters contributing to the process with less time and expense, a task otherwise difficult in experimental techniques. During the last two decades, CFD has become an important tool for understanding the flow phenomena (Armenante et al., 1997), developing new processes, and optimizing existing processes (Sahu et al., 1998).

The capability of CFD tools to forecast the mixing behaviour in terms of mixing times, power consumption, flow pattern and velocity profiles is considered as a successful achievement of these methods and acceptable results have been obtained.

The present work is a contribution to hydrodynamic of flows induced by curved blades agitator in mechanically agitated vessel. Our objective is to provide a complete knowledge of the structures flow.

2. Numerical Model

2. 1. Mixing System

The stirred vessel consists of a cylindrical flat bottom unbaffled tank of diameter D agitated by a curved blades agitator of diameter d positioned at the centre of the tank rotating around a shaft of diameter da. The blade thickness is e. The geometrical ratios used are *d/D=0.508, da/D=0.05*.

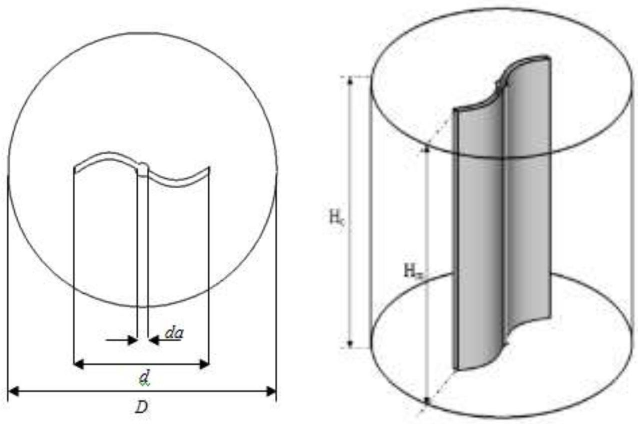


Fig.1. Mixing system

2. 2. Governing Equation

The governing equations of an incompressible viscous flow in agitated vessel are written as follow:

Mass conservation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

Momentum conservation

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\pi}{2}\left(\frac{d}{D}\right)^2 \frac{1}{Re}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\pi}{2}\left(\frac{d}{D}\right)^2 \frac{1}{Re}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

2. 3. Numerical Methodology

In CFD method the first step is to divide the flow geometry into smaller domains, grids, through a number of available discretization methods. Then differential equations governing fluid flow are approximated by a coupled algebraic set of equations which are solved by using a numerical tool along with appropriate boundary conditions.

In this work the numerical simulation is conducted using the commercial Computational Fluid Dynamics code in which a finite volume method developed by Patankar (Patankar S. V 1980) is implemented. A second order scheme is used for the pressure and for the momentum equations. The CFD code has been used to solve, in Cartesian co-ordinates, the continuity and momentum equations for a laminar flow.

Resolution of the algebraic equations was performed using the semi-implicit algorithm pressure linked equation (SIMPLE) with a second-order upwind discretization scheme. Constant boundary conditions have been set respecting a rotating reference frame (RRF) approach. Here, the impeller is kept stationary and the flow is steady relative to the rotating frame, while the outer wall of the vessel is given an angular velocity equal and opposite to the velocity of the rotating frame.

3. Validation

Initially before the presentation of our results, it is necessary to validate our simulation; for this task, we have compared our results with previous work on a two-blade paddle agitator straight (d/D=0.5, da/D=0.05) (Youcefi 1993) and (Bertrand 1983). We validated our work according to two parameters the local and global.

Depending on the locale that is the tangential velocity on the blade and its extension, we have validated with the experimental study (Youcefi 1993), as shown in Figure 2 good agreement is demonstrated. Following the global setting that is the power number *Np*, we have validated with the experimental study (Bertrand 1983), as shown in Figure 3, good agreement is clearly confirmed.

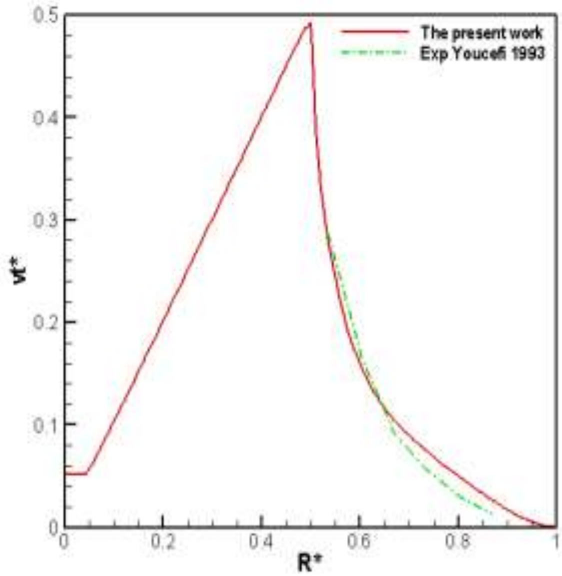


Fig.2. tangential velocity on the blade and its extension.

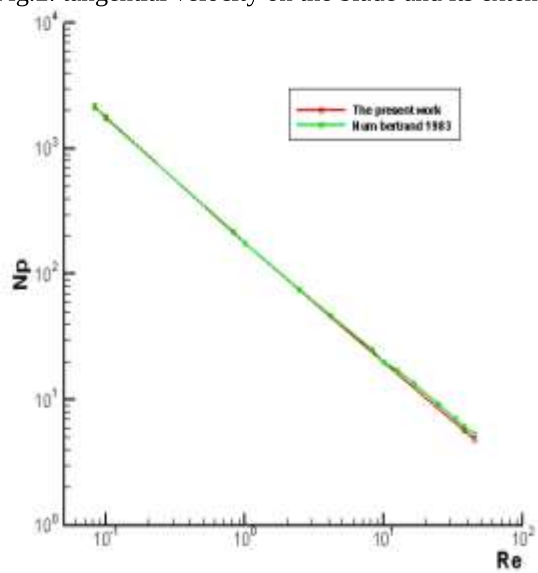


Fig.3. Power number as Function of the Reynolds Number.

4. Results and Discussion

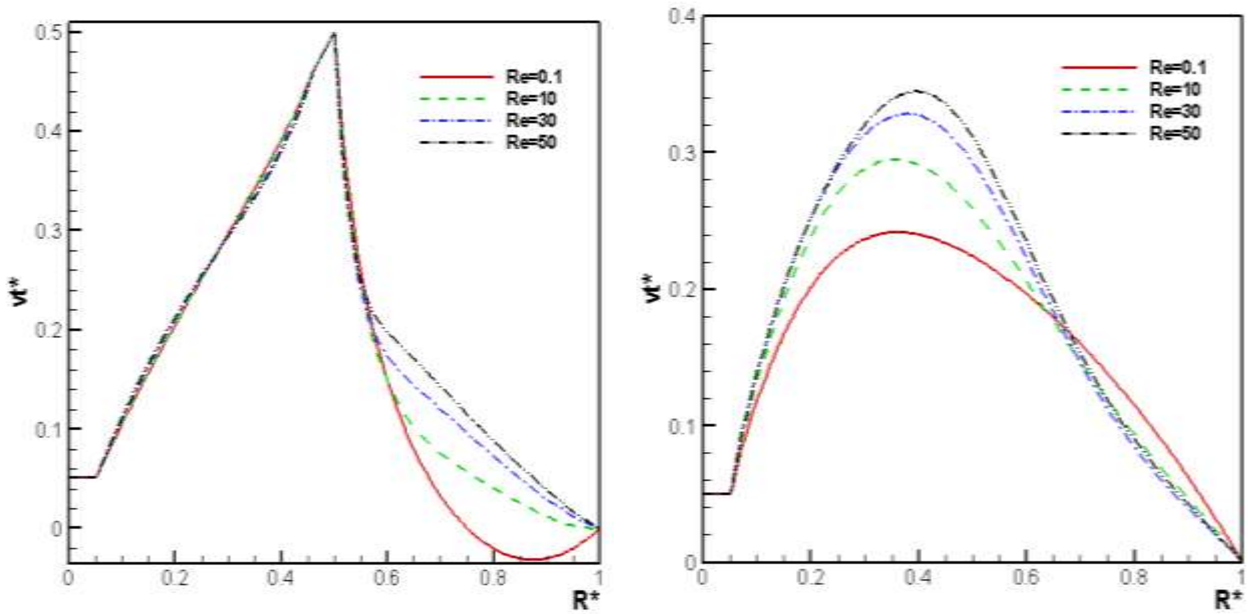
4. 1. Influence of Inertia on Evolution of Velocity Components

We chose a curved two-bladed size d/D=0.5, while varying the Reynolds number, which is defined by:

$$Re = \frac{\rho N d^2}{\mu} \tag{4}$$

Where N is rotational speed

Then we analyzed the effect and influence of inertia on the velocity components, and finally a presentation of the following results:



(a): Impeller plane (b): Median plane

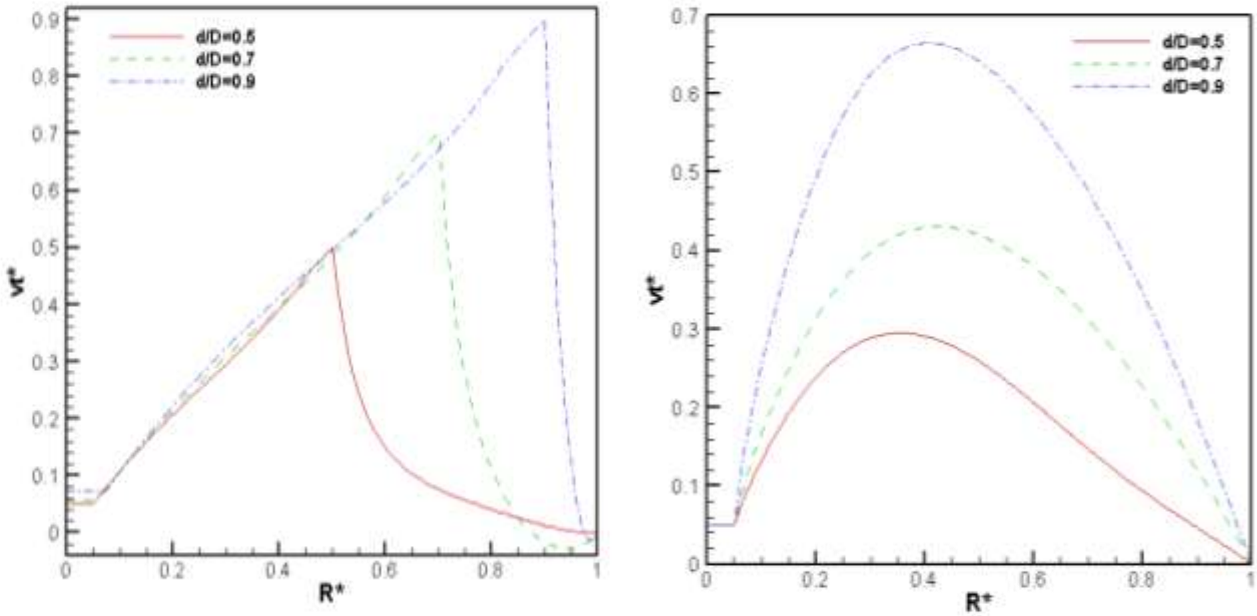
Fig.4. Tangential velocity for different Reynolds numbers.

Figure 4.a shows the variation of the tangential velocity on the impeller plane. Note for all values of Reynolds, the velocity is equal to the diameter of the agitator, it takes the maximum value at the external end of the blade (R * = 0.5) then begins to decrease from the blade tip until becoming negligible on the tank wall. The decay curves are faster with weak Reynolds number.

In figure 4.b, we can see that the curves have the same departure and arrival point and the velocity profile has a parabolic shape. It is a relative proportionality to Re number.

4. 2. Influence of the Blade Diameter on Evolution of Velocity Components

To perform this test, three geometrical configurations have been realized: d/D= 0.5, 0.7 and 0.9 respectively:



(a): Impeller plane (b): Median Plane

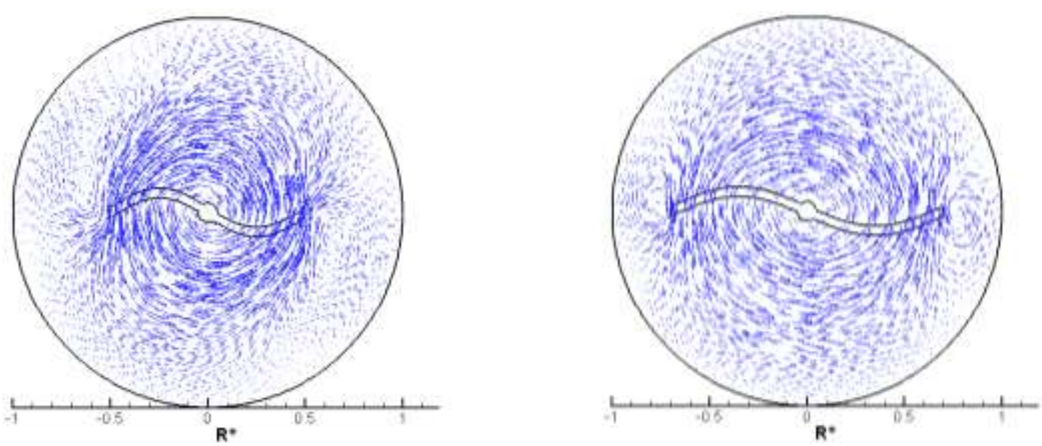
Fig.5. Tangential velocity for different blade diameter, Re=10

At the impeller plane, we remark that the velocity maximal is located at the paddle tip for any of the cases investigated, therefore, the maximum value of the velocity is proportional to the size of agitator; We can see negative velocity values for size 0.7 0.9 near the wall at the point R*=0.9, it results in the existence of a tourbillion witch the flow will be delayed in this area.

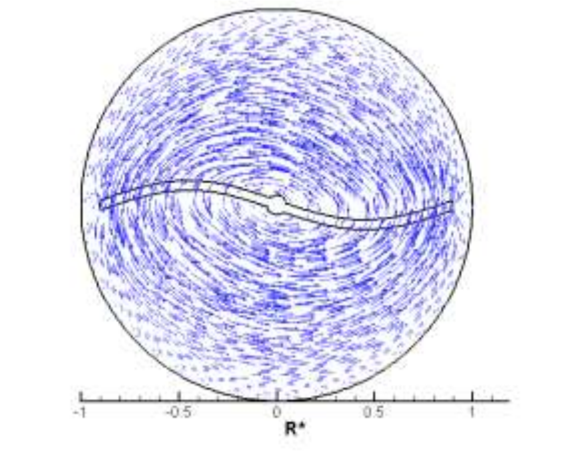
At the median plan, the figure shows that the velocity profile has a parabolic shape with a local summit in order to 0.3 for the size 0.5, 0.45 for the size of 0.7 and a maximum of 0.65 for 0.9, and the curves have the same departure and arrival point.

4. 3. Influence of the blade diameter on evolution of velocity fields

We chose a curved bladed agitator size d/D=0.5, setting the Reynolds number on 10, then we varied the blade diameter,



(a) (b)



(c)

Fig.6. Velocity fields for different blade diameter: (a): d/D=0.5, (b): d/D=0.7, (c): d/D=0.9

In figure 6, we can remark that the increase in blade diameter decreases the volume of the vortex, The size of the vortex is inversely proportional to the blade diameter.

4. 4. Power Number

The stirring power is a global variable accessible by the experience and fairly easy to measure. As we know that the number of power does depend only of the geometry of the stirred system and the number of Reynolds:

$$Np = \frac{P}{pN^3d^5} \quad (4)$$

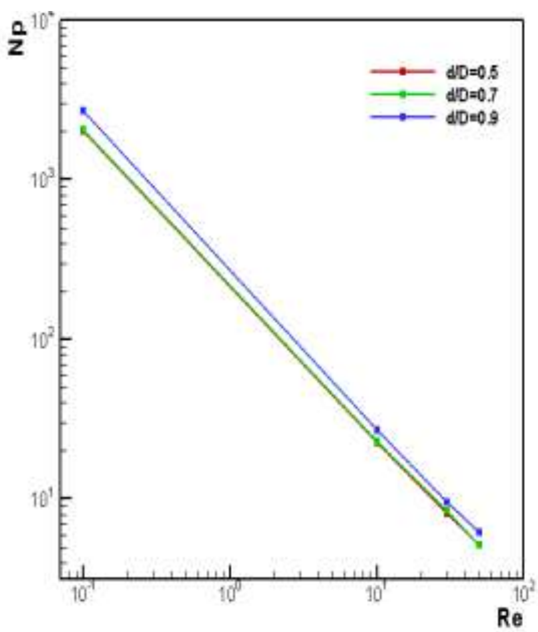


Fig.7 Power number for different Re number

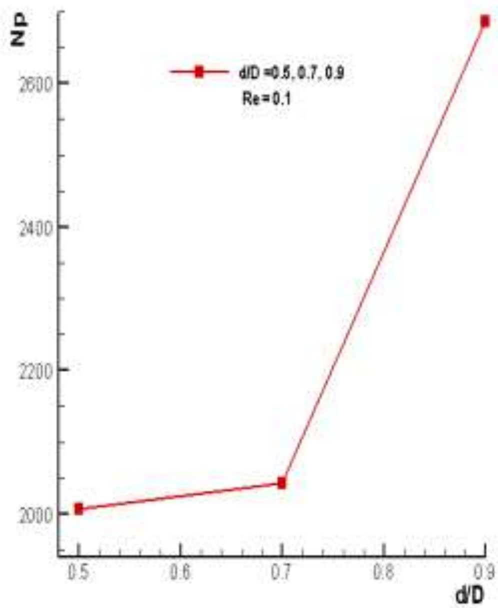


Fig.8 Power number for different blade diameter.

5. Conclusion

Our work is a contribution to the study of the difficulties encountered in the phenomena of agitation tank environments Newtonian rheology.

Note that such problems arise in different industrial sectors of great economic, industrial plastics, inks and paints, cosmetics and beauty products, food industries, where a numerical characterization of laminar flow and power consumption in agitated vessel with curved blade agitator has been developed in order to provide a physical analysis of mixing in Newtonian fluid. For each configuration, the power consumption was calculated.

After examination it has been proven that the size of the impeller plays a great role on the flow structure, the increase in blade width is beneficial to enlarge the well stirred region but that requires more power consumption.

This study has demonstrated that the parametric analysis of laminar flow for viscous fluid in geometrics involved in industrial process, such as agitated tanks, can be efficiently handled through CFD.

References

Nagat, S, (1975). “Mixing principles and applications, KODANSHA LTD, Tokyo.

H.Ameur, M.Bouzit, M.Helmaoui, (2011). “Numerical study of fluid flow and power consumption in a stirred vessel with a Scaba 6SRGT impeller,” Chemical and process Engineering, 32 (4), 351-366.

M. Bouanini, A.Youcefi, B.Mebarki, L.Rahmani, O.Hami, B.Draoui, (2008).“ Numerical model for laminar flow in agitated vessel by inclined blades anchor” I.RE.M.E, vol 2, N3, 364–373.

Patankar S. V. (1980)“Numerical heat transfer and fluid low.” (Mc-Graw Hill).

A. Youcefi, “Experimental study of fluid flow viscoplastic around a tow-blade stirrer in a stirred tank”, Ph.D Thesis, National Polytechnic Institute, Toulouse.

J. Bertrand. (1993) “Agitation of viscous case of paddle, anchor and gate,” PhD Thesis, Ntional Polytechnic Institute, Toulouse.

P. M. Armenante, C. Luo, C. C. Chou, I. Fort and J.Medek. (1997) “Velocity profiles in a closed, unbaffled vessel: Comparison between experimental LDV data and numerical CFD prediction,” Chem.Eng. Sci. 52 (20), 3483-3492.

A. K. Sahu, P. Kummer and J. B. Joshi. (1998). “Simulation of flow in stirred vessels with axial flow impellers: zonal modeling and optimization of parameter,” Ind. Eng. Chem. Res. 37 (6), 2116-2130.