

Coil and Jacket's Effects on Internal Flow Behavior & Heat Transfer in Stirred Tanks

B. Lakghomi, E. Kolahchian, A. Jalali, and F. Farhadi

Abstract—Different approaches for heating/cooling of stirred tanks, coils and jackets, are investigated using computational fluid dynamics (CFD). A time-dependant sliding mesh approach is applied to simulate the flow in both conditions. The investigations are carried out under the turbulent flow conditions for a Rushton impeller and heating elements are considered isothermal.

The flow behavior and temperature distribution are studied for each case and heat transfer coefficient is calculated. Results show different velocity profiles for each case. Unsteady temperature distribution is not similar for different cases. In the case of the coiled stirred vessel more uniform temperature and higher heat transfer coefficient is resulted.

Keywords—CFD, Coil & Jacket, Heat transfer, Stirred tank.

I. INTRODUCTION

STIRRED tanks are frequently used in chemical processes for various operations such as Liquid mixing, Gas-Liquid and Liquid-Liquid mass transfer, Crystallization and chemical reactions. These operations are strongly affected by mixing which is pertinent to impeller's type and geometry. Because of increasing importance of product's quality, especially in chemical and pharmaceutical industries, a complete understanding of mixing and hydrodynamic effects on operations in stirred tanks is essential. Many studies have been done on impellers with different geometries using experimental and computational fluid dynamic methods [1]. CFD methods are used to obtain a clearer view of mixing with less cost. Heat transfer in stirred tanks can be simulated using these methods.

Heat transfer in stirred tanks is of great importance in chemical industry. Coils and jackets are applied for heating or cooling of process fluid in agitated vessels. Both arrangements have positive influences and drawbacks in controlling bulk temperature and they should be weighed carefully before deciding which arrangement should be chosen in each design [2].

Coils have the tendency to affect the flow because they drag the flow circulation.

In last two decades, many researches have been done on turbulent mixing in stirred tanks [3]. In early 90's studies, rotating impeller was not simulated independently and

experimental data was employed as boundary conditions at the surface of impeller, which is named as Impeller "Boundary Condition". In the middle of 90's, fully predictive methods such as Sliding Mesh (SM), Inner-Outer Method (IO), and Multiple Reference Frame (MRF) were developed, which provided simulation of impeller without experimental data.

A. Simulation Geometry

For modeling the geometry and generating grids, Gambit preprocessor is applied. An unstructured tetrahedral grid is used in both cases. The number of grids in coiled stirred tank is 344740 and in jacketed one is 300967. The studied model is a stirred tank with a Rushton impeller. The geometry of simulated models is as following:

TABLE I
DIMENSIONAL DATA OF AGITATED VESSEL

tank diameter	0.35m	Shaft height	0.3m
Tank height	0.45m	Disk diameter	0.08m
Initial water level in tank	0.4m	Blades height	0.03m
Baffles height	0.45m	Blades width	0.05m
Baffles width	0.03m	Blades thickness	0.003m
Baffles thickness	0.005m	Coil pipe diameter	0.001m
Shaft diameter	0.02m	Coil diameter	0.2m

B. Numerical Methods and Procedures

Fluent 6 is a commercial CFD package, which can be used to solve Navier-Stokes equations using Finite-Volume methods [4]. In this study, this software is used to solve momentum, continuity, turbulence and energy equations in two stirred tanks one of which is heated with a coil and the other is heated with a jacket. Since the flow is turbulent, Reynolds averaged equations must be used.

In this approach, main variables are divided into two parts: average term and deviation term. For example for velocity:

$$u_i = \bar{u}_i + u'_i \quad (1)$$

Whereas $\bar{u}_i$ is the average term and u'_i is the deviation term. If this kind of equations are replaced in continuity and momentum equations and they are averaged the result will be: (over bar is not shown)

B.Lakghomi, E.Kolahchian & A.Jalali, are senior students at Chemical Engineering Department of Sharif University of Technology (SUT), Iran (e-mails: lakghomi@che.sharif.edu, Aydinjalali@che.sharif.edu).

F. Farhadi is an Associate Professor at Chemical Engineering Department of Sharif University of Technology (SUT), Iran (e-mail: farhadi@sharif.edu).

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial t} = 0 \quad (2)$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\rho u'_i u'_j) \quad (3)$$

All of the terms are analogous to those of Navier-Stokes equations. But, it should be considered that these are average values and also another term showing turbulence has appeared $-\rho u'_i u'_j$. For calculation of this term, Boussinesq theory [5] is applied:

$$-\rho u'_i u'_j = \mu_t \left(\frac{\partial \mu_i}{\partial x_j} + \frac{\partial \mu_j}{\partial x_i} \right) - \frac{2}{3} (\rho_k + \mu_t) \frac{\partial \mu_i}{\partial x_j} \delta_{ij} \quad (4)$$

K- ϵ Standardized method, which is a two-equation method, is applied to simulate turbulence. Three-equation methods such as LES can give more comprehensive and accurate results but they require much more computational effort, so their usage would not be beneficial.

For simulation of rotation of impeller, time-dependent, fully predictive sliding mesh technique is employed. In this method flow domain is divided into two parts: Inner Rotating and Outer Stationary. An interface between two regions is defined and the grids of each region are sliding on each other and fluxes are computed over the intersection of grids. In the rotating impeller region a modified set of balance equations is solved [6]:

$$\frac{\partial}{\partial x_j} (u_j - v_j) = 0 \quad (5)$$

$$\frac{\partial}{\partial t} \rho u_i + \frac{\partial}{\partial x_j} \rho (u_j - v_j) u_i = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (6)$$

v_j is a term that arises because of mesh motion.

This method provides more accurate results compared to other methods [4]. Especially, in the case of a coiled stirred tank since the interaction between impeller and coil is noticeably high, using SM technique will be advantageous. Coil and tank walls are considered as no slip surfaces.

For tracking the changes of surface shape with time, VOF model is applied. VOF is a two-phase model, mostly applied for stratified flows [6]. This model is based on the presumption that phases do not diffuse in each other. For each phase a variable, its volume fraction, is defined. The continuity equation will be [4]:

$$\frac{\partial \alpha_q}{\partial t} + \vec{v} \cdot \nabla \alpha_q = 0 \quad (7)$$

Volume fraction equation will only be solved for secondary phase. For the primary phase, volume fraction is solved due to the constraint below:

$$\sum_{q=1}^n \alpha_q = 1 \quad (8)$$

In this way momentum equation will be:

$$\frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla P + \nabla \cdot [\mu (\nabla \vec{v} + \vec{v} T)] + \rho \vec{g} + \vec{F} \quad (9)$$

Which is dependant on volume fraction due to ρ & μ . Energy equation will be:

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\vec{v} (\rho E + P)) = \nabla \cdot (K_{eff} \nabla T) + S_h \quad (10)$$

In which temperature and energy are treated as weighted average:

$$E = \frac{\sum_{q=1}^n \alpha_q \rho_q E_q}{\sum_{q=1}^n \alpha_q \rho_q} \quad (11)$$

The sliding zone is rotating at 200-rpm angular velocity and impeller is considered stationary relative to it. In the coiled tank, walls are assumed adiabatic and coil surface is isothermal and 100°C because it is carrying condensing steam. In the case of jacketed tank, walls are isothermal at 100°C because jackets contain condensing steam. For pressure-velocity coupling PISO algorithm is used which is suitable for unstructured meshes and two-phase VOF model [4].

For discrimination of momentum, energy and turbulence equations, second-order up winding scheme is used, which show further development of accuracy over first order methods, due to [7].

Convergence is considered to be achieved when scaled residuals fell down 5×10^{-7} for energy and 5×10^{-4} for others for each time step. The time step is set to 0.0005. Higher values for time step will lead to convergence problems, considering that both sliding mesh and VOF require small time steps to converge [8].

II. VELOCITY PROFILE

Coil drags the flow, so the areas with higher velocity are wider in the case of the jacketed vessel. The velocity profile is affected by existence of coil and the area near the coil will have less velocity (Fig. 1). Coil directs the velocity vectors moving upward to the center of vessel and changes the flow behavior in this way. So, the areas in top of the impeller in the coiled vessel bear higher velocities (Fig. 1). The velocity contours are not symmetrical in this case because of effects of coil on flow.

III. POWER CONSUMPTION

As coil drags the flow, more power is required while mixing a coiled stirred tank. This is shown through our calculations. It is shown by that power can be calculated by [3]:

$$P = \int \rho \varepsilon dv \quad (12)$$

According to these calculations, essential power for impeller in the coiled stirred tank is 6.4 W, while in jacketed stirred tank 6.2 W power is required. So, it is apparent that using a coil to heat (or cool) a tank costs more energy in comparison with using a jacket.

In this case, the difference of power consumption between the coiled tank and jacketed tank is not so much, but if a process fluid with a higher viscosity is used, this difference will be noticeable. As a result, for highly viscous fluids application of coil is completely rejected regardless of other selection criteria.

IV. HEAT TRANSFER AND TEMPERATURE DISTRIBUTION

As it is observed, the velocity profile is quite different near a coil and a jacket. This factor seems to affect the temperature distribution in the tank. The results of CFD simulation approve this theory. As it is observed in Fig. 2, in the jacketed case, there is always a zone in the center of vessel that tolerates a noticeable lag in being heated. In this case, heat at walls near the impeller is distributed uniformly but far from the impeller at walls heat accumulates, and there is always a noticeable lag in heating the center of vessel.

Temperature standard deviation is a factor that can be used in order to investigate temperature uniformity in the tank. It is shown in Fig. 3 that temperature average deviation is noticeable in the jacketed case in comparison with the coiled vessel, and it increases with passage of time. This means that heat accumulates in some areas, and is not well distributed to the center of tank.

In a coiled stirred tank, more uniform temperature distribution is expected because of two reasons: (1) since heat transfer is dependant on velocity profile and turbulence, coil's placement in a more turbulent region leads to better heat transfer, (2) Coil surface has a uniform distance all through the vessel whereas in the case of jacket the central zone is always far from the walls. This is backed with results from CFD simulation. By passage of the time, heat is uniformly distributed in most of the coiled vessel as shown in Fig. 4. The areas that obviate this behavior involve a region in bottom of the impeller, because of the shape of the velocity profile, and an area close to the coil. Around the impeller, heat transfer rate is higher in comparison with other areas because of higher a turbulence. As it is shown in the Fig. 3 temperature standard deviation is much less than the jacketed situation and it decreases with time, which can be a sign for uniform mixing.

Heat transfer coefficient can be calculated by both empirical equations and CFD approaches. In the case of jacketed tank, the empirical equation is [9]:

$$h_p = (0.74k / D_T) Re^{0.67} Pr^{0.33} (\mu / \mu_w)^{-1.4} \quad (13)$$

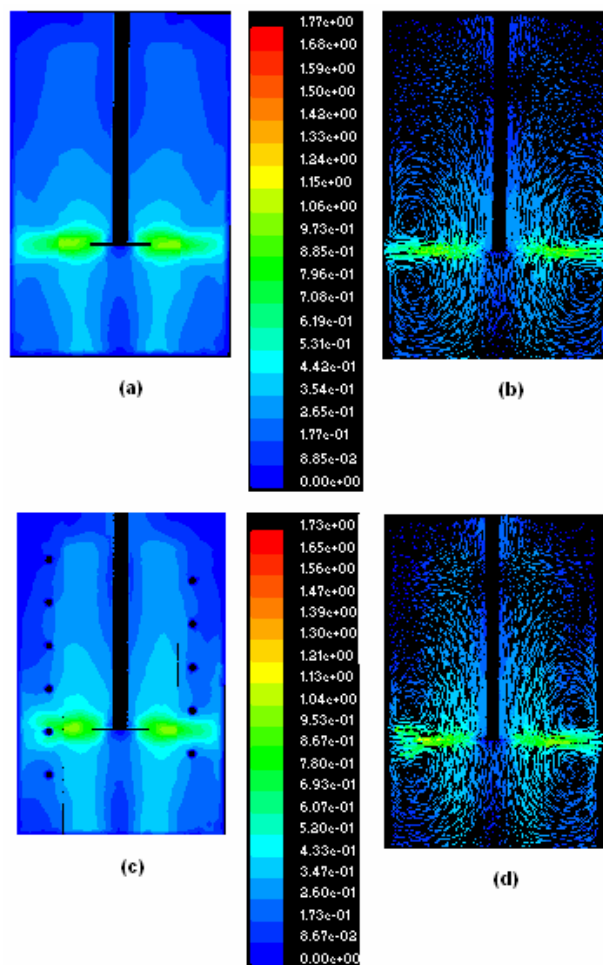


Fig. 1 (a) Velocity contours for the jacketed tank, (b) Velocity vectors for the jacketed tank, (c) Velocity contours for coiled tank, (d) Velocity vectors for coiled tank

For the case of present study, heat transfer coefficient using this approach is 4048W/ (m2.k).Whereas, numerical simulation shows the value of 2831W/ (m2.k).

In the case of coiled vessel, the suggested equation is [10]:

$$h_p = (0.17k / d_{co}) Re^{0.67} Pr^{0.33} (D / D_T)^{0.1} (\mu / \mu_w)^{-1.4} (d_{co} / D_T)^{0.5} \quad (14)$$

The value of heat transfer coefficient resulted from this equation is 5425 W/ (m2.k) for the present case. The value resulted from our CFD simulation is 3863.

Since all of the empirical equations are based on the presumption that the temperature is same all over the tank, they are likely to overestimate the real value of heat transfer coefficient.

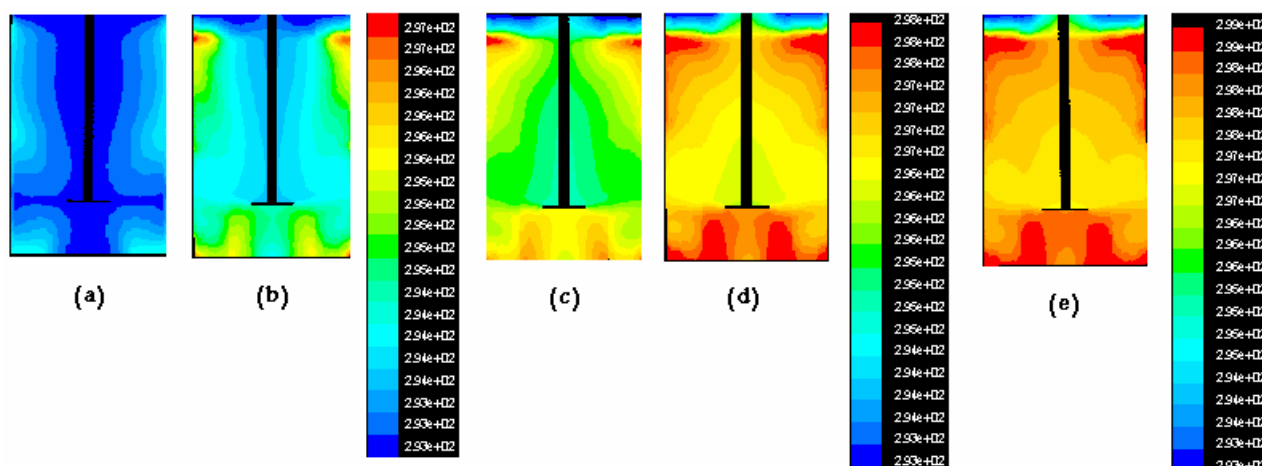


Fig. 2 Temperature contours in selected range for jacketed tank by passage of time: (a) 1.2s, (b) 2.7s, (c) 4.8s, (d) 6.3s, (e) 7.8s. (Temperatures higher than the maximum value shown in temperature range bar are shown with the highest amount)

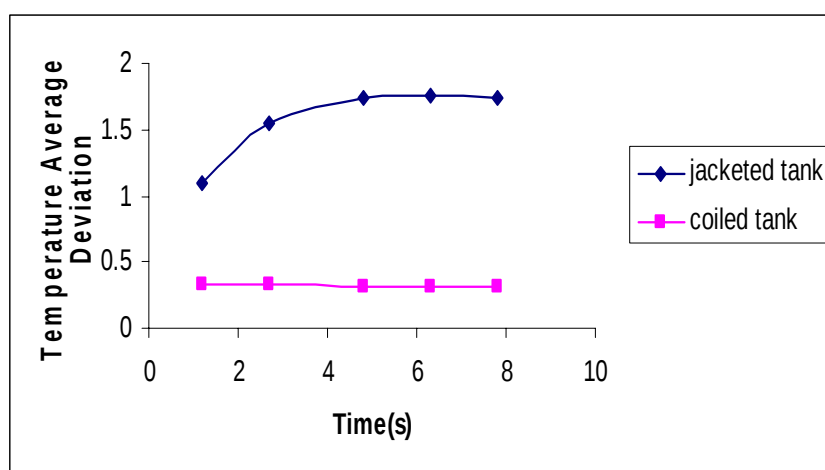


Fig. 3 Temperature Standard Deviation

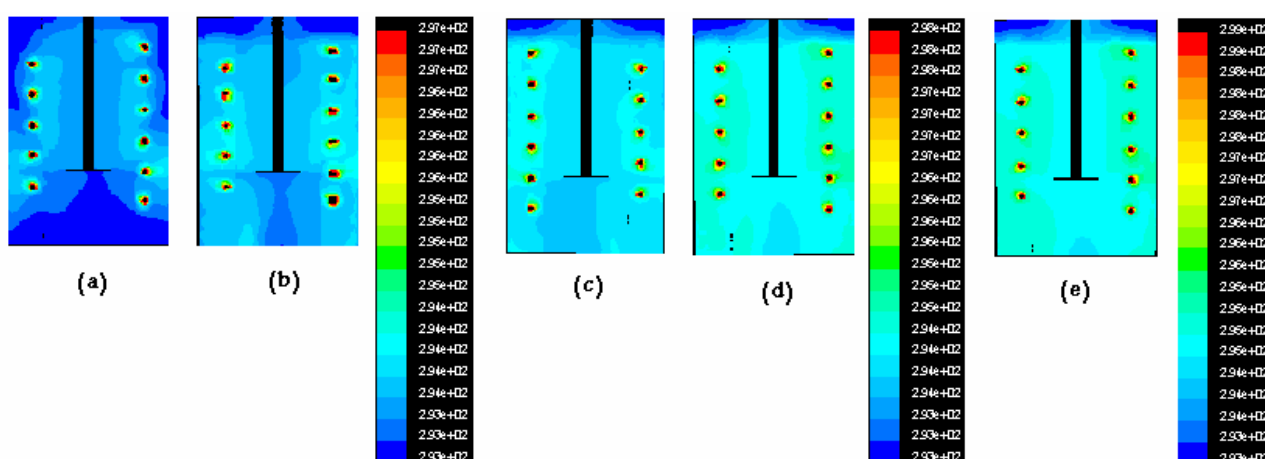


Fig. 4 Temperature contours in selected range for coiled tank by passage of time: (a) 1.2s, (b) 2.7s, (c) 4.8s, (d) 6.3s, (e) 7.8s. (Temperatures higher than the maximum value shown in temperature range bar are shown with the highest amount)

Another important factor that affects the temperature distribution is that jackets are placed at the walls and are not the same distant from all spots. Moreover, the shape of velocity vectors cause the same areas to be heated more and more and temperature average deviation grows by time.

As calculations show, although heat transfer coefficient and heat flux rate are greater for the coiled vessel, average temperature increase is higher for jacketed case. This is because of larger area of the jacket. The area of coil can be increased to improve this.

V. CONCLUSION

1. For the case that temperature control in the center of the tank is critical, coil is the better choice.
2. Coil leads to more uniform temperature distribution and a higher heat transfer coefficient.
3. In the coiled stirred tank, required power for impeller is higher than the jacketed case.
4. In the case of viscous fluids application of coil is rejected.

NOMENCLATURE

D	Impeller Diameter
D_T	Tank Diameter
E	Energy
$\vec{F}$	Force
g	Gravitational Acceleration
h_p	Process Heat Transfer Coefficient
k	Heat Conductivity Coefficient
K	Turbulence Kinetic Energy
P	Pressure
S_k	Source Term
t	Time
T	Temperature
u	Velocity
V	Velocity of Moving Mesh
Re	Reynolds Number
Pr	Prandtl number

Greek Letters

ρ	Density
μ	Viscosity
α	Volume Fraction
τ	Shear stress
η	Viscosity Correction Exponent
ε	Turbulence Dissipation Rate

REFERENCES

- [1] K.H. Javed, T. Mahmud, J.M. Zhu, Numerical simulation of turbulent batch mixing in a vessel agitated by a Rushton turbine, Chemical Engineering and Processing 45 (2006) 99–112.
- [2] S.M.C.Peixoto, J.R.Nunhez, Improving internal flow of coiled stirred tanks, second international conference on CFD in the Minerals and Process industries, CSIRO, Melbourne, 1999.
- [3] G.L.Lane, P.T.L.Koh, CFD Simulation of a Rushton Turbine in a baffled tank, International conf on CFD in Mineral & Metal processing, CSIRO 1997.
- [4] FLUENT user's Guide, 2003.
- [5] L.Rudniak, P.M. Machniewski, A.Milewska, E.Molga, CFD modeling of stirred tank chemical reactors: homogeneous and heterogeneous reaction systems, Chemical Engineering Science 59 (2004) 5233 – 5239.
- [6] A.Bakker, R.D.Laroche, M.Wang, R.V.Calabrese, Sliding Mesh Simulation of Laminar flow in stirred reactors, Trans.I.Chem.E. Vol.75, Part A., 1997.
- [7] J. Aubin, D .F. Fletcher, C.Xuereb, Modeling turbulent flow in stirred tanks with CFD, Experimental Thermal and Fluid Science 28 (2004) 431–445.
- [8] E.L.Paul, V.A.Atiemo, S.M.Kresta, Hand Book of Industrial Mixing, 2004.
- [9] Ranade, V.V & Dommeti, S.M.S, Computational snapshot of flow generated by axial impellers in baffled stirred vessels, Trans.I.Chem.E, Vol.74, Part A, 1996.
- [10] Holland, F.A, Chapman, F.S, Chem.Eng. Feb.15, 1965, P.175.
- [11] Oldshue, J.Y., and Gretton, A.T., Chem.Eng.Prog. Vol.50, 1954, p. 615.