

CFD Investigation of Interface Location in Stirred Tanks with a Concave Impeller

P. Parvasi, R. Janamiri, A. Sinkakarimi, I. Mahdavi, M. Safdari, M. H. Sedaghat, A. Hosseini, and M. Karimi

Abstract—In this work study the location of interface in a stirred vessel with a Concave impeller by computational fluid dynamic was presented. To modeling rotating the impeller, sliding mesh (SM) technique was used and standard k- ϵ model was selected for turbulence closure. Mean tangential, radial and axial velocities and also turbulent kinetic energy (k) and turbulent dissipation rate (ϵ) in various points of tank was investigated. Results show sensitivity of system to location of interface and radius of 7 to 10cm for interface in the vessel with existence characteristics cause to increase the accuracy of simulation.

Keywords—CFD, Interface, Concave impeller, turbulence model.

I. INTRODUCTION

MIXTURE is one of the most common operations in chemical and biochemical processes and identification of the type of fluid flow can play an important role to optimize it. Stirred vessels are widely utilized in chemistry, pharmaceutical processes, sewage refinement, and other industrial fields. The most processing operations containing single-phase flow mixture are carried out in mechanical stirred vessels. The optimized design and mixture operation efficiency are important parameters in production quality and expenses. Fluid flow is 3-dimensional and complicated in stirred vessels. There is an extremely turbulent rotational flow in the vicinity of the impeller. In the recent years, computational fluid dynamic (CFD) techniques are comprehensively employed instead of using trial and error methods to determine flow fluid details for a collection of fluid, impeller geometry, and tank [1]. Another advantage of

using CFD prediction compared to experimental method is the possibility of modeling the system with the real dimensions.

Concave impeller with the trade mark (CD-6), (1980) brought a massive revolution in the field of turbulent gas circulation. The innovative idea of concave blade was developed at Delf University in 1970 by a group of researchers led by J.M. Smith who is BOC professor of process engineering at Surry University in England. Vant Riet (1976) studied various types of impellers and presented the meaning of concave impeller usage [2]. Warmoeskerken and Smith (1989) expanded the idea and explained the optimized efficiency of concave impeller compared to the flat impellers in terms of decreasing the hole formation next to the blades [3]. Semi-circular impellers are commonly used in industry. Recently, blades with higher concavity have been proposed by Hjorth [4] and Middleton [5]. Another type of concave impeller is Bakker turbine (1998) with the trademark (BT-6) [6]. This new product is the result of effort and collaboration in Chemineer Companz. Satio [7] studied the performance of such impellers and he concluded that at most of the conditions with these deeper blades, gas spreads through the blade instead of spreading through the big holes next to the blades.

Till date, all of the works have been carried out in two-phase systems and none of them has been in single-phase system. In this work, the investigation is based on a single-phase flow system.



Fig. 1 Concave impeller

II. COMPUTATIONAL METHOD

CFD model is comprised of the numerical solution of conservation equations in laminar and turbulent flow regimes. Hence, theoretical predictions are obtained by the solution of continuity and Reynolds average Navier Stock equations, simultaneously. Continuity and Momentum equations for non-compressible Newton fluid are as follows:

Payam Parvasi is with the Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Dashtestan, Iran (phone: + 09122049487; e-mail: oayamparvasi@yahoo.com).

Reza Janamiri is with the Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Iran (phone: +98 936-495-8029; e-mail: rezajanamiri@yahoo.com).

Azam Sinkakarimi is with the Faculty of Chemical Engineering, Islamic Azad University, Ahar Branch, Iran. (e-mail: azamsinkakarimi@hotmail.com)

Iman Mahdavi, Department of Chemical Engineering, Islamic Azad University, Mahshahr Branch, Mahshahr, Iran (e-mail: imanmahdavi7@gmail.com)

Mohammad Safdari is with the National Iranian Drilling Company, Iran (phone: +989171779108; e-mail: mamadsafdari@inbox.com)

Mohammad Hossein Sedaghat, Faculty of Chemical Engineering, Islamic Azad University, Dashtestan Branch, Dashtestan, Iran (phone: +98 917-773-7924; e-mail: m.sedaghat88@yahoo.com)

Amir Hosseini is with the Faculty of Chemical Engineering, Science and Research University, Bushehr Branch, Iran. (phone: +989173707875; e-mail: s.amirhosseini7875@yahoo.com)

Mohsen Karimi is with the Faculty of Chemical Engineering, Islamic Azad University, Ahar Branch, Iran (e-mail: m_karimi6400@yahoo.com)

$$\frac{\partial u_j}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial u_i}{\partial t} + u_i \frac{\partial u_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2} - \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

$$\tau_{ij} = \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3)$$

u_i is velocity in direction i , ρ is density, P is pressure, τ_{ij} is tension tensor. At turbulent condition, these set of equations are accurately solved using Direct Numerical Simulation (DNS) to achieve the velocity field variations. Governing equations are Navier Stock Average time. The results are discretized and linearized using finite volume method.

A. CFD Method

In this work, a set of non-linear equations are solved by linearization of continuity and momentum equations. Concave impeller is formed and meshed at standard condition. Computational meshing is comprised of two volumes: the inner volume containing rotational cylinder that surrounds the turbine and the outer volume containing the constant volume of the rest of the tank. Structured meshes are comprised of non-uniform hexagonal grids across these two volumes. Fig. 2 shows the inner volume of the impeller with the constructed meshes over the interfaces. For a better description of the impeller, meshing is denser in the vicinity of it. In other words, while proceeding from the impeller to the walls. The density of meshing is reducing and in baffles, it is increasing again.

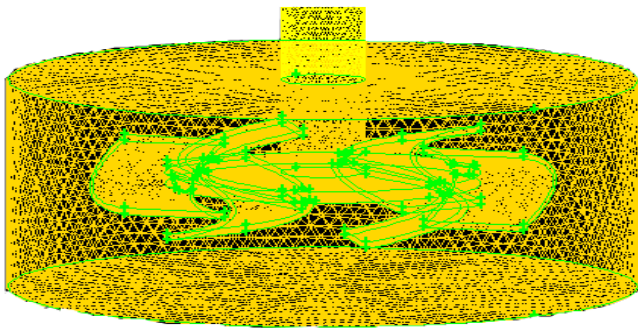


Fig. 2 The inner volume of concave impeller with triangular meshing for a clockwise rotation

TABLE I
 SYSTEM CHARACTERISTICS

Impeller	Dimension
Tank diameter(m)	0.3
Impeller diameter(m)	0.1
Disk diameter(m)	0.66
Disk thickness (m)	0.0035
Blade height (m)	0.25
Blade length (m)	0.25
Blade thickness (m)	0.002
Blade angle(degree)	45
Bottem clearance(m)	0.01

The number of computational grids in the tank is 640000. In the current work, increasing the number of computational grids, no changes observed in the convergence velocity and

only the computations became lengthier. Therefore, no increase was occurred in the number of grids.

Walls boundary condition is non-slip and dynamic boundary condition has been applied for fluid volume. Turbine's velocity is $N=300\text{rpm}$. To simulate the turbine's rotation, it was solved firstly by MRF and the results were used as the initial guess for Sliding Mesh (SM) model. Secondary degree separation method and standard turbulence model $k-\epsilon$ has done in the single phase system.

Velocity-pressure coupling is done by SIMPLE pressure technique. Calculations are done in the time period of 0.001 until the residual results of continuity equations, velocity vectors, k and ϵ have reached to lower than 10^{-8} . Water was used as the experimental fluid in 25°C . Table I shows the dimensions of the tank and rotating turbine.

III. RESULTS AND DISCUSSIONS

In the middle of the tank there is an especial flow pattern that separated the upper and lower rotational zones. As it is obvious in Fig. 3, a stronger rotational pattern has been created in a bigger volume of the tank. A low velocity zone is resisting at the upper side of the tank.

Cylinders with the length of 6.5m with variable radiuses have been used to investigate the location of interface middle zone. Radiuses are sat on 5.75, 7.5, 9, 10.5 and 11.75.

Fig. 4 shows the average angle velocities in various parts of the tank in $r/R = 0.7, 0.6, 0.5$ and 0.38 in terms of z/H at the rotating velocity of 300 rpm. Velocities have been normalized with the turbines edges velocity $V_{tip} = \text{IIND}$.

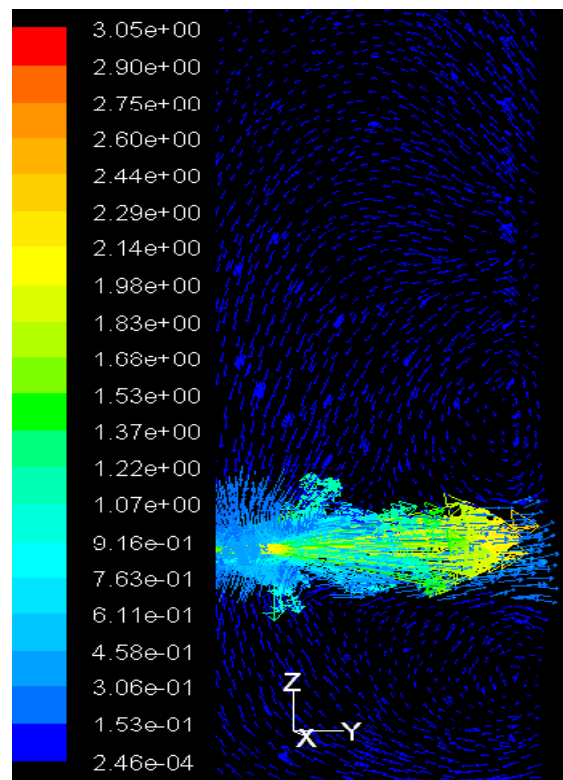


Fig. 3 Flow pattern

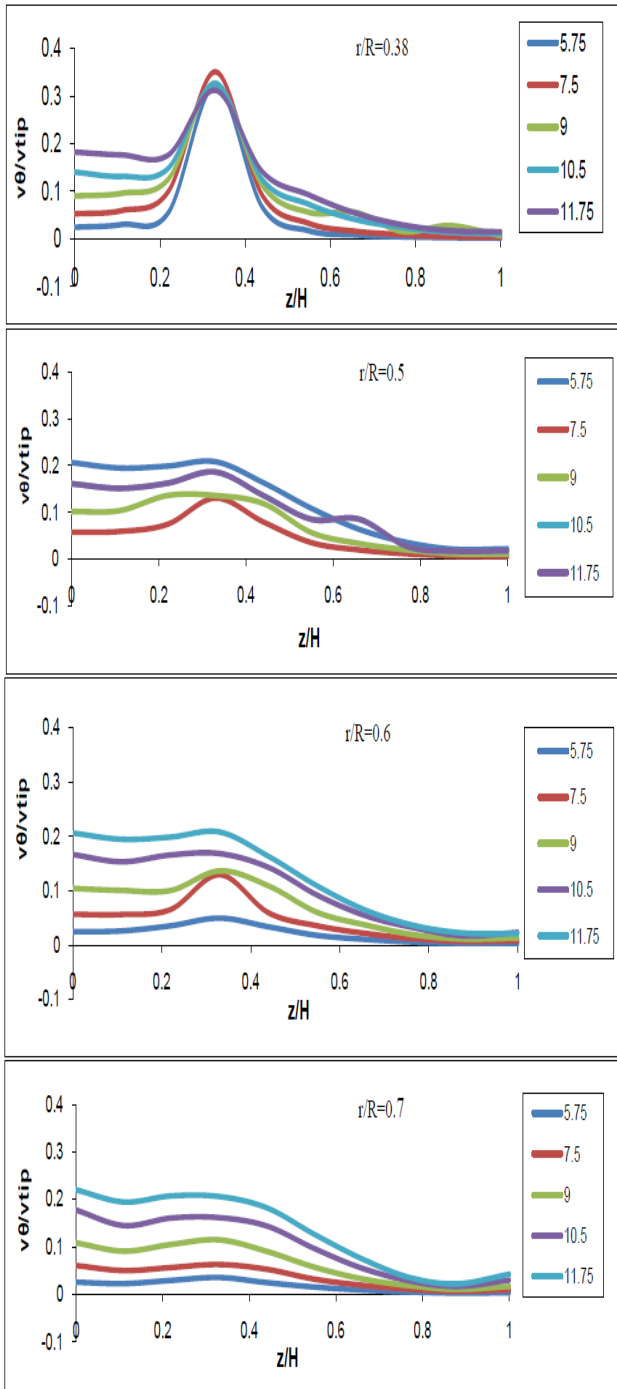


Fig. 4 Normalized average angle velocity profile

Angle flow is in direction of rotation and in all surfaced of the axis. As it is obvious in the figure the plot becomes out of its general shape with increase in distance to the turbine. In the radius of 10.5m ($r/R = 0.7$) the general shape of velocity figure has been gone. In $r/R = 0.6$ the performance is much better.

According to Fig. 5, the flow is from blades to out and they change in axis direction. With radius increase to walls the velocity decreases and the plot becomes flatter.

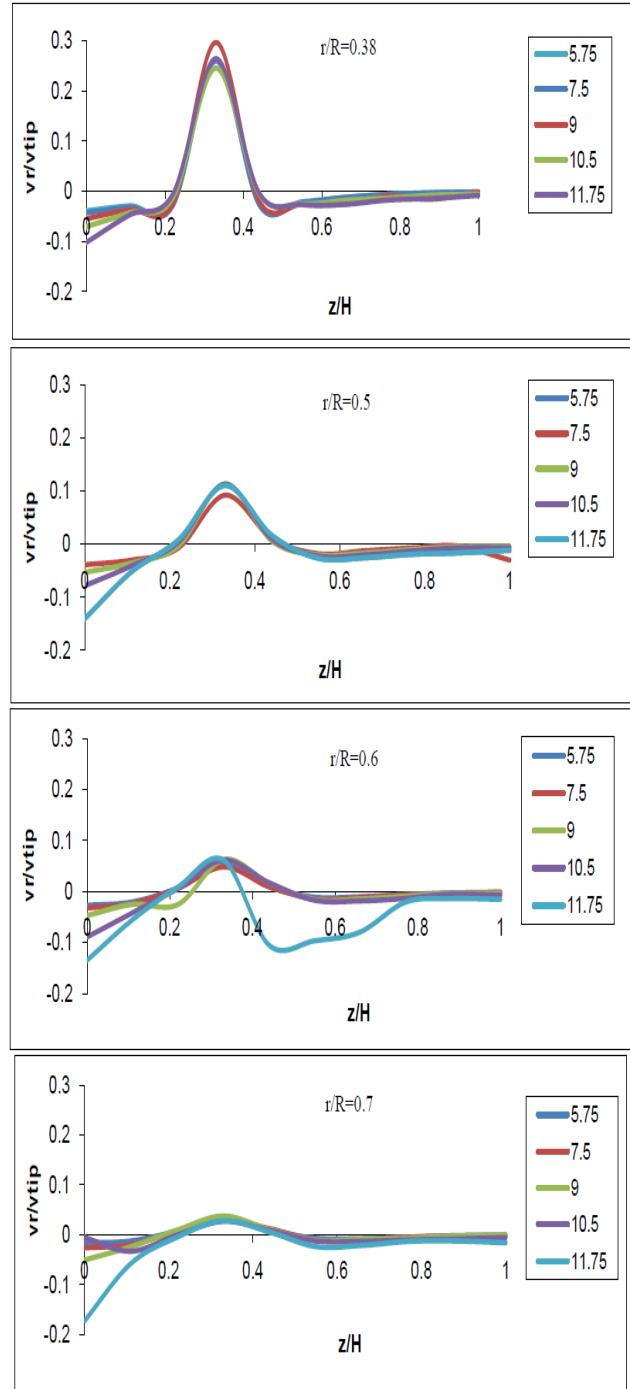


Fig. 5 Normalized average radius velocity profile

According to Fig. 6, the results of axis velocity shows that at the upper parts of the tank the fluid is flowing upward in the wall direction and then it goes downward in a cycle. This pattern is opposite in the bottom of the tank where it is intensified because of non-symmetrical position of turbine in the tank.

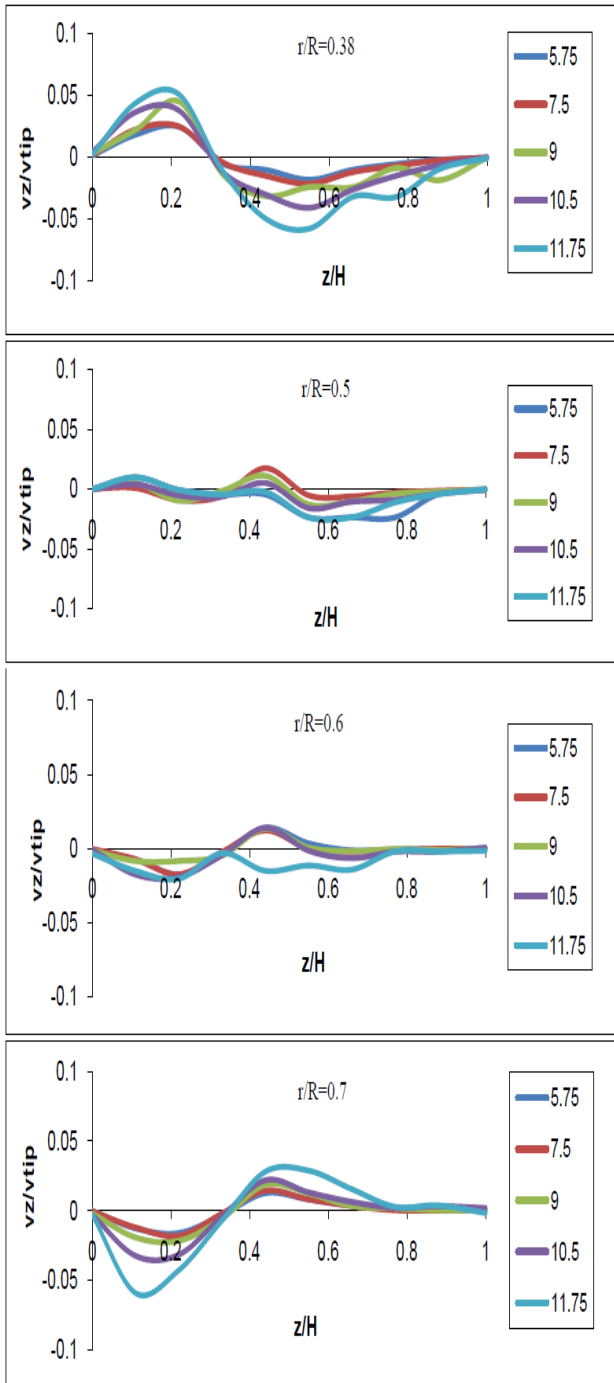


Fig. 6 Normalized average axis velocity profile

Figs. 7-9 show the normalized turbulent kinetic energy and the rate of energy loss at different parts of the interface in the tank.

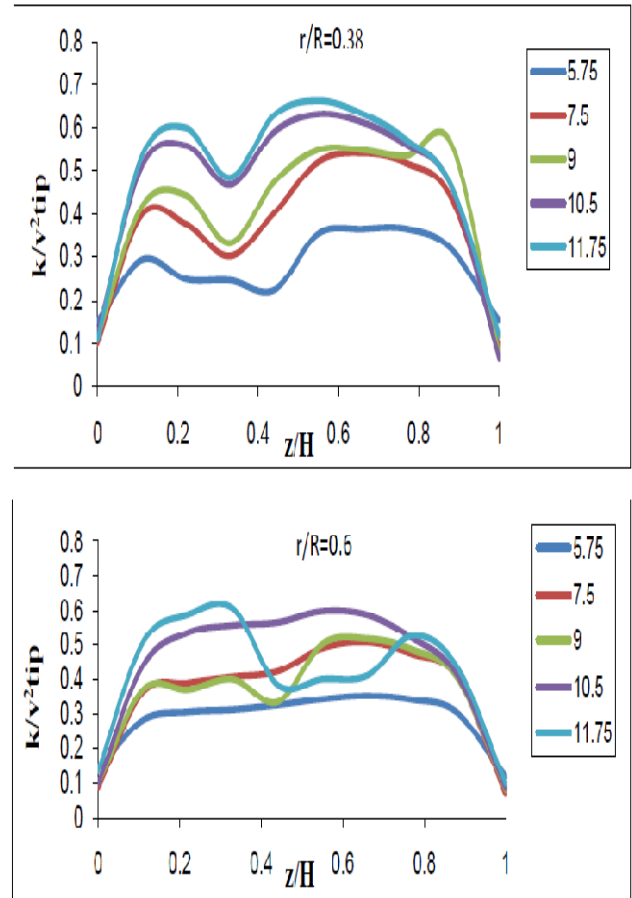


Fig. 7 Normalized turbulent kinetic energy at different parts of the interface

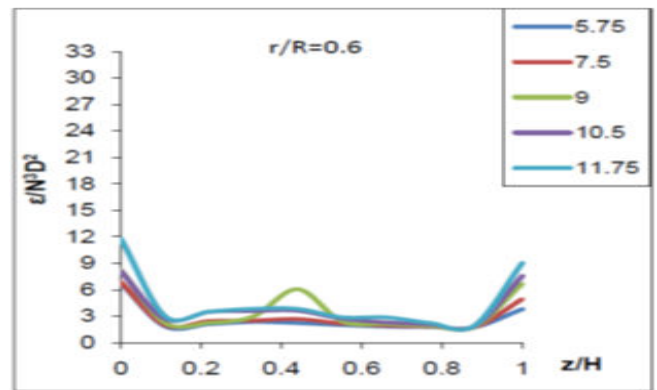


Fig. 8 Normalized turbulent energy loss at different parts of the interface

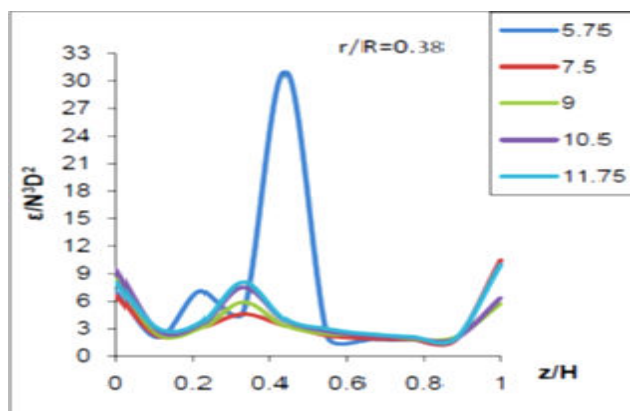


Fig. 9 Normalized turbulent energy loss at different parts of the interface

Regarding these figures, it is shown that for the interface in so close or too far distances from the turbine the general shape of the plot diverges that leads to increase in simulation error and decreases the accuracy.

REFERENCES

- [1] Ranade VV, Bourne JR, Joshi JB. "Fluid mechanics and blending in agitated tanks". *Chem Eng Sci*,46:1883-93(1991).
- [2] Van, t Riet, k., Boom, J .M and Smith, J.M, "power consumption, impeller coalescence and recirculation in aerated vessels", *Trans IChemE*,54:124-131(1976).
- [3] Wamoeskerken, M. M. C. G .and Smith, J. M., "The hollow blade agitator for dispersion and mass transfer", *Trans IChemE, Chem Eng Res Des*,67:193-198(1989).
- [4] Hjorth, S., Imeller apparatus, *US Patent 4779990*, (1998).
- [5] Middleton, J. C., Agitators, *US Patent 5198156*,(1993).
- [6] Bakker, A., Myers, K. J. and Smith, J. M., "How to disperse gases in liquids", *Chem Eng*, 101(12): 98-104(1994).
- [7] Satio, F., Nienow, A. W., Chatwin, S. and Moore, I. P. T., "Power, gas dispersion and homogenization characteristics of Scaba SRGT and Rushton turbine impellers", *J Chem Eng Japan*, 25(3): 281- 287(1992).