Kasetsart J. (Nat. Sci.) 34 : 536 - 543 (2000)

Computational Fluid Dynamics for Predicting Mixing Behavior in Bakers' Yeast Fermenter

Thongchai Srinophakun[1] and Jessada Jitjaroenchai[2]

ABSTRACT

The Computational Fluid Dynamics software (CFD) is used to address the understanding of mixing phenomena of Bakers' Yeast Fermenter. This 1500 liter fermenter performed the fed-batch culture of *Saccharomyces cerevisiae* with molasses and ammonia with low yield and yeast concentration. The mixing behaviour is considered as the major defection of the amount of yield. An analysis of mixing phenomena using CFD shows the profile of liquid comparing between the current plant and the suggested configurations. The alternative configurations are reported in terms of sensitivity analysis. These parameters are: increase off bottom clearance to 0.48 meter, increase turbine diameter to 0.48 meter, lower operating liquid height to 1.38 meter and higher rotational speed to 400 rpm.

Key words: CFD, bakers' yeast fermenter, mixing

INTRODUCTION

The computational fluid dynamics (CFD) has been very powerful and spanned a wide range of industrial and non-industrial application areas. The application of CFD to predict internal and external flow patterns has risen dramatically because of the widespread availability of engineering workstations together with efficient solution algorithms and sophisticated pre- and post-processing facilities enabling the use of commercial CFD codes. The codes that are now on the market may be extremely powerful, but their operation still requires a high level of skill and understanding of the operator to obtain meaningful results in complex situations. This paper demonstrates the application of CFD to the low production yield problem in the pilot scale fermenter of Pilot Plant Development and Training Institute (PDTI) at King Mongkut's University of Technology Thonburi (KMUTT). The previous research presented many parameters which affected the production yield namely kinetics of fermentation, pH, the oxygen concentration and mixing behavior of the fermenter. Among those factors, the mixing behavior can be observed using the CFD model and applied to hint the possible ways to improve the low production yield problems.

There are a few works concerned on the computational fluid dynamics application in the stirred tanks. Abid *et al*. (1994) studied the three dimensional simulation of the stirred tanks filled

[1] Department of Chemical Engineering, Faculty of Engineering, Kasetsart University, Bangkok 10900, Thailand.

[2] Department of Chemical Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Bangkok 10140, Thailand.

with highly viscous fluid. In this work two types of impellers were under consideration : the paddle agitator and two-blade impeller. The Navier-Stokes equations were solved numerically using the SIMPLER algorithm for the pressure and velocity linkage. The ratio of turbine blade to the tank diameter D/T, the turbine width to the tank diameter W/T and the tip speed variation were also investigated. The results showed that the recirculation zone occurred near the tank bottom due to the friction of the tank wall. The eddies size of the liquid is increased with the Reynold's number for rotating turbine. However, this work did not consider the aspect of mixing of material which included the viscous dissipation function and the power consumption.

Rubens (1997) presented his work on the application of CFD to stirred tanks. He proposed the simplified methodology to obtain CFD results, using a commercial code, PHOENICS version 2.1. Two case studies were considered: fluid flow of polystyrene and flow of water in a stirred tank reactor. Attention was directed towards the influence of a disk type impeller inside the tank as well as the two dimensional type simulation. The graphical display showed streamlines along with velocity profiles for the simulated purpose.

The present work considered the mixing in the stirred tank. The simulations were performed using the commercial code, PHOENICS version 1.5, which was the shareware version. The results of vectors field, contour plots of the flow field were also presented using PHOTON version 1.0.

Pilot scale bakers' yeast fermenter

PDTI studied and developed the Bakers' yeast fermentation process in the industrial scale. The experiments were performed from the lab scale to the pilot scale. In pilot scale, batch and fed batch fermentation were also studied. The optimum substrate concentration was found in batch experiments as follow : temperature 30 °C, pH 4.5, shaking rate 150 rpm, fermentation time 24 hours. The efficiency of the transformation from sugar to cells was 0.28. The temperature and pH in the fed batch culture in the 2.5 liter fermenter were controlled at 30 °C and pH 4.5-5.5, respectively. This gave a high productivity and a good quality of cells. The production yield of batch culture at 0.183 was higher than of the fed batch at 0.313. The pilot scale fermenter (1,500 liter) was designed, using the experimental data from the lab scale in order to produce 1,000 kg of yeast per month. This pilot scale fermenter is presently operating for commercial purpose. However, the production yield of the pilot scale is lower than the lab scale.

MATERIALS AND METHODS

Mathematical modeling

To represent the fermenter behavior, mathematical model is generated under these following assumptions:

1. Two dimensional simulation in the radial and the axial directions of cylindrical coordinate.
2. Unsteady flow.
3. Incompressible fluid.
4. Isothermal system.
5. Symmetrical flow pattern in the angular direction.
6. Turbulent flow.
7. One phase simulation for liquid.

According to the assumption of symmetrical flow in angular direction and two dimensional simulation, the governing equations for this fermenter system can be simplified by reduction of some terms. The reduced form is obtained by ignore the gradient in angular direction. Thus, the governing equations are :

Mass conservation :

$$\frac{\partial \rho}{\partial t}+\frac{1}{r}\frac{\partial(\rho r v_r)}{\partial r}+\frac{\partial(\rho v_z)}{\partial z}=0 \qquad \ldots(1)$$

Angular direction momentum :

$$\rho\left(\frac{\partial v_\theta}{\partial t}+v_\rho\frac{\partial v_\theta}{\partial \rho}+\frac{v_\rho v_\theta}{\rho}+v_\zeta\frac{\partial v_\theta}{\partial \zeta}\right)=\mu\left[\frac{\partial}{\partial \rho}\left(\frac{1}{\rho}\frac{\partial(\rho v_\theta)}{\partial \rho}\right)+\frac{\partial^2 v_\theta}{\partial \zeta^2}\right]+\rho\gamma\theta \qquad \ldots(2)$$

Radial direction momentum :

$$\rho\left(\frac{\partial v_r}{\partial t}+v_r\frac{\partial v_r}{\partial r}+\frac{v_\theta^2}{r}+v_z\frac{\partial v_r}{\partial z}\right)=-\frac{\partial p}{\partial r}+\mu\left[\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial(r v_r)}{\partial r}\right)+\frac{\partial^2 v_r}{\partial z^2}\right]+\rho gr \qquad \ldots(3)$$

Axial direction momentum :

$$\rho\left(\frac{\partial v_z}{\partial t}+v_r\frac{\partial v_z}{\partial r}+v_z\frac{\partial v_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left[\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial v_z}{\partial r}\right)+\frac{\partial^2 v_z}{\partial z^2}\right]+\rho gz \qquad \ldots(4)$$

Where v_θ, v_r, v_z are the velocity component in angular, radial and axial, respectively. ρ is the liquid density and μ is the liquid viscosity while γ is the gravitational constant. All variables involved in the equations have SI units. The boundary conditions were made from the real situation. In this system, the tank wall friction or no-slip conditions have to be made. The no-slip conditions were set for the cylindrical tank wall and bottom of the tank. With this conditions the liquid velocity components parallel to the tank wall is zero. The rotating turbines conditions were set for two blade at the different level inside the flow field. This boundary condition was made by setting the angular (θ) velocity of liquid adjacent to the turbine blade to be equal to the rotating turbines. Baffle condition was built inside the flow field to disturb the angular direction of the flow component. The last boundary condition is for the body force. Gravity filed is the body force that has an important role to the moving fluid in the vertical direction

Initial condition is also needed to define the value of all dependent variables at the starting point of simulation. In this study, simulation begins from the stagnant liquid in the tank. When simulation begins the turbine suddenly starts moving with the full speed. Therefore, the initial conditions were set for all velocity components to be zero.

The standard design as shown in Table 1 which has proved to be the effective design in both energy and mixing aspects is set for the comparison of real operating tank geometry and the standard geometry. Therefore, the margin of each variable to the standard will be considered in the sensitivity analysis.

RESULTS AND DISCUSSIONS

Five different versions of sensitivity analysis were set to investigate all of the interested parameters. The results of simulation can be represented in many different methods: contours and vector plots, stagnant liquid height and maximum velocity components. The contours and vector plots allow the user to visualize the result in the overview aspect. The amplitude of all velocity components and its direction can be easily seen in Figure 1. The stagnant liquid height is the term

Table 1 Comparison of the standard and the real operating geometry (Tank diameter = 0.96 m).

Variables	Standard geometry		Real scale
	Low	High	
Tank diameter		0.96	
Off bottom clearance	0.16 m	0.48 m	0.32 m
Number of baffles	4	4	4
Baffle width	0.096 m	0.096 m	0.096 m
Turbine blade diameter	0.24 m	0.48 m	0.397 m
Turbine blade width	0.048 m	0.096 m	0.064 m
Liquid height (for 1 level turbine)	0.96 m	0.96 m	1.65 m

defined for representing the amount of liquid, which is stagnant and does not promote mixing. This height normally occurs on the top part of liquid in the tank. The lower of this value indicates the better of mixing. The maximum velocity components are very useful in examing the magnitude of turbulent in each direction. Axial and radial velocity components show the amount of mixing in the vertical direction. The higher value of these two components, the better in mixing obtains.

Changing of the off bottom clearance

There are two aspects; to vary the off bottom clearance to 0.16 and 0.48 meter. The velocity results on the plane parallel to the x direction are presented in Figure 1.

There are three vector parts, lowest, base case, and the highest off bottom clearance. It is obviously seen the turbine blades have highest radial flow velocity vector. All of this sub-picture has the same reference value of velocity vector at 1.00 m/s. The color shade inside the velocity field has the same meaning of the contour plot. Because of three dimensional vectors, the representing color is necessary in order to show the magnitude of the vector. In the first sub-picture of Figure 1 (left side), the flow is quite complicated around the bottom of the tank due to the turbine position are very low. But the liquid is quite stagnant at the top of the tank. Similar to the middle one, which is the base case configuration, the top lump of water does not move much comparing to the last sub-picture. At the value of clearance of 0.48 meter all of the liquid inside the tank performs well mixing.

In Table 2, the lowest stagnant liquid height identifies from the highest off bottom clearance. The maximum radial and axial velocity can also be seen for the highest off bottom clearance. This high velocity component improves the mixing for the axial and radial direction in the tank.

Changing of liquid height

The suitable working liquid height for the standard geometry mixing tanks with one level turbine should be equal to the diameter of the tank. There is no standard configuration for the double level turbines. Therefore, the simulations with different liquid height should be observed. The two different liquid heights of 0.96 and 1.65 meter was investigated . At the beginning of fermentation, the liquid height is 0.96 meter (or equal to 700 liter). And the maximum working liquid height in the stirred tank is 1.65 meter. The velocity profiles on the normal plane are shown in Figure 2 and Table 3. The shape of the flow field for each liquid height is different because of the limitation of the

Table 2 Variation in the off bottom clearance.

	C = 0.16 m	C = 0.32 m	C = 0.48 m
Stagnant liquid height, m	0.41	0.33	0.165
Maximum radial velocity, m/s	0.606	0.631	0.632
Maximum axial velocity, m/s	0.7509	0.869	1.04

Table 3 Variation in the operating liquid height.

	H = 0.96 m	H = 1.65 m
Stagnant liquid height, m	0	0.33
Maximum radial velocity, m/s	0.813	0.631
Maximum axial velocity, m/s	1.28	0.869

Table 4 Variation in the impeller diameter.

	D = 0.24 m	D = 0.40 m	D = 0.48 m
Stagnant liquid height, m	0.41	0.33	0.25
Maximum radial velocity, m/s	0.188	0.631	0.968
Maximum axial velocity, m/s	0.448	0.869	1.49
Maximum angular velocity, rad/s	3.9	6.5	8.24

Table 5 Variation in the rotational speed.

	N = 200 rpm	N = 345 rpm	N = 400 rpm
Stagnant liquid height, m	0.38	0.33	0.247
Maximum radial velocity, m/s	0.366	0.631	0.823
Maximum axial velocity, m/s	0.504	0.869	1.36
Maximum angular velocity, rad/s	3.77	6.5	8.48

Table 6 Best parameter comparing to the base case.

	Base case	Best case
Stagnant liquid height, m	0.33	0
Maximum radial velocity, m/s	0.631	1.2
Maximum axial velocity, m/s	0.869	1.7
Maximum angular velocity, rad/s	6.5	10.7

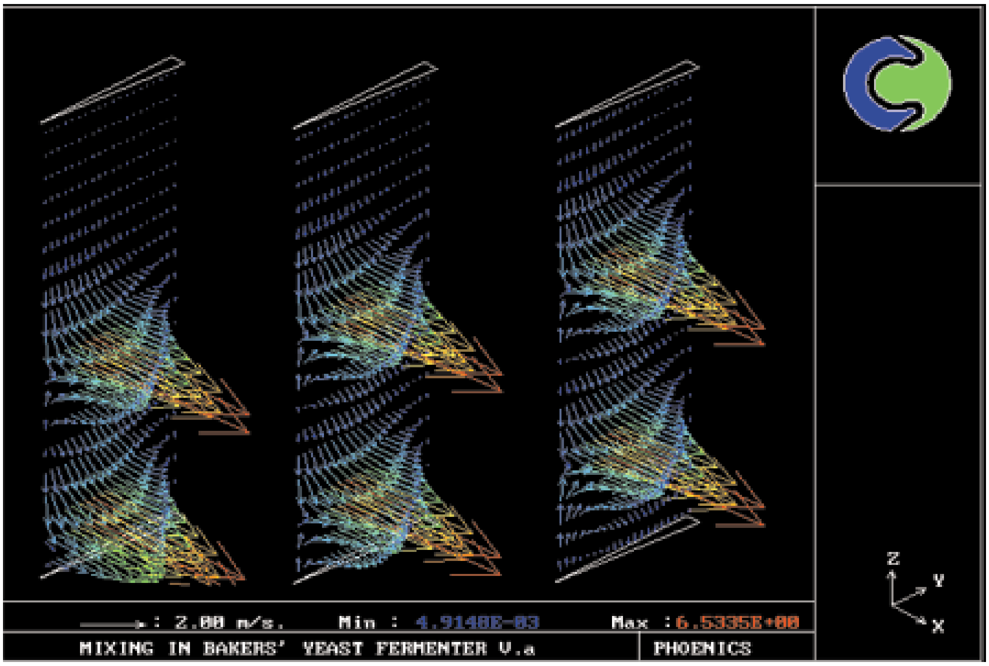

Figure 1 3-D view vector results of the bottom clearance.

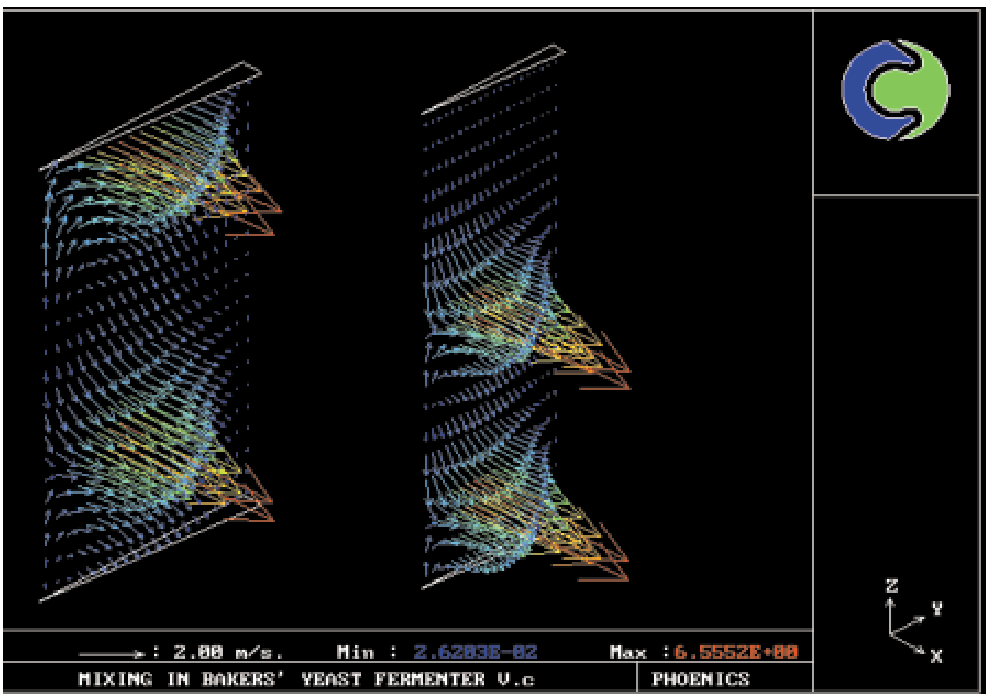

Figure 2 3-D view for the simulation with different liquid height.

screen width.

It can be seen that at the height of 0.96 meter liquid level is just over the top level turbine. The investigation of flow pattern inside the fluid element is going throughout the tank volume. This is the well mixing phenomena. For the base case result, the explanation is the same as before. That is the top part of liquid is stagnant. In the real operation, liquid level is changed every minutes. So, this result does not imply unwell mixing in the tank. However, every factor needs to be checked simultaneously in the following analysis before makes the conclusion. The zero stagnant height with higher radial and axial velocity can be seen in the low liquid level. Very good mixing occurs at lower height of liquid. It can be seen that the mixing is better at the beginning stage of fermentation. Certainly, increasing of liquid height in fed-batch culture will change the mixing characteristic. The mixing performs well until the stagnant liquid height is developed.

Changing of turbines diameter

Comparing to the standard geometry, the pilot scale geometry is in the middle of the range of standard geometry. Therefore two different value of turbines diameter should be investigated. One smaller and one bigger than the pilot scale size. The contour plots of angular velocity of each version are shown in Figure 3. The results show exactly the positions of the turbine tips from the smallest to the largest (on the left to the right picture). This contour plots cannot represent to mixing phenomena. The only way to explain is the implementing of velocity vectors.

The results show that with the smallest turbine diameter, the mixing pattern and velocity are rather small. The fluid elements do not mix

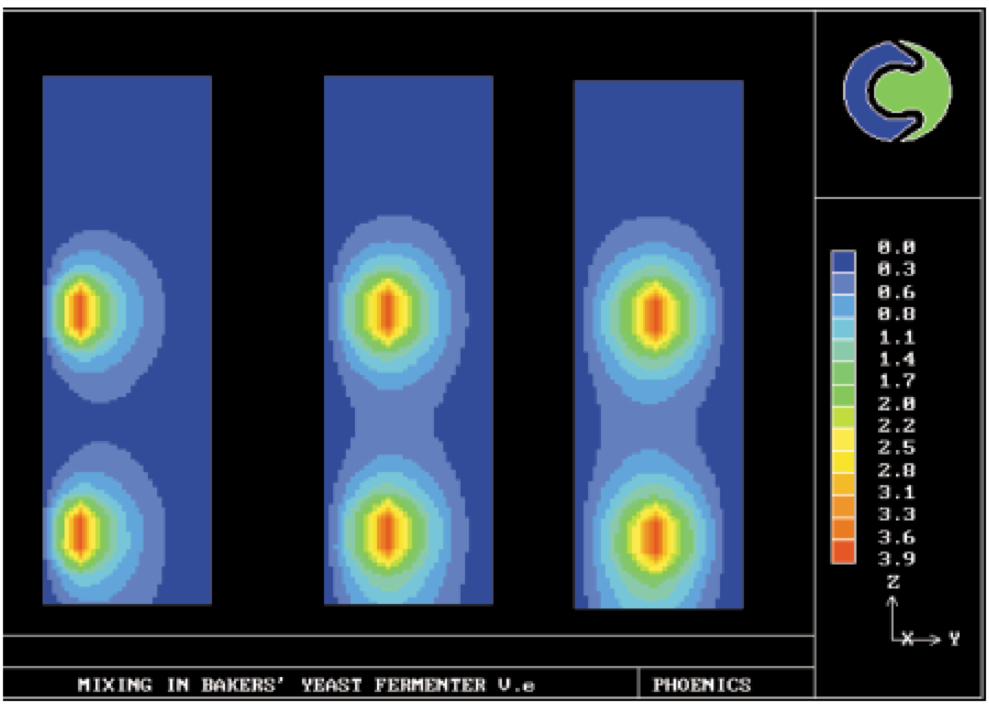

Figure 3 Contour plots for different turbine diameter.

throughout the tank volume. This is very bad mixing flow pattern. However, the rest of the results show that the larger turbine blade gives the better mixing flow pattern. Of course; the power consumption in mixing will increase. Therefore, it can be concluded that the biggest turbine gives the best mixing. The conclusion are shown in Table 4. The lowest stagnant liquid height, highest of all velocity components can be obtained from the larger impeller turbine. Therefore, it can be concluded that the larger turbine is effective in the mixing aspect, which certainly consumes more power for rotation than in the small turbine.

Changing in rotational speed

Three different speeds of impeller were under consideration: 200, 345 and 450 rpm. The base case design speed is 345 rpm. As expected, the faster speed introduces more turbulent, bigger recirculation zone and eddies style. Therefore, the better mixing stage achieved. In Table 5, it can be seen that the fastest rotational speed provides the lower stagnant liquid height and highest velocity for every component.

CONCLUSIONS

The best parameters can be gathered from each simulation: increase off bottom clearance to 0.48 meter, increase turbine diameter to 0.48 meter, lower operating liquid height to 1.38 meter and higher rotational speed to 400 rpm. Table 6 shows the specific values comparison.

More turbulent, larger eddy size, high magnitude of velocity components and zero stagnant liquid height can be observed in the best case design. The results also point out that the mixing in the best case is more than 80% increasing in all velocity components However, the new design requires some investment cost. Another suggestion was made for the pilot scale for the alternative way of improvement without additional investment cost namely the maximum operating liquid height reduces to about 1.1 to 1.2 meters.

LITERATURE CITED

Abid, M., C. Xuereb, and J. Bertrand. 1994. Modeling of the 3D hydrodynamics of 2-blade impellers in stirred tanks filled with a highly viscous fluid. The Canadian Journal of Chemical Engineering 72 : 184-193.

Rubens, M.F. 1997. Application of CFD to stirred tanks, PHOENICS Journal 1: 15-25.

Received date : 01/06/00
Accepted date : 31/10/00