Effect of Regularization Parameter on Flow Field and Power Consumption in Agitated Vessel by Two Blade Impeller

Amine Benmoussa, Lakhdar Rahmani and Mebrouk Rebhi

University of Tahri Mohamed Bechar, Energarid Laboratory BP417 Route De Kenadza, Bechar, Algeria

Abstract: The existence of residual stress, or yield stress, in fluid mechanics has been first modeled by Bingham [1], who foresaw a value for the limit tension beyond which the material would present a viscous flow using a linear model. Such physical system not only raises the viscosity but also imposes a resistance that shall be exceeded for the material to flow normally, requiring the destruction of the material structure so that it flows in a viscous manner. For this model there is lack of numerical methods able to deal with the inequality restriction. Thus, regularization methods have been proposed where, in a way, they eliminate the restriction and, on the other hand, they introduce constitutive nonlinearities in the relation between the stress tension and the shear rate. In this work, the characterization of hydrodynamic fields of incompressible yield stress fluid with regularization model of Bercovier and Engelman in a cylindrical vessel not chicaned equipped with two blade stirrer was undertaken using a code Fluent CFD 6.2.13 based on discretization to finite volume method the Navier Stokes equations formulated variables to study the influence of inertia by varying the Reynolds number and the influence of plasticity by the number variation Bingham fluid flow and show that the existence a flow threshold characterized by the number of Hedström can lead to a quasi-immobilization zones within the system agitation for the simulated geometry.

Key words: CFD modeling · Stirred vessel · Laminar flow · Yield stress · Power consumption

INTRODUCTION

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 to dead zones in the flow which lower mixing efficiency (E. Galindo and A. W. Nienow., 1993) [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 (H.Ameur and M.Bouzit., 2012) [2].

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 works Poulain et al. [3], Roustan and Bouaifi [4] and Niedzielski Kunczewicz [5] Rajeev et al. [6]. Marouche et al. [7] and numerical studies namely (Pedrosa and Nunhez 2000) [8], Amadei et al. [9], Burgos et al. [10] and G. Ascanio et al [11], Pham and Mitsoulis [12], Yan and James [13], Anne-Archard et al. [14] Frederick et al. [15], M. Bouanini et al [16] H.Ameur et al [17], B.Mebarki et al. [18], L. Rahmani et al. [19-23].

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.

For a long time, 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 [2]. Developing new processes and optimizing existing processes [24].

Computational fluid dynamics (CFD) is playing a key role in helping to understand the flow inside stirred tanks. It is becoming a useful tool in the analysis of the highly complex flow inside stirred vessels. The design of these vessels to date assumes uniform temperature and perfect mixing, which are strong assumptions that clearly are not the case. This aspect can be critical especially for highly exothermic reactions and for non-Newtonian fluids, typical of polymer reactions. For a mixing polymerization reactor, selection of the impeller type largely determines the physical properties of the polymer being formed [8].

Our objective is to employ advanced computational fluid dynamics (CFD) to study field flow characteristics and power consumption for stirring yield stress fluids with Bercovier and Engelman regularization model; we have also studied the effect of inertia and the plasticity, also the influence of rheological parameters on the hydrodynamic flow behavior.

### Numerical Model

**Description of the Model:** The system consists of a cylindrical flat bottomed vessel of diameter D equipped with a gate impeller of diameter d positioned at the centre of the tank rotating around a shaft.

![Mixing system](image)

**Table 1: Dimension of mixing system**

<table>
<thead>
<tr>
<th>D</th>
<th>d</th>
<th>da</th>
<th>e</th>
<th>w</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.96</td>
<td>0.023</td>
<td>0.027</td>
<td>0.067</td>
</tr>
</tbody>
</table>

To study the yield stress fluids which constitute the purpose of this paper, the traditional Bingham law is given by:

\[
\vec{D} = 0 \quad \text{for} \quad \|\vec{\gamma}\| < \tau_0 \tag{1}
\]

\[
\vec{\tau} = \begin{pmatrix} \tau_0 \\frac{\tau}{\gamma} \end{pmatrix} \vec{D} \quad \text{for} \quad \|\vec{\gamma}\| > \tau_0 \tag{2}
\]

where: \( \vec{D} \) and \( \vec{\tau} \) are, respectively, the rate of strain tensor and the stress tensor.

\[
\vec{D} = 1/2 \left( \nabla \vec{v} + \nabla \vec{v}^T \right) \tag{3}
\]

\[
\|\vec{\gamma}\| = \sqrt{\frac{1}{2} \cdot \vec{\gamma} \cdot \vec{\gamma}} \tag{4}
\]

According to equations (1) and (2), the flow domain for a Bingham fluid is characterized by two distinct regions. In the regions where \( \|\vec{\gamma}\| < \tau_0 \) the material behaves like a rigid solid and in the regions where \( \|\vec{\gamma}\| > \tau_0 \) the material flow with an apparent viscosity \( \eta_{ap} \) [19].

\[
\eta_{ap} = \eta_{\infty} + \frac{\tau_0}{\gamma} \tag{5}
\]

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 \( \|\vec{\gamma}\| \) approaches \( \tau_0 \).

A regularized version of the Bingham model has been proposed by Bercovier and Engelman [23], it consists of adding a small regularization parameter \( \delta \) in the denominator of equation (2) which becomes:

\[
\tau = 2 \left( \frac{\tau_0}{\gamma} + \frac{1}{\eta_{\infty}} \right) \frac{\vec{D}}{\gamma + \delta} \tag{6}
\]

**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 [18]:

---

1026
Continuity
\[
\frac{\partial \mathbf{V}}{\partial t} + \nabla (\rho \mathbf{V}) = 0
\]  
(7)

Momentum
\[
\rho \left[ \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} \right] = \mathbf{f} - \nabla P + \nabla \tau = 0
\]  
(8)

Introducing the second invariant of the stress tensor, the equation (7) becomes:
\[
\rho \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla) \mathbf{V} - \eta(\gamma) \Delta \mathbf{V} - 2D \cdot \nabla \eta(\gamma) + \nabla P = 0
\]  
(9)

The dimensionless numbers governing the flow are:
\[
Re = \frac{\rho N D^2}{\eta_{\infty}}
\]  
(10)
\[
He = Re.Bi
\]  
(11)
\[
Bi = \frac{\tau_0}{\eta_{\infty}N}
\]  
(12)

**Numerical Methodology:** Computational fluid dynamics (CFD) is playing a key role in helping to understand the flow inside stirred tanks. 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 [25] 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.

**Results and Interpretations:** Initially before the presentation of our results, we have compared our results with a numerical work of Rahmani [21]. The results show a very good concordance.

![Fig. 2: Tangential Velocity on the Median Plan for $\delta = 0.01$](image)

**Effect of the Regularization Parameter:** In order to analyze the influence of the regularization parameter, Figure 3 shows 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 $\text{reg} = 0.01$ gives a sufficient approximation of the model. For small values of $\text{reg}$ there was a problem of convergence.

![Fig. 3: Tangential Velocity for Different Regularization Parameters on the impeller plan, $Re = 13.8$](image)
Fig. 4: Tangential Velocity for Different Regularization Parameters on the median plan, Re = 13.8

**Effect of Inertia:** We observed that the fields are identical and the existence of three zones: a first shear zone at the blades, the second is recirculation zone located just near the axis of the stirrer, the latter is an rigid zone represented by the rest of the tank. With increase of Reynolds number we see always the co-existence of the three zones mentioned above but only their shapes and sizes have changed considerably.

**Effect of Yield Stress:** To analyze the influence of yield stress, we study the flow of tree Bingham fluids of yield stress \(\tau_0\) equals 0, 1, 5 and 30 Pa. These values correspond to dimensionless numbers of Hedström 0, 8294, 41472 and 248832.

The value of He equal to zero corresponds to the Newtonian case. The speed of rotation \(N\) was set at 1 rpm. 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. This is all the more clearly the number of Hedström increases. The existence of a yield stress can therefore change radically hydrodynamics practically canceling the velocity on a large part of area.

**Cross Effect Between Inertia and Viscoelasticity:** This figure shows that when \(Bi\) increases, the effect of plasticity become dominating and consequently the flow does not occur.

Fig. 5: Velocity contour for different Reynolds number

**Power Consumption:** The power consumption is a macroscopic result obtained by integration on the impeller surface of the local power transmitted by the impeller to
the fluid. It is quite equivalent to say that the power consumption \( P \) is entirely given by the impeller to the fluid [26]. In these conditions:

\[
P = \int_{\text{vessel volume}} \mu \Phi, dV
\]  

The power number is calculated according to this equation:

\[
N_p = \frac{P}{\rho N^3 d^5}
\]  

Fig. 9 represents the variation of the \( N_p \) number as function of the Reynolds number in logarithmic coordinates which the evolution seems linear.

Fig. 6: Tangential Velocity for Different Hedstrom Numbers on the Impeller Plan

Fig. 7: Tangential Velocity for Different Hedstrom Numbers on the Median Plan

Fig. 8: Tangential Velocity for Different Binham number on impeller plan, \( Re=13.8 \)

Fig. 9: Tangential Velocity for Different Binham number on median plan, \( Re=13.8 \)

Fig. 10 shows that the product \( N_p Re \) is not precisely constant but varies slightly with the Reynolds number. Precisely, it is interesting to note that the \( N_p Re \) product varies slightly with low Reynolds, but when \( Re \) increase the variation becomes considerable.

In Figure 10, We can see that the increase of the regularization parameter increases the power consumption, which is explained by: the increase of regularization parameter can modify the behavior of the fluid that increases the plasticity which makes extend the rigid zone in the vessel or a significant power consumption.
CONCLUSION

After an efficient study of the influence of various parameters on the flow, it show the important significations by the viscoplastic behavior and in particular, the quasi-immobilization of the fluids in a broad area when the effects of yield stress characterized by the number of Hedström are dominant; and we can use Bingham number in order to distinguish the flow regime in the case of viscoplastic fluids. We also noted that for the low regularization parameters (0.00055 and less) a difficulty of convergence which gives a bad approximation of the Bingham law that is obvious also for the other sizes characterizing the flow.

ACKNOWLEDGEMENTS

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

Nomenclature:

- \( d \) : impeller diameter, m
- \( \dot{\gamma} \) : rate of strain tensor, dimensionless
- \( D \) : vessel diameter, m
- \( H \) : impeller height, m
- \( H_v \) : vessel height, m
- \( d_s \) : shaft diameter, m
- \( e \) : impeller thickness, m
- \( N \) : rotational speed, rd/s
- \( N_p \) : power number, dimensionless
- \( P \) : power consumption, W
- \( V_r \) : radial velocity, m/s
- \( V_t \) : tangential velocity, m/s
- \( W \) : agitator to wall clearance, m
- \( \delta, \text{reg} \) : regularization parameter of Bercovier and Engelman
- \( \text{Bi} \) : Bingham number, dimensionless
- \( \text{He} \) : Hedström number, dimensionless
- \( \text{Re} \) : Reynolds number, dimensionless

Greek Symbols:

- \( \eta \) : apparent viscosity, Pa\,s
- \( \eta_c \) : cut off viscosity for low shear rate, Pa\,s
- \( \eta_l \) : limiting viscosity at infinite shear rate, Pa\,s
- \( \rho \) : density, kg/m\(^3\)
- \( \tau \) : stress tensor, N\,m\(^2\)
- \( \tau_y \) : yield stress, N\,m\(^2\)
- \( \gamma_c \) : shear rate, s\(^{-1}\)
- \( \bar{\tau} \) : cut off shear rate, Bingham model
REFERENCES


