Numerical Analysis of thermal behavior in agitated vessel with Non-Newtonian fluid

A Benmoussa*, L Rahmani
ENERGARID Laboratory, University of Tahri Mohamed, Bechar, Algeria

ABSTRACT
For some ten years, several analytical problems have not been solved by the Navier-Stokes equation; hence numerical methods also known by 'Computational Fluid Dynamic' (CFD) have been developed. In the present paper, we have described in depth numerical study of the basic fluid mechanics problem and heat transfer of yield stress fluid flow with regularization model of Bercovier and Engelman [1] in a cylindrical vessel not chicaned equipped with an anchor stirrer by using computational fluid dynamics (CFD) based on the finite volumes method discretization of Navier - Stokes equations formulated in variables (U.V.P). We have studied the effect of inertia and the plasticity influence; we have analyzed also the influence of rheological parameters on the hydrodynamic flow behavior, such as the velocity components and the power consumption.

1. INTRODUCTION

Agitation techniques are widely used in process engineering and their fields of application are very varied. Among these agitation techniques, mechanical agitation is generally used, where it is necessary to create or accelerate the transfers of energy and matter between one or more phases in the presence, to achieve and promote certain physico-chemical transformations of the material.

Correct mixing is required to achieve complete homogeneity of process substances in a vessel, and to ensure that all reactive components can interact and create the desired product. Thermal transfer is also directly affected by the degree of homogeneity in a system; the right system can maintain a uniform temperature throughout the tank volume and can respond quickly to change.

Mechanical stirring plays crucial role in the success of many process engineering operations; where the quality of the final product is a function of the effectiveness of the mixing process.

These operations generally face difficulties related to the implementation of non-Newtonian fluids such as chemical and polymer solution, detergents, petroleum products…, when moved, stirred or mixed. If the agitation is a unit operation aiming at promoting a physical process, such as homogenization or enhancement of heat transfer; predicting the power required for its implementation will be primary concern.

Yield stress 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

*Corresponding Author: benmoussa.a@hotmail.fr

to dead zones in the flow witch lower mixing efficiency [2]. 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 [3].

Understanding mechanisms of agitation still remains difficult especially in the case of non-Newtonian fluids. In the area of these fluids, there is a wide variety of behavior possible.

For this class of fluids there is several experimental and numerical works namely Youcefi [4] Belhadri [5], Poullain et al. [6], Roustan and Bouaifi [7], and Niedzielski Kuncewicz [8] Rajeev et al. [9] Rudolph et al. [10] Marouche et al. [11], Pedrosa and Nunhez [12], Amadei et al. [13], Burgos et al. [14], Armenante et al. [15], Anne-Archard et al. [16] Frederick et al. [17], Rahmani et al. [18] Benmoussa et al [19-20].

There is a wide range of mixing geometries available for viscous non-Newtonian fluids, and the selection of an appropriate design for a given application is not an easy task. Several criteria may be used depending on the process requirements, such as specific power consumption, mixing time, pumping efficiency, shear rate distribution and flow field characteristics. The absence of dead zones is of foremost importance for good homogenization.

2. NUMERICAL MODEL

2.1. Stirred 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 center of the tank rotating around a shaft of diameter da, 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.

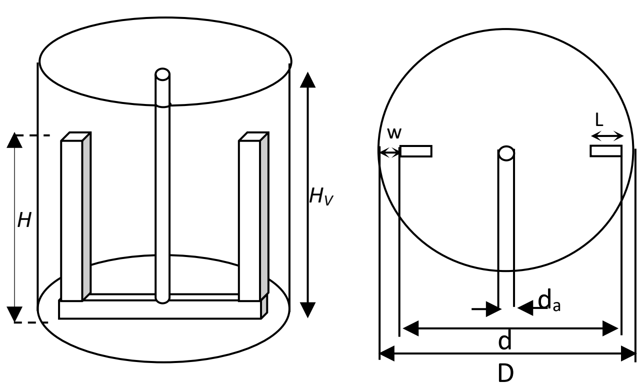


Figure 1: Mixing system

2.2. Fluid model

To study the yield stress fluid which is the subject of this paper, we use viscoplastic fluid model of Bercovier- Engelman [20] consists of adding a small regularization parameter δ :

$$\bar{\bar{D}} = 0 \text{ for } \|\tau\| < \tau_0 \quad (1)$$

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

With:

$$\overline{\overline{D}} = \frac{1}{2}\left(\nabla V + \nabla V^T \right) \tag{3}$$

$$\left| \overline{\overline{\tau}} \right| = \sqrt{\frac{1}{2} \overline{\overline{\tau}} : \overline{\overline{\tau}}^T} \tag{4}$$

2.3. Governing equations

The flow of non-Newtonian fluids is governed by:

Continuity

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

Momentum

$$\rho \frac{\partial \vec{V}}{\partial t} + \left(\vec{V}.\nabla \right) \vec{V} - \eta(\dot{\gamma}) \Delta V - 2 \dot{D} \cdot \nabla \eta(\dot{\gamma}) + \nabla P = 0 \tag{6}$$

Energy equation

$$\frac{\partial T}{\partial t} + \vec{V}\, \overrightarrow{grad}\, T = a\, \Delta T \tag{7}$$

where $$a = \frac{\lambda}{\rho C_p}$$

2.4. Numerical methodology

In this work the numerical simulation is conducted using the commercial Computational Fluid Dynamics code in which a finite volume method developed is implemented.

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. The unstructured meshes of 40610 elements was used. The boundary conditions for velocity are fixed on the impeller and the vessel.

* On the impeller $v_r = v_t = 0$

* On the vessel $v_r = v_t = -.\pi ND$

In the fluid the dimensionless temperature has zero initial value, and we imposed a dimensionless temperature equal 1 on surfaces of vessel.

* On the impeller $T = 0$

* On the vessel $T = T_P$

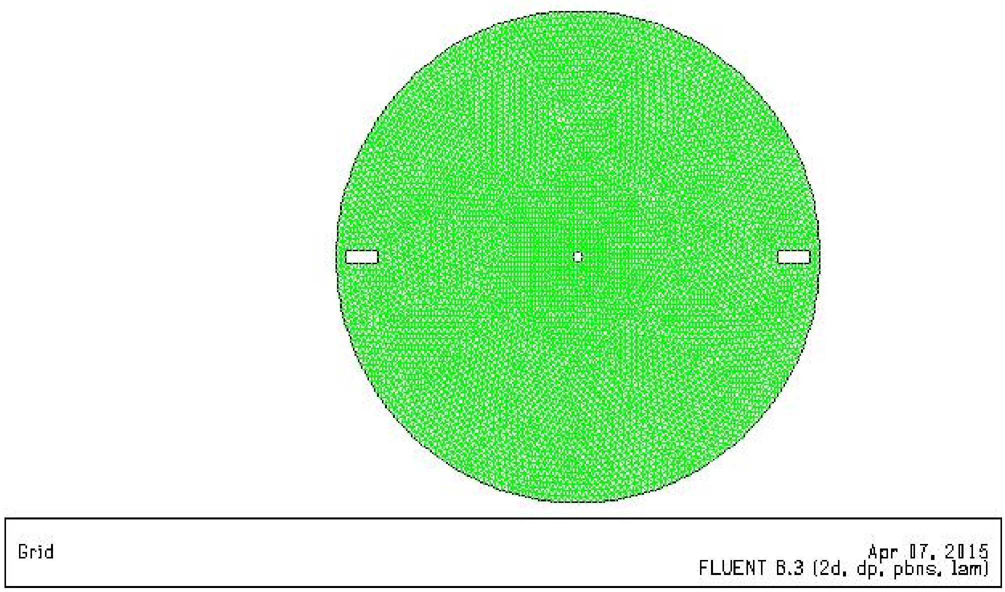


Figure 2: Grid mesh

3. RESULTS AND INTERPRETATIONS

First before the presentation of our results, we have compared our results with a numerical work of Rahmani [20], the results show a very good agreement.

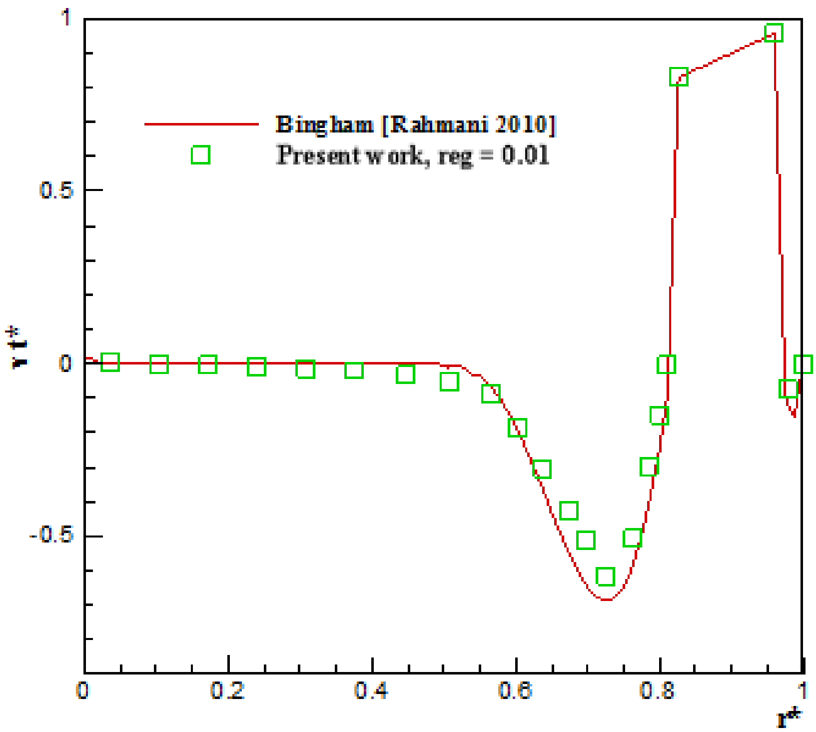


Figure 3: Tangential velocity on the impeller plan for Re = 13.8

3.1. Effect of Regularization Parameter

To get a better approximation of the ideal model, we tested the influence of regularization parameter, figures 4 and 5 show the tangential velocity on the impeller and the median plans respectively.

For different regularization parameter compared with the Bingham model, we note that the value δ= 0.01 gives a sufficient approximation of the model.

3.2. Effect of inertia

For this purpose, the mobile rotational speed is studied in figure 6.

With increase in Reynolds number, the recirculation zone which is inside the vessel moves with an inclination towards the attacks edges of the agitator which shows the effect of the forces of inertia which are dominant compared to viscosity forces. In the vicinity of the blades there is a deviation of the current lines, this deviation grows more and more as the Reynolds number increases, which means that in this zone the fluid is submitted to a large shear. From the external end of the blade to the wall of the tank the current lines are substantially parallel to the wall which means that the flow in this area is tangential.

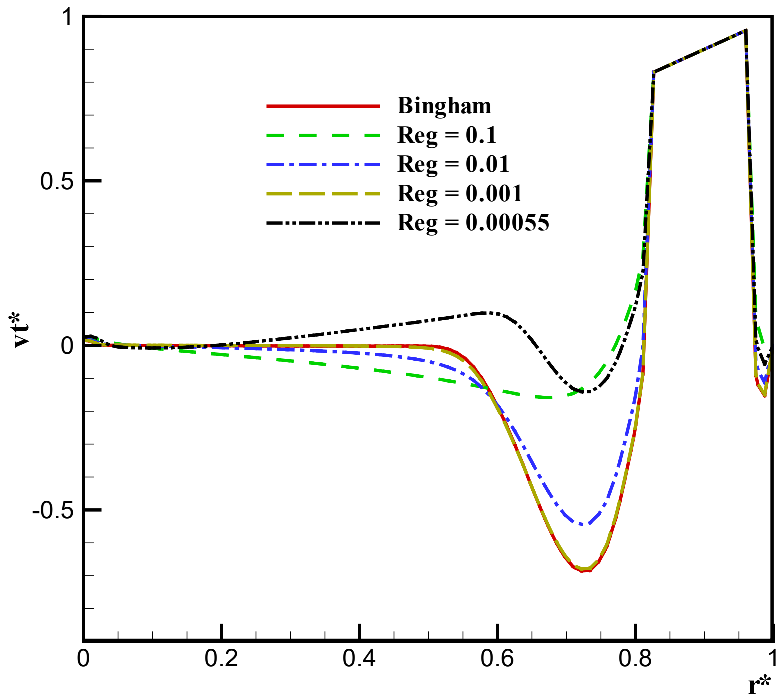


Figure 4: Tangential Velocity for Different regularization parameter on impeller plan for Re=13.8

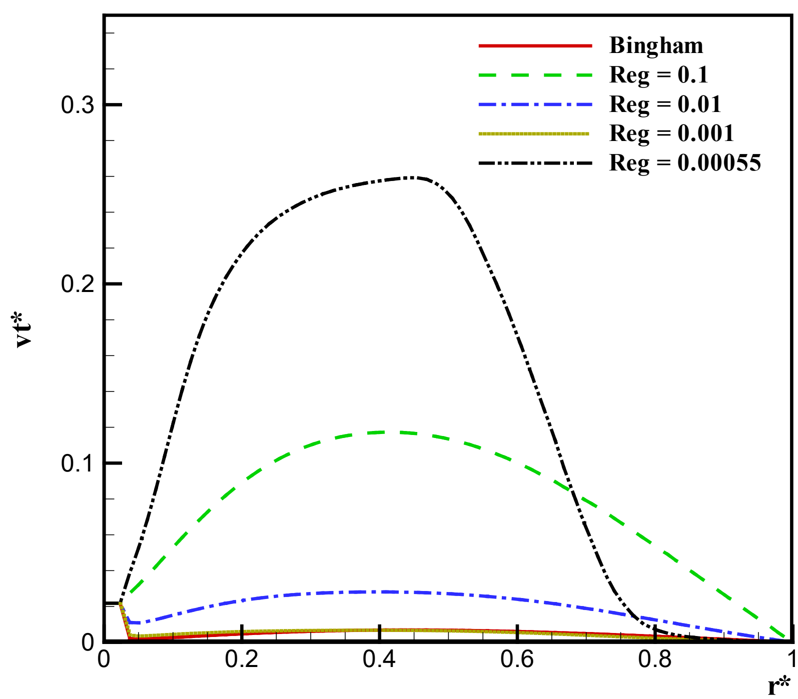


Figure 5: Tangential Velocity for Different regularization Parameters on the median plan, Re=13.8.

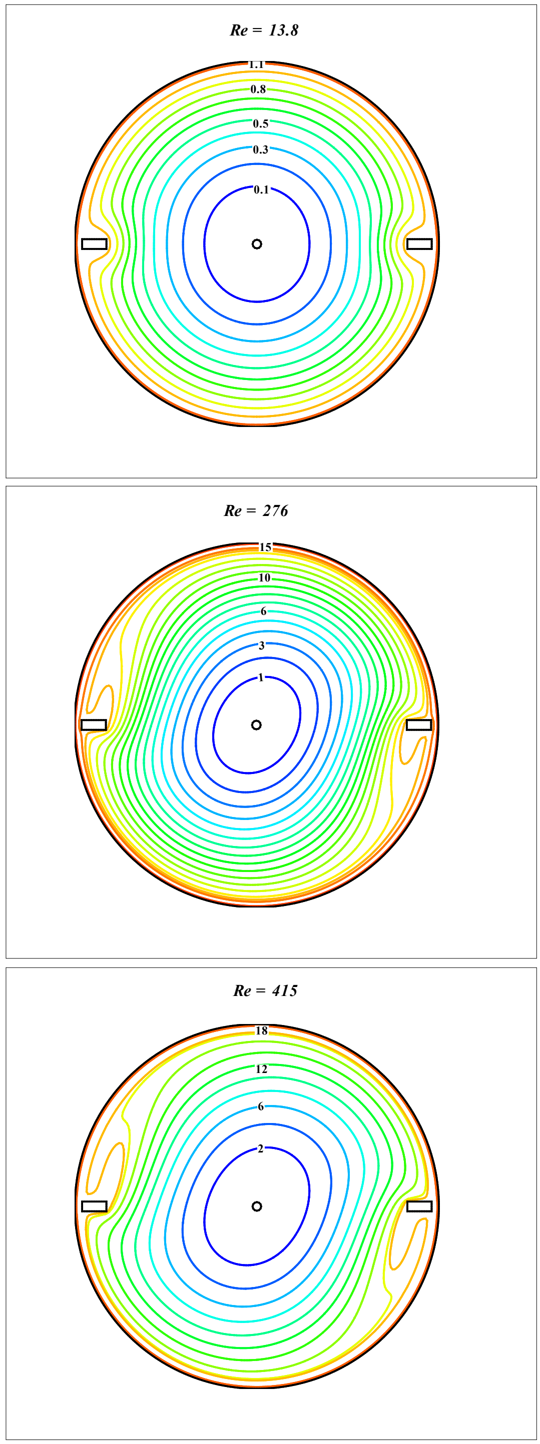
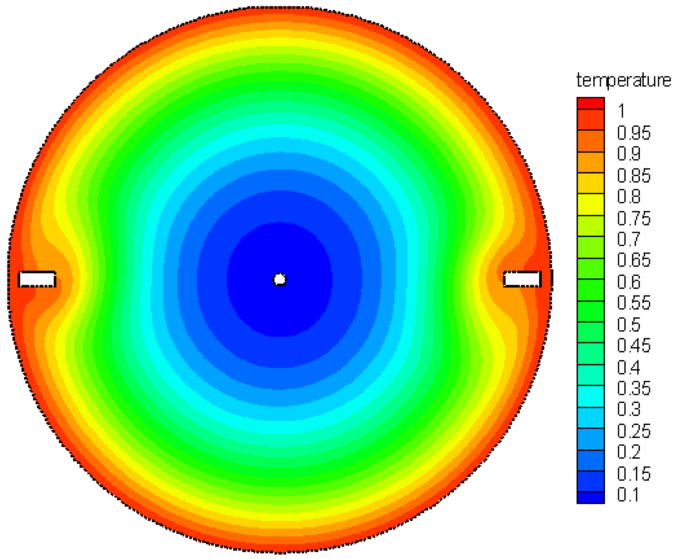
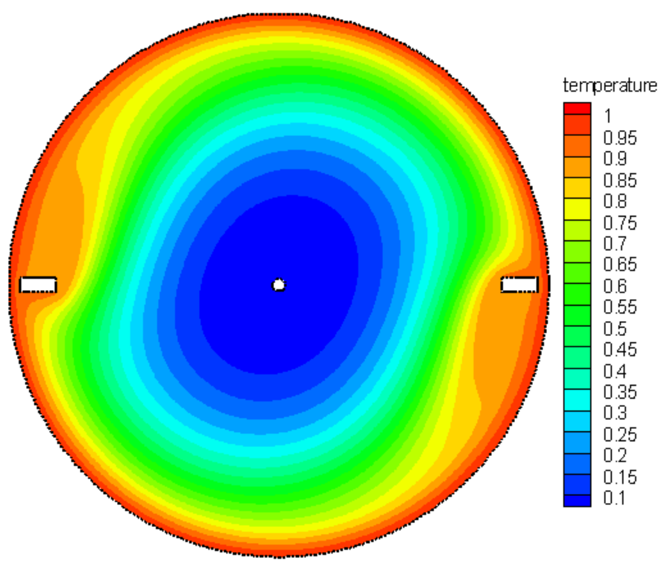


Figure 6: Stream lines for different Reynolds number.

3.3. Thermal field



(a)



(b)

Figure 7: Thermal field for different Reynolds number (a) Re= 13.8, (b) Re= 276 for δ= 0.01

It is clear that these movements in the radial direction contribute to support heat exchange in the vessel; the trajectory of the heated fluid tends to be extended and deflected only by the action of tangential velocities. We also find that in the immediate vicinity of the heating surface, the fluid adhering to the wall of the vessel shows a significant temperature gradient and the lowest temperatures reside within the recirculation zones.

3.4. Nusselt number

This figure shows the evolution of the Nusselt number as function of dimensional time by varying the Reynolds number for Pr = 7.

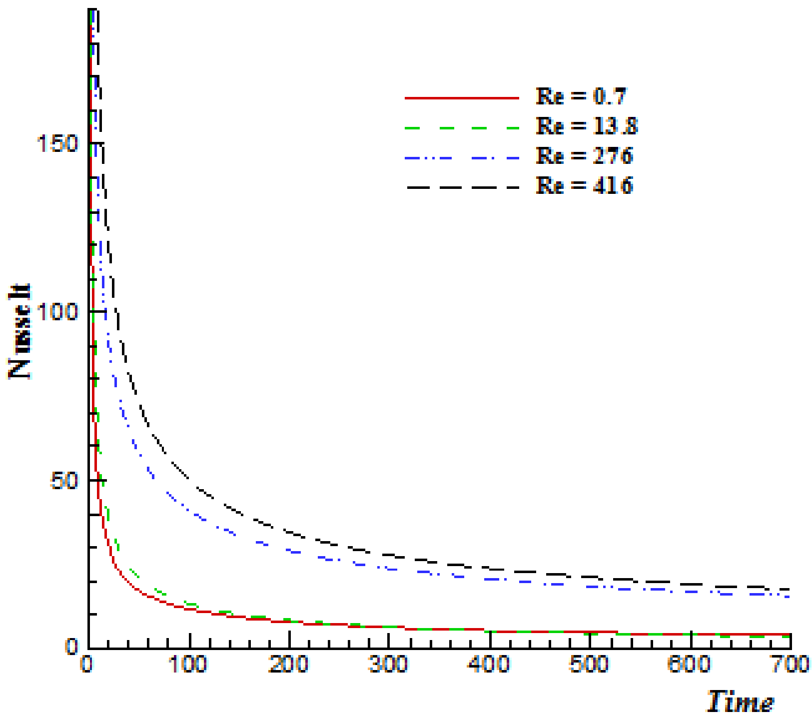


Figure 8: Effect of Reynolds number on the evolution of Nusselt number.

The examination of these curves shows:

- A first unsteady phase, which corresponds to the formation of the thermal boundary layer; This stage is characterized by a relatively large decrease in the number of Nusselt. An increase in the Nusselt has also been observed. As the Reynolds increases, we can explain it by: the inertial effect drives the Bercovier-Engelman fluid to take a Newtonian behavior and therefore the temperature distribution becomes quite fast.
- A second phase, in which the number of Nusselt tends to a constant, which indicates the establishment regular thermal regime; from that moment, the distribution of temperature becomes similar in time.

3.5. Power consumption

The power number is calculated according to this equation:

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

The figures (9 and 10) represent the variation of the power number and the product Np*Re as a function of BI number.

We note that the power number varies linearly as a function of Bingham number (Bi) in logarithmic scale. We no longer observe dispersion but a steady evolution of Np*Re. The BI number thus integrates well on this global quantity that is the number of power, the effects of inertia and the effects of threshold.

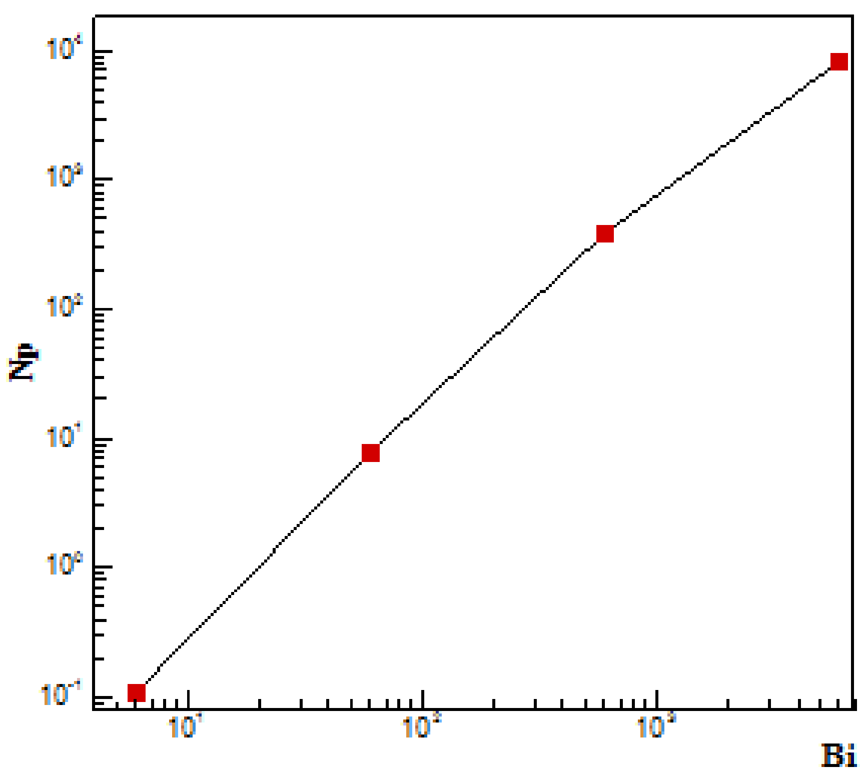


Figure 9: Variation of Np as function of BI number 6, 60, 600 and 6000.

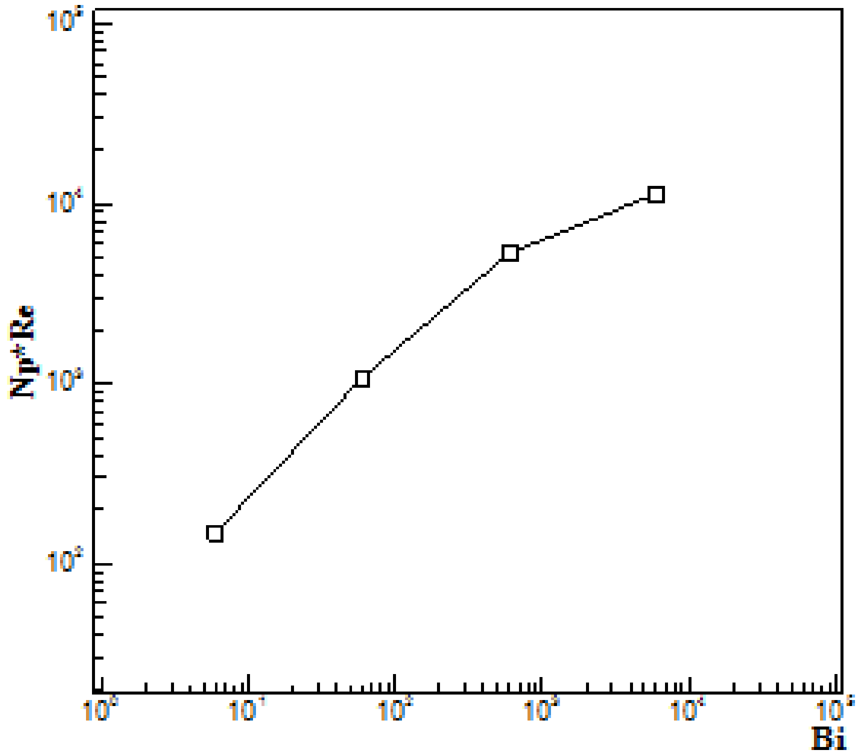


Figure 10: Variation of the product Np*Re as function of BI number 6, 60, 600 and 6000.

4. CONCLUSION

The study of hydrodynamic and thermal behaviors generated by anchor stirrer in a stirred tank is approached by numerical simulation way, this allows to visualize the influence of various parameters on the flow, and we can use the Bi number to distinguish the flow regime in the case of viscoplastic fluids. We also found that the thermal performance is intimately related to the hydrodynamics state of the entire stirred tank. Two different processes combine to ensure homogenization.

REFERENCES

[1] M. Bercovier and M. Engelman, (1980) A finite element method for incompressible non-Newtonian flows, Journal of Computational Physics, vol. 36, no. 3, pp. 313-361.

[2] E. Galindo and A. W. Nienow, The performance of the Scaba 6SRGT agitator in the mixing of simulated xanthan gum broths, Chemical Engineering and Technology 16 (1993), 102-108.

[3] H.Ameur, M.Bouzit, “Agitation of yield stress fluids by two-blade impellers Canadian Journal of Chemical Engineering and Technology, vol. 3, no. 4, 2012.

[4] Youcefi, S.; Bouzit, M.; Ameur, H.; Kamla, Y and Youcefi, A (2013) ‘Effect of some design parameters on the flow fields and power consumption in a vessel stirred by a Rushton turbine’, Chemical and Process Engineering, Vol. 34, No. 2, pp 293-307.

[5] Belhadri. M, “Ecoulements des fluides à seuil au travers de singularités convergentes et divergentes’’. Thèse de Doctorat, Université de Grenoble (1996).

[6] Poullain, P., B. Cazacliu, L. Doubliez and P. Coussot, 2001. Détermination du champ de contraintes dans “un malaxeur à partir de mesures de vélocimétrie (PIV), in 36ème Colloque du G.F.R., pp: 186-191.

[7] Bouaifi.M and Roustan. M, Power consumption, mixing time and homogenisation energy in dual-impeller agitated gas-liquid reactors, Chemical Engineering and Processing, 40 (2001), 2, pp. 87-95

[8] Niedzielska, A. Kuncewicz C., Heat transfer and power consumption for ribbon impellers, Chemical Engineering Science, 60 (2005), 8, pp. 2439-2448.

[9] Rajeev. K, Thakur, Ch. Viall, G. Gjelveh, M. Labbafi, Mixing of complex fluids with flat-bladed impllers; effect of impller geometry and highly shear-thinning behavior, Chemical Engineering and Processing, 43 (2004), 10, pp.1211-1222.

[10] Rudolph, L.; Schafer, M.; Atiemo-Obeng, V and Kraume, M (2007) ‘Experimental and numerical analysis of power consumption for mixing of high viscosity fluids with a co-axial mixer’, Chemical Engineering Research and Design, Vol. 85, No. 5, pp. 568- 575.

[11] Marouche, M., Anne-Archard, D., and Boisson, H.C., [2002]. A numerical model of yield stress fluid in a mixing vessel. Applied Rheology 12, 182–191.

[12] S.M.C.P. Pedrosa, J.R. Nunhez. The behavior of stirred vessels with anchor type impellers, Computers and Chemical Engineering 24 (2000) 1745-1751.

[13] Amadei. B and Savage. W, An analytical solution for transient flow of Bingham viscoplastic materials in rock fractures, International Journal for Rock Mechanics and Mining Sciences, 38 (2001), pp. 285-290

[14] Burgos. G.R, Alexandrou. A.N, and Entov. V, On the determination of yield surfaces in Herschel-Bulkley finds, Journal of Rheology, 43 (1999), 3, pp. 463-483

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

[16] Anne-Achard, D.; Marouche, M and Boisson, H.C (2006) ‘Hydrodynamics and Metzner–Otto correlation in stirred vessels for yield stress fluids’, Chemical Engineering Journal, Vol. 125, pp. 15-24.

[17] Frédéric. S.; Pascal, J and Albert, M (2007) ‘Viscoplastic fluid mixing in a rotating tank Yield stress’, Chemical Engineering Journal, Vol. 62, pp. 2290-2301.

[18] Rahmani. L, Draoui. B, Mebarki.B, Bouanini. M, Hami. O’ 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.

[19] A Benmoussa, L Rahmani and B Draoui, Simulation of Viscoplastic Flows in a rotating Vessel Using a Regularized Model Int. Jnl. of Multiphysics Volume 11, Number 4, pp 349-358, 2017.

[20] Amine Benmoussa, Lakhdar Rahmani and Mebrouk Rebhi, regularization model effect on yield Stress fluids behaviour in a rotating vessel, Advances and Applications in Fluid Mechanics. Volume 19, Number 3, 2016, Pages 507-515.