Draft Tube Length Influence on the Flow
Inside Reactors with Intubated Impeller

LUCIAN MANDREA1, CORINA ALICE BABUTANU1,2, GHEORGHE BARAN1,2, GABRIELA OPRINA2, FLORENTINA BUNEA2
1University “Politehnica” of Bucharest, 313 Splaiul Independenþei, 060042, Bucharest, Romania
2INCDIE ICPE-CA, 313 Splaiul Unirii, 030138, Bucharest, Romania

The efficiency of a suspension homogenization/mixing reactor is evaluated by a uniform concentration of the suspension and low specific energy consumption (kW/m³). For a reactor with given geometry and volume, free or intubated impellers are used for the mixing of the suspension. In this work is studied the influence of the draft tube length. The streamlines corresponding to the flow of water are obtained through the numerical integration of Navier-Stokes equations. The increasing of the draft tube length modifies the flow, leading to the reduction of the number and intensity of the vortices inside the reactor (outside the draft tube). These vortices can lead to suspensions agglomeration, facilitating the sedimentation and thus increasing the energy consumption of the homogenization process.

Keywords: mixing reactor, homogenization process, draft tube, fluid flow

Mixing processes are used in numerous industrial branches (chemical, pharmaceutical, food and metallurgical process industries) as well as in municipal wastewater treatment, in dispersion applications such as liquid-solid, liquid-gas, liquid-solid-gas etc [1].

The homogenization of the substances leading to a uniform value of the concentration, the heat transfer and the shortening of the reaction time in certain industrial processes are achieved through mixing. This process also avoids suspension sedimentation (which leads to a decrease in the effective volume of the vessel), scum formation and particles agglomeration. Different studies were elaborated referring to mixing processes [2-4], industrial reactors [4, 5] and process installations [6].

In this paper the authors aim to determine the stream lines shape of a Newtonian fluid (water) in a mixing reactor and to present the stream lines aspect in two different flow cases.

The mixing reactors efficiency is given by the homogenization degree and the specific energy consumption (e.g. kW/m³). These parameters depend on the reactor geometry and the mechanical equipment chosen for mixing.

When an intubated impeller for the homogenization process is selected, the performances of the reactor depend on the ratio \( D/d \) and on the \( \Delta z, h \) and \( z_0 \) values (fig. 1). The \( z_0 \) value is modified when the value of interior tube length \( L \) is changed. The average flow rate produced by the propeller is constant and leads to a constant value of the Reynolds number \( Re \).

The computational domain considered for the numerical integration is represented by the reactor from fig.1 which has the possibility to operate with interior recirculation tubes of different lengths.

The variable parameter for the two studied cases is the draft tube length \( L \) which determines the value of \( z_0 \) on the vertical, according with the figure 1.

The other geometrical parameters of the reactor, the physical properties of the fluid and the flow are constant. The cases of the numerical modeling correspond to a steady flow of the fluid.

* email: mandrea_lucian@hotmail.com; Tel.: 0214029486

The movement is considered laminar, both in the lower part of the draft tube as well as outside it (the body of the reactor). Thus, the rotation speed of the propeller is selected so that to provide a laminar motion.

The influence of the reactor geometry and draft tube length on the flow

A suspension mixing reactor with a given volume is considered. The working fluid is a liquid with known physical properties (density, viscosity, temperature etc.), in our case water, in which there are solid suspensions with known granulometry curve and terminal velocity. The angles \( \alpha \) and \( \theta \) are adequately chosen to assure the flow acceleration in the free surface vicinity (\( \theta \neq 0 \)) and in the tapered part of the reactor (\( \alpha \neq 0 \)) [7]. Thus, liquid phase velocities bigger than the terminal velocity of the suspensions are obtained [7]. The size of \( \Delta z \) must avoid...
cone formation at the aspiration of the impeller; the cone determines a diminishing of the flow rate entrained by the impeller at a given speed. The ratio $D/d$ determines the energy consumption and the flow regime in the cylindrical part of the reactor, where the Reynolds number is defined as:

$$Re = \frac{VD}{\nu} = \frac{Q}{\frac{\pi}{4}(D^2 - d^2)} \frac{1}{\nu} \frac{\pi}{4}(D + d) = \frac{4Q}{\nu \pi (D + d)}$$  \hspace{1cm} (1)$$

where $V$ is the average velocity, $D$ the equivalent diameter, $\nu$ the cinematic viscosity and $Q$ the volumetric fluid flow rate. There is not a unanimously accepted critical value of Re number in literature [1, 8, 9]. Thus, in this study a laminar flow regime is assumed.

Due to the action of the hydrodynamic forces on the particles, through Magnus effect, a movement of particles with different size and weight along the reactor is produced [10]. The intensity of Magnus effect depends on the ratio $D/d$, but not on the Reynolds number of the flow.

The influence of $z_0$ upon the flow is theoretically evaluated through Navier-Stokes equations integration. Due to the lack of Recr value for the flow between two coaxial cylinders, the Reynolds number inside the draft tube (Re$_r$) is chosen as reference value:

$$Re_r = \frac{Vd}{\nu} = \frac{d}{\nu} \frac{4Q}{\pi d^2} = \frac{4Q}{\pi \nu d}$$  \hspace{1cm} (2)$$

The relation between the two Reynolds numbers given by the expressions (1) and (2) is

$$Re = Re_r \frac{d}{D + d}$$  \hspace{1cm} (3)$$

Thus, if the flow is laminar inside the tube, it is also laminar in the cylindrical part of the reactor. This hypothesis also assures the boundary condition for velocity at the outlet of the draft tube, given by the formula corresponding to Hagen-Poiseuille movement.

**The influence of $z_0$ dimension on streamlines**

Assuming a steady laminar axial symmetric flow, the Navier-Stokes equations written in cylindrical coordinates are used. Transforming the equations into non-dimensional relations, eliminating the pressure and introducing the stream function, the partial differential equation is obtained:

$$\psi^{\prime\prime\prime} + \frac{2}{r} \psi^{\prime\prime} + \frac{\psi^\prime}{r} - 2 \frac{\psi^{\prime\prime\prime} + \psi^{\prime\prime}}{r} + \frac{3}{r^2} \left( \frac{\psi - \psi_w}{r} \right) = Re \left( \frac{3}{r^2} \psi^{\prime\prime} + \frac{\psi^{\prime\prime\prime}}{r^2} - 2 \psi^{\prime\prime} \psi^{\prime\prime}_v - 3 \psi^{\prime\prime}_v \psi^\prime_v + \psi^{\prime\prime}_v \psi^\prime_v - \psi^{\prime\prime\prime}_v \psi^\prime_v + \psi^{\prime\prime\prime}_v \psi^\prime_v \right)$$  \hspace{1cm} (4)$$

where $\psi$ is the current function defined by the relations:

$$\frac{\partial \psi}{\partial r} = u, \frac{\partial \psi}{\partial \phi} = -v$$

The terms $u$ and $v$ are the horizontal respectively vertical velocity components of the water.

The radius $r$ which corresponds to the flow region varies between $(D-d)/2$ and $D/2$.

Applying the Taylor finite series development method the algebraic associated relation is obtained:

$$\alpha \psi = \beta + \Re \delta$$  \hspace{1cm} (5)$$

where $\alpha$, $\beta$, and $\delta$ are given by:

$$\alpha = 6 \left( \frac{1}{h^2} + \frac{1}{k^2} - \frac{1}{h^2 r^2} \right) + \Re \left[ \frac{3}{h^2} + \frac{2}{k^2} \right] \frac{\psi_2 - \psi_k}{r h}$$

$$\beta = \left[ \frac{4}{h^2} + \frac{1}{k^2} - \frac{3}{r^2 h^2} \right] \frac{\psi_1 + \psi_2}{k^2} - \frac{1}{h^2} \left( \frac{1}{h^2} + \frac{3}{r^2} \right) \frac{\psi_1 - \psi_k}{r h} + \frac{4}{h^2} + \frac{1}{k^2} \psi_2 + \psi_k + \psi_k - \psi_1 - \psi_2 + \psi_3 + \psi_3 - \psi_1 - \psi_2 + \psi_3$$

$$\delta = \frac{\delta_1 + \delta_2}{r}, \text{where}$$

$$\delta_1 = \frac{\psi_2 - \psi_1}{4 h k} + \frac{\psi_2 - \psi_3}{k^2} + \frac{\psi_3 - \psi_2}{h^2} \frac{3}{r h} + \frac{\psi_k - \psi_3}{r h} + \frac{\psi_k - \psi_2}{r^2 h} + \frac{\psi_k - \psi_3}{r k^2} + \frac{\psi_3 - \psi_2 - \psi_{2h} + \psi_k}{4 h k^2}$$

where $h$ and $k$ are the horizontal and vertical steps of the grid.

$z_0$ is reached by making a number of integer steps $k$ on vertical axis.

In this study the flow inside a reactor with $D/d = 300/60$ is analyzed. The results of the numerical integration for two different length of the draft tube at $Re = 12$ are shown in figures 2a and b.

The radius $r$ which corresponds to the flow region varies between $(D-d)/2$ and $D/2$.
The two flow cases were solved using original computing programmes in Pascal language.

Figures 2a and 2b show that for low Reynolds numbers vortices appear inside the reactor. Their number, intensity and position are determined by $z_0$. In the $z_0^1$ case, the flow area without vortices between the two cylinders is bigger than in the $z_0^2$ case. By increasing $\theta$ angle the disappearance of the vortex near the free surface is possible.

Conclusions
For a reactor with a given geometry, equipped with an intubated impeller, at a constant flow rate, the draft tube length determines the streamlines through the number, position and intensity of the vortices.

It is better to diminish the $\theta$ angle from the upper part of the reactor in order to avoid the formation of vortices in this part of the reactor.

Diminishing $z_0$, the suspension homogenization is facilitated (has a positive effect) by the disappearance of the vortices inducing sedimentation from the lower part of the reactor.

The flow pattern obtained in this study suggests further directions for geometry optimization: the reasonable increasing of the ratio $D/d$, which is limited by the energy consumption, and the decreasing of the $\theta$ angle in order to accelerate the flow in the free surface vicinity.

Acknowledgements: This paper has been elaborated within the contract no. 31041/2007-2010, financed by The National Centre for Programme Management, NASR, and contract no. 232/2007, IDEAS program, financed by Romanian National University Research Council.

References
1. JINESCU, V., Utilaj tehnologic pentru industrie de proces, Editura Tehnicã, Bucureºti, 1989
2. JINESCU, V., PRODEA I. M., Rev. Chim. (Bucharest), 58, no. 6, 2003, p. 582
4. MITSUHARU TERASHIMA, RAJEEV GOEL, KAZUYA KOMATSU, HIDENARI YASUI, HIROSHI TAKAHASHI, Y.Y. LI, TATSUYA NOIKE, CFD simulation of mixing in anaerobic digesters, Bioresource Technology no. 100, 2009, p. 2228
5. JINESCU, V., POPESCU, I. C., Rev. Chim. (Bucharest), 58, no.12, 2007, p. 1278
6. JINESCU, GH, VASILESCU, P., JINESCU ,C., Dinamica fluidelor reale în instalajii de proces, Editura Semne, Bucureºti, 2001
7. BÂRAN, GH., MÂNDREA, L., Influenºa geometriei reactoarelor de omogenizare asupra curgerii, Revista Hidrotehnica nr. 8, 1998, p.147

Manuscript received: 27.01.2010