Proceedings of Engineering and Technology - PET
2ème conférence Internationale des énergies renouvelables CIER-2014

Hydrodynamics of yield stress fluid flow in Agitated Vessel with Circular Gate Impeller

Amine benmoussa#1, Lakhdar Rahmani#2, Mebouk Rebhi#3

University of Bechar, Energarid Laboratory
BP417 Route de Kenadsa, 08000 Bechar, Algeria
[1]benmoussa.a@hotmail.fr
[2]rahmani_l1@yahoo.fr
[3]merebhi@yahoo.fr

Abstract— The mixing and agitation of fluid in agitated vessel is one of the most important unit operations for many industries such chemical, biotechnological, pharmaceutical, petrochemical, cosmetic, and food processing. Understanding mixing mechanisms, which crucial to industrial scale-up still remains difficult especially in the case of non- Newtonian fluid, there is a wide variety of behavior possible in the area of these fluids.

In the present study, the hydrodynamic characteristics of incompressible yield stress fluid in a cylindrical vessel by a circular gate impeller have been investigated. The results were compared with literature on straights blades gate, a very good concordance was observed. The velocity profile, the velocity fields and power consumption was analyzed.

***Keywords*— 2D Numerical study, Agitated vessel, Gate impeller, Laminar flow, yield stress, Finite Volume Method, CFD modeling.**

I. INTRODUCTION

Viscoplastic fluids are an important class of non-Newtonian fluids. These fluids flow only when the shear stress is above a certain threshold, the yield stress, and this leads in particular to dead zones in the flow witch lower mixing efficiency [1]. Agitation of such fluids results in the formation of a zone of intense motion around the impeller (the also called the cavern) with essentially stagnant regions elsewhere [2].

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 [3].

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 [4].

In general, stirred vessels have been done 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 [5], developing new processes, and optimizing existing processes [6].

The capability of CFD tools to forecast the mixing behavior 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.

Our objective is to provide a complete knowledge of the structures flow.

II. NUMERICAL MODEL

A. *Mixing System*

The system consists of a cylindrical flat bottomed vessel of diameter D equipped with a circular gate impeller of diameter d positioned at the centre of the tank rotating around a shaft of diameter d_a, with clearance to the wall w=0.02.

The geometrical ratios used are d/D=0.96, da/D=0.023, and L/D=0.067.

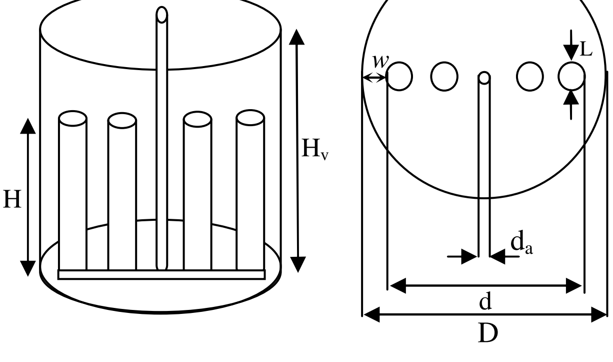


Fig.1 Mixing system

B. *Comportment law of Fluid*

To study the yield stress fluids which constitute the purpose of this paper, mixing systems are dealt with by numerical simulation using Viscoplastic fluids modeled by a Bingham law:

$$\overline{\overline{D}} = 0 \; for \; \left\|\tau\right\| < \tau_0 \tag{1}$$

$$\overline{\overline{\tau}} = \left(\frac{\tau_0}{\dot{\gamma}} + \eta_\infty\right)\overline{\overline{D}} \; for \left\|\overline{\overline{\tau}}\right\| > \tau_0 \tag{2}$$

Where $\overline{\overline{\mathrm{D}}}$ and $\overline{\overline{\tau}}$ are, respectively, the rate of strain tensor and the stress tensor. τ_0 is the yield stress and the shear rate is defined as $\dot{\gamma} = \sqrt{2 \,\mathrm{tr}\, \overline{\overline{\mathrm{D}}}^2}$ where tr stands for the trace.

$\overline{\overline{D}}$ and $\overline{\overline{\tau}}$ are defined as:

$$\overline{\overline{D}} = 1/2\left(\nabla V + \nabla V^T\right)$$

$$\left|\overline{\overline{\tau}}\right| = \sqrt{\frac{1}{2}\overline{\overline{\tau}} : \overline{\overline{\tau}}^T}$$

According to equations (1) and (2), the flow domain for a Bingham fluid is characterised by two distinct regions. In the regions where $\left|\overline{\overline{\tau}}\right| \prec \tau_0$ the material behaves a rigid solid, and in the regions where $\left|\overline{\overline{\tau}}\right| \succ \tau_0$ the material flow with an apparent viscosity η_{ap} .

$$\eta_{ap} = \eta_\infty + \frac{\tau_0}{\dot{\gamma}} \tag{3}$$

The major difficulty with the constitutive equation (1) and (2) when used for numerical simulation is the discontinuity associated with infinite value of the viscosity when $\left|\tau\right|$ approaches τ_0 . This is reached closed to the yield surface delimiting plug regions. Some numerical works have been published on methods adapted to the initial set of equations {(1), (2)} completed by continuity equation and momentum equation in stress formulation. But most of the published works on this topic use modified versions which consist in replacing equations. (1)+(2) by a unique and continuous equation written with a variable viscosity η_{ap} [7]:

$$\tau = 2\eta_{ap}\overline{\overline{D}} \tag{4}$$

C. *Governing Equation*

Incompressible and isothermal flow of non-Newtonian fluids is governed by the law of conservation of mass and momentum expressed in velocity pressure stress formulation which is necessary to add a behavior law for the fluid to close the system. It is possible to use the Navier-Stokes equations insofar one takes into account the existence of a field of viscosity and its gradient. Under these conditions, the governing flow equations are [8]:

Continuity

$$\rho\frac{\partial \mathrm{V}}{\partial t} + \nabla(\rho\,\vec{\mathrm{V}}) = 0 \tag{5}$$

Momentum

$$\rho\left[\frac{\partial \vec{\mathrm{V}}}{\partial t} + \left(\vec{\mathrm{V}}.\nabla\right)\vec{\mathrm{V}}\right] = \vec{\mathrm{f}} - \nabla \mathrm{P} + \nabla.\tau = 0 \tag{6}$$

Introducing the second invariant of the stress tensor, the equation 9 becomes

$$\rho\frac{\partial \vec{\mathrm{V}}}{\partial t} + \left(\vec{\mathrm{V}}.\nabla\right)\vec{\mathrm{V}} - \eta(\dot{\gamma})\Delta \mathrm{V} - 2\,\dot{\mathrm{D}}\cdot\nabla\eta(\dot{\gamma}) + \nabla \mathrm{P} = 0 \tag{7}$$

The boundary conditions are:

- On the impeller: *V=0*
- On the vessel: *V=-ωR.*

III. RESULTS AND DISCUSSION

Initially before the presentation of our results, it is necessary to validate our simulation; for this task, we have compared our tangential velocity with previous works on a straights blades gate stirrer. Fig 2 gives an example of comparison between our results and a numerical work of Rahmani [10]. The results show a very good concordance.

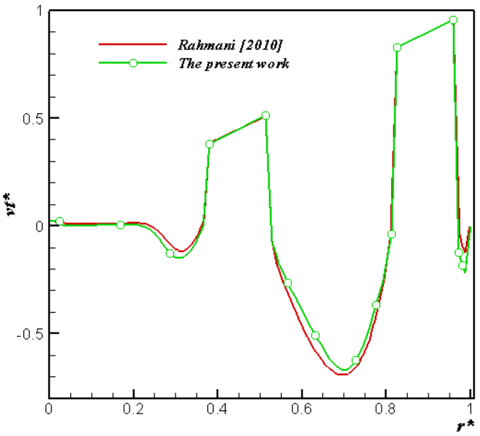


Fig.2 Tangential velocity for on the impeller plan, Re=13.8

A. *Effect of inertia on Evolution of velocity component*

Figure 3 and 4 shows the tangential velocity profiles on the impeller plane and the median plan respectively.

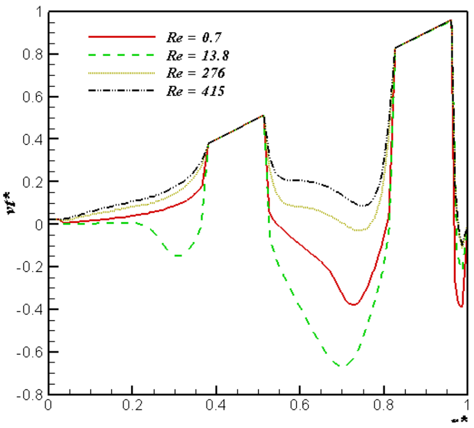


Fig.2 Tangential velocity for different Reynolds number on the impeller plan

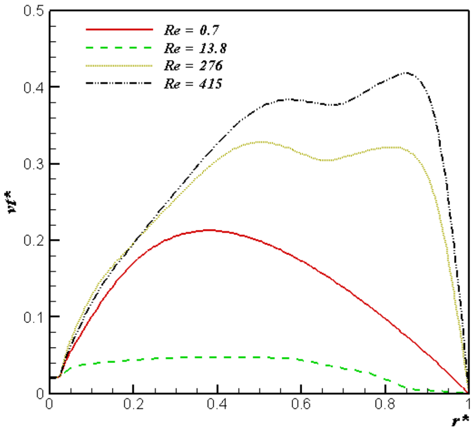
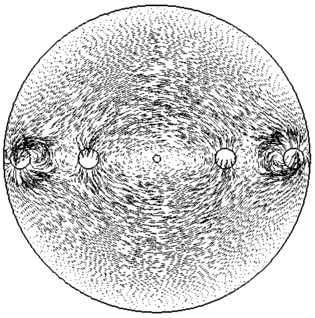


Fig.3 Tangential velocity for different Reynolds number on the median plan

In the impeller plan the component of tangential velocity increases from the centre to the first blade, a linear increase between the two extremities of the first blade then a decrease until a relative minimum, then an increase until the Internal end of the second blade, a linear increase between the extremities of the second blade. At the end a sharp decline up till the wall which is null. Also we notice that between the axis of the agitator and the first blade, and the two blades of agitator, the velocity increases proportional with Reynolds number. On the median plan we can see that the curves have the same departure and arrival point. It is a relative proportionality to Reynolds number.

B. *Effect of inertia on Evolution of velocity field*

The figure 4 Represent the velocity fields for four different numbers of Reynolds. We can observe on these vector maps that the yield stress effect is more drastic in the case of low inertia. However, if the inertia is increased, the rest of the tank began to be sheared a little small to move towards the case of a Newtonian fluid the effects of plasticity then became negligible.



(a)

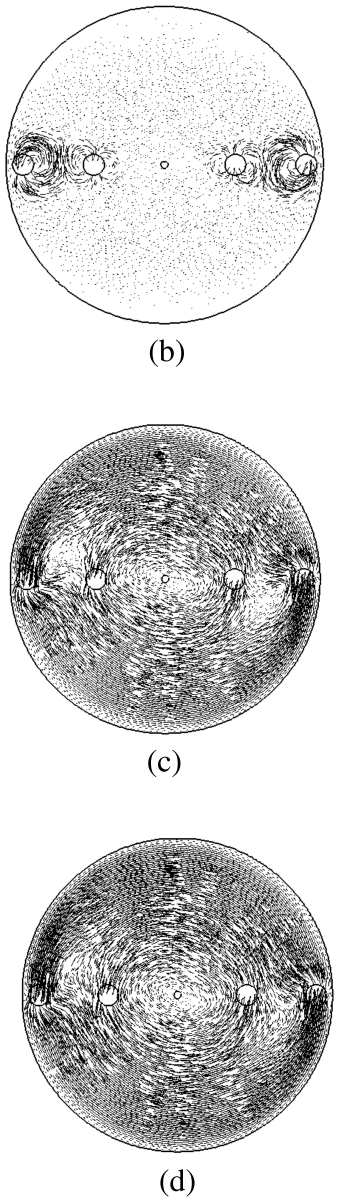


Fig.4 Velocity fields for different Reynolds numbers
(a): Re=0.7, (b): Re=13.8, (c): Re=276, (d): Re=415

C. *Effect of yield stress*

To analyze the influence of yield stress, we study the flow of tree Bingham fluids of yield stress τ0 equals 0, 1 and 30 Pa. These values correspond to dimensionless numbers of Hedström 0, 8294 and 248832. The value of He equal to zero corresponds to the Newtonian case. The speed of rotation N was set at 1 rpm. The results are shown in fig 4, 5 and 6. Conversely to inertia effects cited in the previous paragraph, we observe that for lower values of Hedström number (He=0), the velocity on the blade plane is significantly increased, on the other hand of the median plane is practically null.

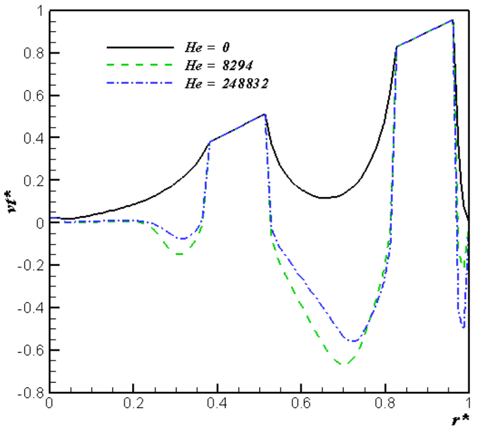


Fig.4 Tangential velocity for different Hedström numbers on the impeller plan

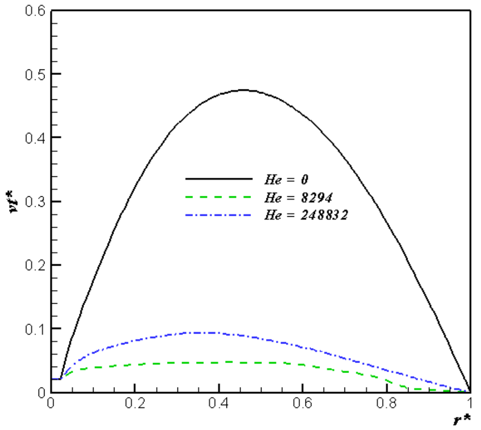
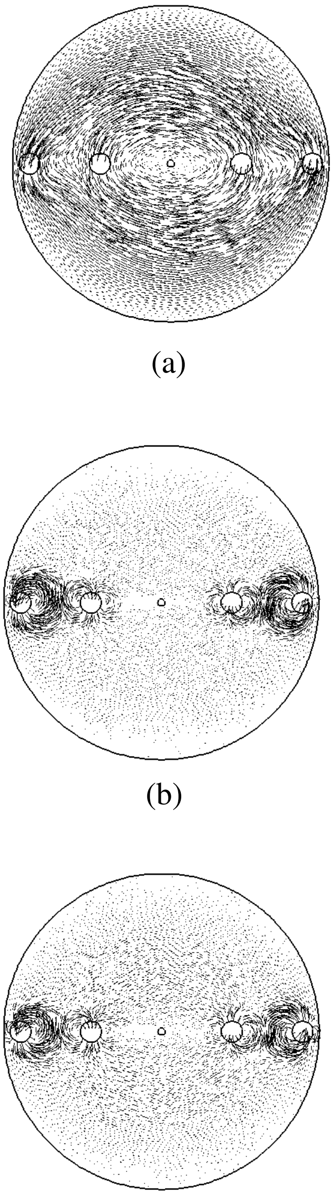


Fig.5 Tangential velocity for different Hedström numbers on the median plan

In Fig. 6 we can see that the increase in yield stress reduces the shear zone near the two blades of the impeller, which validates what we just said that the Hedström number equal to zero is the case of a Newtonian fluid or shear zone is nonexistent.


(a)

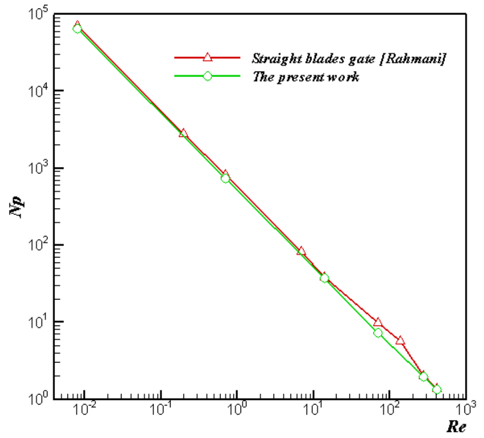
(b)

(c)

Fig.6 Velocity field for different Hedström numbers: (a) He=0; (b) He=8294; (c) He=248832

D. Power consumption

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



$$Np = \frac{P}{\rho . N^3 . d^5} \tag{8}$$

Fig.7 Variation of power number as function of the Reynolds number

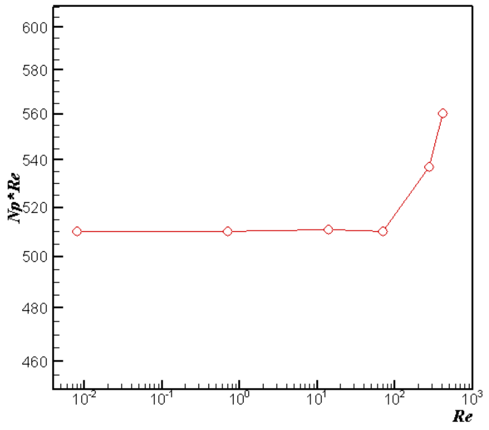


Fig.8 Variation of the product Np.Re as function of the Reynolds number

Fig 7 represents the variation of the Np number as function of the Reynolds number in logarithmic coordinates which the evolution seems linear what means that the Np.Re product is constant and depends only on the geometry of system. However, the Fig 8 shows that the product Np.Re is not precisely constant but varies slightly with the Reynolds number. Precisely, it is interesting to note that the Np.Re product vary slightly with low Reynolds number (from Re = 0.01 to 50), but when Re increase the variation becomes considerable.

Finally we have completed this contribution with a comparative study between a circular and straight blades gate. It was noted that the field of flow remains the same one except a significant consumption of the straight blades gate compared to a circular.

IV. CONCLUSION

Our work is a contribution to the study of the difficulties encountered on viscoplastic fluids which behavior is progressed through time: practically, Newtonian at the beginning of the reaction, then becomes shear thinning and finally viscoplastic as long as the concentration increase.

Characterization of laminar flow and power consumption in agitated vessel with circular gate impeller has been investigated in order to provide a physical analysis of mixing.

ACKNOWLEDGMENT

Many thanks are due to ENERGARID laboratory, especially the group of “rheology and agitation”

REFERENCES

[1] E. Galindo, A. W. Nienow, “The performance of the Scaba 6SRGT agitator in the mixing of simulated xanthan gum broths,” Chemical Engenering an Technology, vol 16, pp. 102-108, 1993.

[2] H.Ameur, M.Bouzit, “Agitation of yield stress fluids by two-blade impellers Canadian Journal of Chemical Engeneering and Technology, vol 3, N° 4 , 2012.

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

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

[5] 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.

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

[7] L. Rahmani, B. Draoui, B. Mebarki, M. Bouanini and O. Hami. “Heat transfer to Bingham fluid during laminar flow in agitated tank,”. International Review of Mechanical Engineering (I.RE.M.E.), Vol. 3, N 2, 2009.

[8] B. Mebarki, B. Draoui, L. Rahmani, M. Bouanini, M. Rebhi, E. Benachour. “Numerical Study of Mechanical Stirring in Case of Yield Stress Fluid with Circular Anchor Impeller.” Sensors & Transducers Journal, vol 132, Issue 9, pp.108-121. 2011.

[9] Patankar S. V, “Numerical heat transfert and fluid low.” (Mc-Graw Hill. 1980.

[10] L. Rahmani1, B. Mebarki1, B. Allaoua, B. Draoui. “Laminar flow characterization in a stirred tank with a gate impeller in case of non-Newtonian fluid,”. Energy procedia 36,pp 418-427, 2013.