ORIENTAL JOURNAL OF CHEMISTRY
An International Open Free Access, Peer Reviewed Research Journal
www.orientjchem.org

Est. 1984

ISSN: 0970-020 X
CODEN: OJCHEG
2012, Vol. 28, No. (1):
Pg. 51-58

Mathematical Modeling and Numerical Simulation of Wastewater Treatment Unit Using CFD

MAHBOOBEH PISHNAMAZI[1], AZAM MARJANI[2], SAEED SHIRAZIAN[2]* and MOHAMMAD SAMIPURGIRI[3]

[1]Petroleum Engineering Department, Iranian Central Oil Fields Company (ICOFC), Tehran (Iran).
[2]Islamic Azad University, Arak Branch, Department of Chemistry, Arak (Iran).
[3]Islamic Azad University, Tehran North Branch, Department of Chemical Engineering, Tehran (Iran).
*Corresponding author: E-mail: shirazian@iust.ac.ir

(Received: December 10, 2011; Accepted: January 25, 2012)

ABSTRACT

This paper presents mathematical modeling and numerical simulation of hydrodynamics of a mixer used for wastewater treatment. The simulation is based on solving conservation equations for mixer. Swirl flow is considered for flow behavior in the mixer. The geometry of mixer was divided in to subdomains including fixed and mobile parts. Impeller was considered as mixing device. The model equations are solved by using computational fluid dynamics (CFD) techniques. Velocity distributions were obtained through solving the governing equations. The simulations results showed that there are some dead zones which reduce the efficiency of the mixing process. The modeling findings also revealed that velocity of impeller is a key parameter for increasing mixing efficiency.

Key words: Mixer, Wastewater treatment, CFD, Simulation, Modeling.

INTRODUCTION

Nowadays, wastewater treatment is of vital importance in all industries. It can be defined as the process of removing contaminants from industrial and municipal wastewater. Wastewater treatment can be carried out using physical, chemical, and biological processes to remove physical, chemical and biological contaminants from water. Its objective is to produce an environmentally-safe fluid waste stream (or treated effluent) and a solid waste (or treated sludge) suitable for disposal or reuse. Using advanced technology it is now possible to re-use sewage effluent for drinking water.

Biological wastewater treatment is the most important and applicable process for wastewater treatment. It includes some processes and operations. Mixer is the most important equipment used in biological wastewater treatment. Mixers play crucial role in optimizing and improvement of whole treatment process. Modeling and simulation of mixers is the best method for designing and optimization of these devices. Modeling can be carried out by solving conservation equations including mass, momentum and energy equations. The main purpose of this work is to develop and solve a comprehensive mathematical model for simulation of fluid flow in a mixer. The governing equations are solved numerically using CFD based on finite element method (FEM). Velocity and pressure distributions are obtained and evaluated to find the optimum conditions [1-5].

Model equations

A 3D comprehensive mathematical model is developed for simulation of flow in a mixer of wastewater treatment unit. The model is built considering following assumptions:

1. Isothermal condition is assumed
2. Incompressible fluid inside the mixer
3. Newtonian fluid
4. Physical properties such as viscosity and density are assumed to be constant
5. Swirl flow is considered for flow behavior inside the mixer

The general equations describing flow behavior in the mixer are the Navier-Stokes equations and can be expressed as follows

where:

ρ viscosity of fluid (kg/m^3)

u velocity vector (m/s)

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho u) = 0 \quad ..(1)$$

$$\rho \frac{\partial u}{\partial t} + \rho(u.\nabla)u = \nabla.\left[-pI + \tau\right] + F \quad ..(2)$$

$$\rho C_p \left(\frac{\partial T}{\partial t} + \left(u.\nabla\right) T \right) = -\left(\nabla.q\right) + \tau : S - \frac{T}{\rho}\frac{\partial \rho}{\partial T}\left(\frac{\partial p}{\partial t} + (u.\nabla) p \right) + Q \quad ..(3)$$

p pressure (Pa)

τ tensor of viscosity stress (Pa)

F body force (N/m^3)

C_p heat capacity (J/(kg.K))

T temperature (K)

q vector of heat flux (W/m^2)

Q heat source (W/m^3)

These equations are applied for incompressible fluids and are general equations for modeling flow behavior. There are various types of these equations depending on the fluid and flow pattern. The main flow pattern in mixers is the Swirl Flow. The governing equations for swirl flow are derived from the Navier-Stokes equations. These equations for cylindrical coordinate may be expressed as follows:

The tensors of stress applied in these equations may be written as follows:

$$\frac{\partial u_r}{\partial r} + \frac{u_r}{r} + \frac{\partial u_z}{\partial z} = 0 \quad ..(4)$$

$$\rho\left(\frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + u_z \frac{\partial u_r}{\partial z} - \frac{u_\varphi^2}{r} \right) = -\frac{\partial p}{\partial r} + \frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rr}) + \frac{\partial \tau_{zr}}{\partial z} - \frac{\tau_{\varphi\varphi}}{r} + F_r \quad ..(5)$$

$$\rho\left(\frac{\partial u_z}{\partial t}+u_r\frac{\partial u_z}{\partial r}+u_z\frac{\partial u_z}{\partial z}\right)=-\frac{\partial p}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}(r\tau_{rz})+\frac{\partial \tau_{zz}}{\partial z}+F_z \quad ..(6)$$

$$\rho\left(\frac{\partial u_\varphi}{\partial t}+u_r\frac{\partial u_\varphi}{\partial r}+u_z\frac{\partial u_\varphi}{\partial z}+\frac{u_\varphi u_r}{r}\right)=\frac{1}{r^2}\frac{\partial}{\partial r}(r^2\tau_{r\varphi})+\frac{\partial \tau_{z\varphi}}{\partial z}+F_\varphi \quad ..(7)$$

Numerical simulation
Geometry of the mixer

$$\tau_{rr}=2\eta\frac{\partial u_r}{\partial r},\qquad \tau_{zr}=\tau_{rz}=\eta\left(\frac{\partial u_z}{\partial r}+\frac{\partial u_r}{\partial z}\right) \quad ..(8)$$

$$\tau_{zz}=2\eta\frac{\partial u_z}{\partial r},\qquad \tau_{r\varphi}=\tau_{\varphi r}=\eta r\frac{\partial}{\partial r}\left(\frac{u_\varphi}{r}\right) \quad ..(9)$$

$$\tau_{\varphi\varphi}=2\eta\frac{u_r}{\partial r},\qquad \tau_{z\varphi}=\tau_{\varphi z}=\eta\frac{\partial u_\varphi}{\partial z} \quad ..(10)$$

The first step for numerical solution of the governing equations is drawing the geometry of mixer. The mixer studied in this work is constituted of two parts, i.e. fixed and mobile parts. The fixed part is the wall of mixer and the mobile part is the impeller which makes the swirl flow. These two parts are shown in Fig. 1. As it can be seen, there are also four baffles in the fixed parts of the mixer. These baffles increase the efficiency of mixing process by removing dead zone parts.

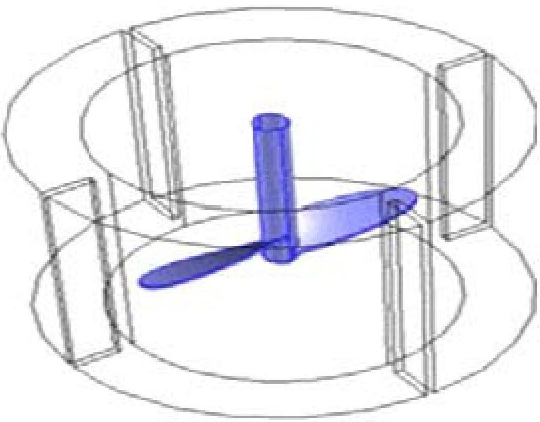

Fig. 1: Geometry of mixer

Meshing is also required to divide the model domain into small parts for numerical solution. Numerical solution of the governing equations is applied in each mesh. The meshed geometry is shown is Fig. 2. The meshes are created using Comsol Multiphysics 4.2 software.

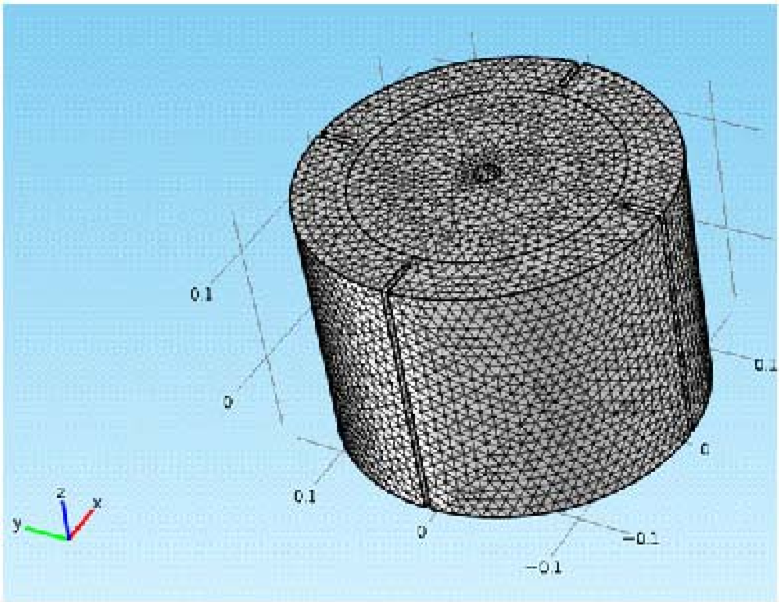

Fig. 2: Meshing of the model domain

Numerical solution

The main objective of the present study is to model a mixer using CFD of momentum transfer. The equations of mixer with appropriate boundary conditions were solved using COMSOL Multiphysics version 4.2 software (Sweden), which uses finite element method (FEM) for numerical solution of the equations. The finite element analysis is combined with adaptive meshing and error control using numerical solver of UMFPACK. This solver is well suited for solving stiff and non-stiff non-linear boundary value problems. The applicability, robustness and accuracy of this method have been proved through distinct researches [6-10]. It should be pointed out that the COMSOL mesh generator creates triangular meshes that are isotropic in size. A large number of elements are then created with scaling. A scaling factor was employed for the mixer. COMSOL automatically scales back the geometry after meshing. Adaptive mesh refinement in COMSOL, which generates the best and minimal meshes, was used to mesh the mixer geometry. The algorithm which was developed for the numerical simulation is shown in Fig. 3. An IBM-PC-Pentium 4 (CPU speed is 2800 MHz) was used to solve the sets of equations.

RESULTS AND DISCUSSION

Velocity distribution

The main parameter for simulation of mixer is the velocity of fluid inside the mixer. The objective of the current simulation was to determine the velocity vector of during mixing process in the mixer. The mixer is applied for wastewater treatment units. The 3D velocity distribution of fluid inside the mixer is shown in Fig. 4. The velocity in the mixer is shown at three different slices. Figs. 5 & 6 also indicate the velocity profile as curves. As it can be seen from the figures, the maximum of velocity is observed at the regions near the impeller. On the other hand, the velocity near the mixer's wall is zero which implies that no-slip condition is imposed at this boundary.

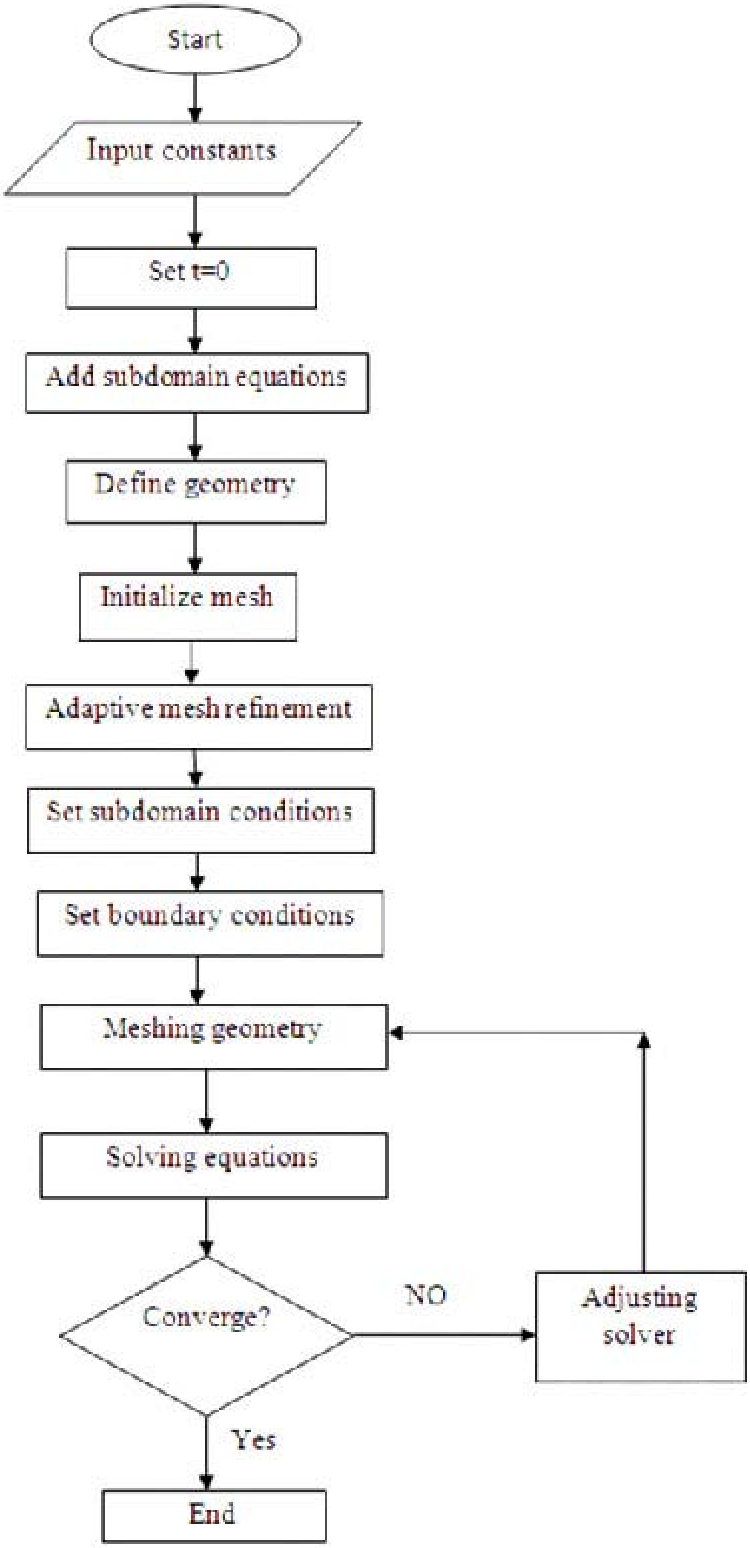

Fig. 3: Algorithm developed for numerical simulation

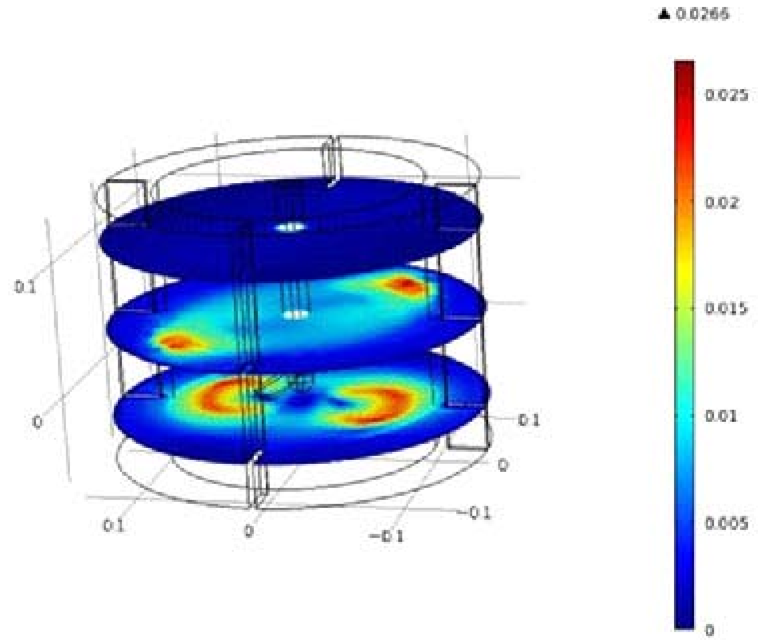

Fig. 4: Slices of velocity distribution in the mixer

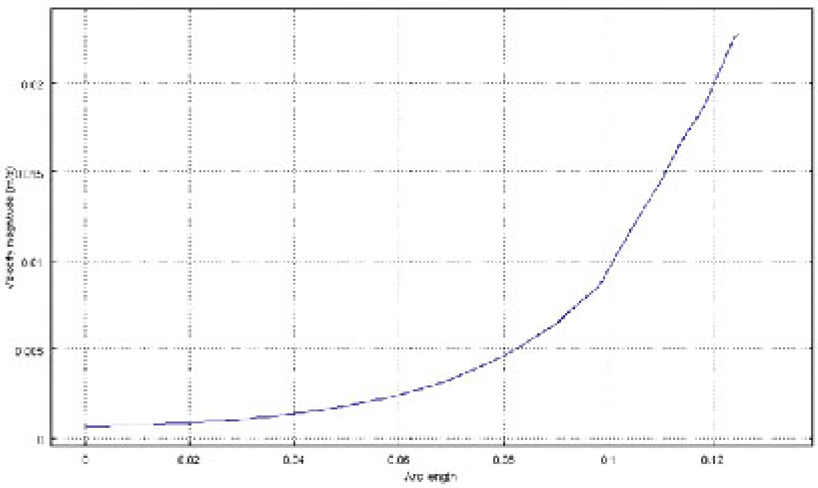

Fig. 5: Liner velocity profile in the mixer

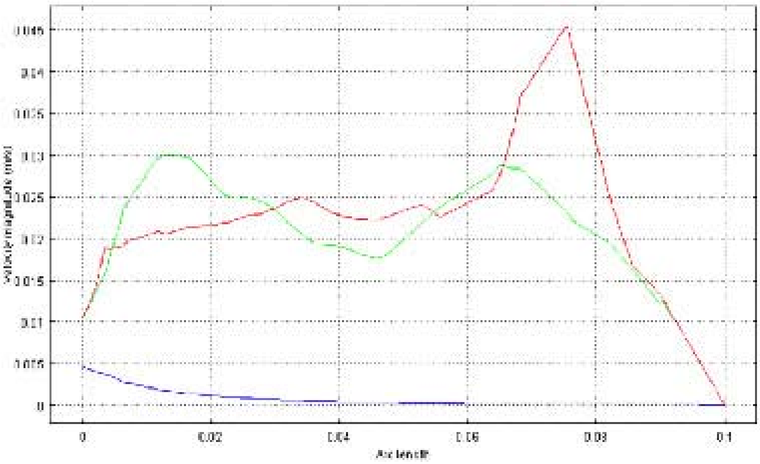

Fig. 6: Profile of velocity in the mixer at different times

CONCLUSIONS

A 3D mathematical model was developed in this work to simulate the flow in a mixer of wastewater treatment unit. The model solves equations of swirl flow. Finite element method (FEM) was applied for numerical solution of the swirl flow equations. Comsol Multiphysics software was used for numerical simulation. Velocity distributions were obtained and indicated that there are some dead zones near the wall of mixer which can be removed by adjusting the impeller velocity. The impeller can also change the flow pattern. The results of this work confirmed that the developed model is capable of simulation of flow in mixers for different geometries.

REFERENCES

1. Shirazian, S., A. Moghadassi, and S. Moradi, Numerical simulation of mass transfer in gas-liquid hollow fiber membrane contactors for laminar flow conditions. *Simulation Modelling Practice and Theory,* **17**(4): p. 708-718 (2009).
2. Shirazian, S. and S.N. Ashrafizadeh, Mass Transfer Simulation of Caffeine Extraction by Subcritical CO_2 in a Hollow-Fiber Membrane Contactor. *Solvent Extraction and Ion Exchange*, **28**(2): p. 267-286 (2010).
3. Shirazian, S. and S.N. Ashrafizadeh, Mass Transfer Simulation of Carbon Dioxide Absorption in a Hollow-Fiber Membrane Contactor. *Separation Science and Technology,* **45**(4): p. 515-524 (2010).
4. Shirazian, S. and S.N. Ashrafizadeh, Near-Critical Extraction of the Fermentation Products by Membrane Contactors: A Mass Transfer Simulation. *Industrial & Engineering Chemistry Research,* **50**(4): p. 2245-2253 (2011).
5. Farzad Fadaei, Saeed Shirazian, Seyed Nezameddin Ashrafizadeh, Mass transfer simulation of solvent extraction in hollow-fiber membrane contactors , *Desalination*, **275**(1-3): Pages 126-132 (2011).
6. Saeed Shirazian, Azam Marjani, Farzad Fadaei, Supercritical extraction of organic solutes from aqueous solutions by means of membrane contactors: CFD simulation, *Desalination,* 277(1-3): Pages 135-140 (2011).
7. Azam Marjani, Saeed Shirazian, Simulation of heavy metal extraction in membrane contactors using computational fluid dynamics, *Desalination*, **281**: Pages 422-428 (2011).
8. Saeed Shirazian, Azam Marjani, Fatemeh Azizmohammadi, Prediction of SO_2 Transport

Across Ceramic Membranes Using Finite Element Method (FEM), *Oriental journal of Chemistry* **27**(2): Page No. 485-490 (2011).

9. Brucato, A., Ciofalo, M., Grisafi, F., and Micale, G., Numerical prediction of flow fields in baffled stirred vessels: a comparison of alternative modelling approaches, *Chem. Eng. Sci.,* . **53**: 3653-3684 (1998).

10. Mavros, P., Flow visualization in stirred vessels-a review of experimental techniques, *Trans. Inst. Chem. Eng. Part A Chem. Eng. Res. Des.,* **79**: pp. 113-127 (2001).