A New Approach for Modeling of a Fluidized Bed by CFD-DEM

Shayan Karimipour  
University of Tehran

Navid Mostoufi  
University of Tehran, mostoufi@ut.ac.ir

Rahmat Sotudeh-Gharebagh  
University of Tehran, sotudeh@ut.ac.ir

H. Chizari  
Department of Mechanical Engineering, University of Tehran

Follow this and additional works at: http://dc.engconfintl.org/cfb10

Part of the Chemical Engineering Commons

Recommended Citation

This Conference Proceeding is brought to you for free and open access by the Refereed Proceedings at ECI Digital Archives. It has been accepted for inclusion in 10th International Conference on Circulating Fluidized Beds and Fluidization Technology - CFB-10 by an authorized administrator of ECI Digital Archives. For more information, please contact franco@bepress.com.
A NEW APPROACH FOR MODELING OF A FLUIDIZED BED
BY CFD-DEM

S. Karimi¹, H. Chizari², N. Mostoufi*, R. Sotudeh-Gharebagh¹
¹Process Design and Simulation Research Center, Department of Chemical
Engineering, University of Tehran, P.O. Box 11155/4563, Tehran, Iran
²Department of Mechanical Engineering, University of Tehran, Tehran, Iran
* Corresponding author, Tel.: (+98-21)6696-7797, Fax: (+98-21)6646-1024,
E-mail: mostoufi@ut.ac.ir

ABSTRACT
Numerical studies of 3D cylindrical fluidized bed by means of combined
computational fluid dynamics (CFD) and discrete element method (DEM) were
carried out. For motion of particles, Newton’s second law and 3D compressible
Navier-Stokes equations in generalized curvilinear coordinates in its conservative
form were used. Navier-Stokes equations were solved with high order compact finite
difference scheme by fully implicit flux decomposition method. Non-reflecting
boundary conditions (NRBC) were used for the outflow boundary. The convergence
of this method, especially at high Reynolds number, is significantly better than the
SIMPLE method.

INTRODUCTION
Gas-solid fluidized beds have been widely utilized as reactors in the chemical and
petrochemical industries. Successful design and operation of fluidized bed reactors
requires proper prediction of the performance of reactor. Whereas experiment
studies in these systems are quite tedious and expensive, recently modeling of
fluidized bed reactors has been extensively used to study these systems.
Mathematical models for modeling fluidized bed reactors can be grouped into two
main categories: Eulerian-Eulerian (EE) and Lagrangian-Eulerian (LE). In the EE
model, both particles and gas phase are considered as a continuum phase (1, 2, 3, 4).
Recently, many researchers (5, 6) have adopted the LE model, which is also
called discrete element method (DEM), for modeling the phenomena in particle-fluid
systems. Using this model, trajectories of individual particles can be traced by
solving Newton’s second law for each particle while the flow of gas, which is treated
as a continuum phase, is described by Navier-Stokes equation.

Finite volume method is usually used to solve the Navier-Stokes equation. The
SIMPLE method with staggered grid is suitable for solving Navier-Stokes equation.
However, using this method of solution for problems with high Reynolds number is
typically unstable. Furthermore, implementing the high order SIMPLE method is so
difficult in comparison with finite difference schemes. One of the finite difference
methods that can be used for high Reynolds flow is the flux decomposition method.
In this method, convective fluxes are decomposed based on eigenvalues which
results in a better agreement with the physical properties of the fluid, especially at
high Reynolds numbers.
Although the LE approach is more appropriate to study the hydrodynamics of fluidized beds. This model requires large computational resources for large scale systems (\( \frac{L}{D} > 4.4Re^{1/6} \) must be chosen to prevent wave reflection). In recent years, non-reflecting boundary conditions (NRBC) were used by some researchers (7, 8) to mitigate wave reflection problems. The concept of NRBC was proposed by Thompson (9) where the idea of specifying the boundary conditions according to the inward and outward propagating waves was introduced. Thompson showing that wave reflections, and therefore, can be used as a fictitious boundary. By using this boundary condition, computational domain and computational time for solving Navier-Stokes equation is decreased. The amount of time saving depends on the gas inlet velocity. As the gas inlet velocity increases the free domain which is required for particles motion increase and therefore the effect of NRBS reduces. But using NRBS almost half the computational time decreases.

In the present work, a CFD-DEM technique was used to investigate the hydrodynamics of a 3D cylindrical fluidized bed. Newton's second law and 3D compressible Navier-Stokes equation in generalized curvilinear coordinates in conservative form was solved for particle and gas phase respectively. In spite of previous studies that usually used SIMPLE method for solving Navier-Stokes equations, these equations were solved with high order compact finite difference scheme by fully implicit flux decomposition method. Using curvilinear coordinates, the physical domain was changed from cylindrical to semi-Cartesian coordinates in computational domain. Furthermore, NRBC was used for outflow boundary to reduce computation time. Hydrodynamics of the bed was investigated and the results were in good agreement with the expected behavior of gas and solids in the fluidized bed.

**Governing equations**

**Equation of motion**

In the present work, the flow of sphere particles in a 3D cylindrical fluidized bed was investigated. Newton's second law was applied to each particle. The translational and rotational motion of the particles can be described by following equations (10):

\[
m_{i} = \frac{dV_{i}}{dt} = f_{f,i} + \sum_{j=1}^{k_{i}} (f_{c,ij} + f_{d,ij}) + f_{g,i}
\]

\[
l_{i} \frac{d\omega_{i}}{dt} = \sum_{j=1}^{k_{i}} T_{i,j}
\]

Soft sphere method (11) was used for simulation Inter-particle and particle-wall contact forces.

For the gas phase, three-dimensional compressible Navier-Stokes equations in generalized curvilinear coordinates \((\xi, \eta, \zeta)\) were written in conservative form:

\[
\frac{\partial \epsilon Q}{\partial t} + \frac{\partial (E - E_{p})}{\partial \xi} + \frac{\partial (F - F_{p})}{\partial \eta} + \frac{\partial (G - G_{p})}{\partial \zeta} = P
\]
For solving Navier-Stokes equations with flux decomposition method, first the Jacobian matrices must be evaluated:

\[ A_{ij} = \frac{\partial E_i}{\partial Q_j}, \quad B_{ij} = \frac{\partial F_i}{\partial Q_j}, \quad C_{ij} = \frac{\partial G_i}{\partial Q_j} \]
By considering $Q_{n+1} = Q_p + \delta Q_p$ Navier-Stokes equations are changed to:
\[
\epsilon_{n+1} \left( \frac{\partial Q}{\partial t} \right) + Q_{n+1} \left( \frac{\partial \epsilon}{\partial t} \right) + \frac{\partial A}{\partial \xi} \delta Q_p + \frac{\partial B}{\partial \eta} \delta Q_p + \frac{\partial C}{\partial \zeta} \delta Q_p = p^n + \left( \frac{\partial E_v}{\partial \xi} + \frac{\partial F_v}{\partial \eta} + \frac{\partial G_v}{\partial \zeta} \right) n - \left( \frac{\partial E}{\partial \xi} + \frac{\partial F}{\partial \eta} + \frac{\partial G}{\partial \zeta} \right) n
\]  
(12)

The Jacobean matrices are decomposed using eigenvalues ($\lambda$):
\[
A = A^+ + A^- \quad A^\pm = \frac{1}{2} (A \pm \lambda I)
\]  
(13)

Finally, flow field obtain by solving following equation:
\[
\left\{ I + \frac{2\Delta t}{6\epsilon_{n+1} - 4\epsilon^n + \epsilon^{n-1}} \left( \frac{\partial A}{\partial \xi} + \frac{\partial B}{\partial \eta} + \frac{\partial C}{\partial \zeta} \right) \right\} \delta Q_p = R_{imp}
\]  
(14)

\[
R_{imp} = -\frac{\epsilon_{n+1} (3Q^p - 4Q^n + Q^{n-1}) + Q^p (3\epsilon_{n+1} - 4\epsilon^n + \epsilon^{n-1})}{2\Delta t} - \frac{\epsilon^n (\partial (E - E_v) + \partial (F - F_v) + \partial (G - G_v))}{2\Delta t} - \frac{\epsilon^n (\partial (E - E_v) + \partial (F - F_v) + \partial (G - G_v))}{2\Delta t} + \frac{\epsilon^n (\partial (E - E_v) + \partial (F - F_v) + \partial (G - G_v))}{2\Delta t} + \frac{\epsilon^n (\partial (E - E_v) + \partial (F - F_v) + \partial (G - G_v))}{2\Delta t} + \frac{\epsilon^n (\partial (E - E_v) + \partial (F - F_v) + \partial (G - G_v))}{2\Delta t} + \frac{\epsilon^n (\partial (E - E_v) + \partial (F - F_v) + \partial (G - G_v))}{2\Delta t}
\]  
(15)

NON-REFLECTING BOUNDARY CONDITIONS

If $Q$ is the conservative variable, $q$ is the primitive variable and $M$ is the conversion matrix as below:
\[
M = \frac{\partial Q}{\partial q} = \begin{pmatrix}
1 & 0 & 0 & 0 & 0 \\
u & \rho & 0 & 0 & 0 \\
v & 0 & \rho & 0 & 0 \\
w & 0 & 0 & \rho & 0 \\
\frac{1}{2}(u^2 + v^2 + w^2) & \rho u & \rho v & \rho w & \frac{1}{\gamma - 1}
\end{pmatrix}, \quad q = \begin{pmatrix}
\rho \\
u \\
v \\
w \\
\rho
\end{pmatrix}
\]  
(16)

NRBC in the $\xi$-direction in conservative form can be written as
\[
\frac{\partial Q}{\partial t} + D_i + M_{\beta} \frac{\partial q}{\partial \eta} + M_{\beta} \frac{\partial q}{\partial \zeta} = JR_v
\]  
(17)

where
\[
\bar{v} = \begin{pmatrix}
V & \rho \eta_x & \rho \eta_y & \rho \eta_z & 0 \\
0 & V & 0 & 0 & \eta_x \\
0 & 0 & V & 0 & \eta_y \\
0 & 0 & 0 & V & \eta_z \\
0 & \gamma P \eta_x & \gamma P \eta_y & \gamma P \eta_z & V
\end{pmatrix}, \quad \bar{c} = \begin{pmatrix}
W & \rho \xi_x & \rho \xi_y & \rho \xi_z & 0 \\
0 & W & 0 & 0 & \xi_x \\
0 & 0 & W & 0 & \xi_y \\
0 & 0 & 0 & W & \xi_z \\
0 & \gamma P \xi_x & \gamma P \xi_y & \gamma P \xi_z & W
\end{pmatrix}
\]  
(18)
\[ D = Md = \begin{pmatrix} D_1 \\ D_2 \\ D_3 \\ D_4 \\ D_5 \end{pmatrix} = \begin{pmatrix} d_1 \\ ud_1 + \rho d_2 \\ v d_1 + \rho d_3 \\ w d_1 + \rho d_4 \\ \frac{1}{2} (u^2 + v^2 + w^2) d_1 + \rho u d_2 + \rho v d_3 + \rho w d_4 + \frac{d_5}{y - 1} \end{pmatrix} \] (19)

\[ d = PL = \begin{pmatrix} d_1 \\ d_2 \\ d_3 \\ d_4 \\ d_5 \end{pmatrix} = \begin{pmatrix} \xi_x L_1 + \xi_y L_2 + \xi_z L_3 + \frac{\rho}{\sqrt{2c}} (L_4 + L_5) \\ -\xi_x L_2 + \xi_y L_3 + \frac{\xi_x}{\sqrt{2}} (L_4 - L_5) \\ \xi_x L_1 - \xi_x L_3 + \frac{\xi_y}{\sqrt{2}} (L_4 - L_5) \\ -\xi_y L_1 + \xi_y L_2 + \frac{\xi_x}{\sqrt{2}} (L_4 - L_5) \\ \frac{\rho c}{\sqrt{2}} (L_4 + L_5) \end{pmatrix} \] (20)

\[ L = \begin{pmatrix} L_1 \\ L_2 \\ L_3 \\ L_4 \\ L_5 \end{pmatrix} = \begin{pmatrix} U \left( \xi_x \frac{\partial \rho}{\partial \xi} + \xi_z \frac{\partial \rho}{\partial \xi} - \xi_y \frac{\partial v}{\partial \xi} \right) \\ U \left( \xi_y \frac{\partial \rho}{\partial \xi} - \xi_x \frac{\partial u}{\partial \xi} + \xi_z \frac{\partial v}{\partial \xi} \right) \\ U \left( \xi_x \frac{\partial \rho}{\partial \xi} + \xi_y \frac{\partial u}{\partial \xi} - \xi_z \frac{\partial v}{\partial \xi} \right) \\ (U + C) \left( \frac{\xi_x}{\sqrt{2}} \frac{\partial u}{\partial \xi} + \frac{\xi_y}{\sqrt{2}} \frac{\partial v}{\partial \xi} + \frac{\xi_z}{\sqrt{2}} \frac{\partial w}{\partial \xi} + \frac{1}{\sqrt{2c}} \frac{\partial \rho}{\partial \xi} \right) \\ (U - C) \left( -\frac{\xi_x}{\sqrt{2}} \frac{\partial u}{\partial \xi} - \frac{\xi_y}{\sqrt{2}} \frac{\partial v}{\partial \xi} - \frac{\xi_z}{\sqrt{2}} \frac{\partial w}{\partial \xi} + \frac{1}{\sqrt{2c}} \frac{\partial \rho}{\partial \xi} \right) \end{pmatrix} \] (21)

**RESULTS AND DISCUSSION**

In this numerical study, a cylindrical fluidized bed with 7.5 cm diameter and 50 cm height, filled with 10000 particles was considered. The diameter and density of particles were 2 mm and 1100 kg/m³, respectively. A jet of gas was introduced into the bed via a central hole with jet velocity of 8 m/s (\(u \sim 5u_{mf}\)).

Fig. 1 shows snapshots of particles positions in the bed. As can be seen in this figure, particles that are at the center of the bed accelerate very fast and move vertically in the bed. The initial acceleration of the particles is so high that they travel almost 50 cm in the bed. This high acceleration is due to the high vertical drag force exerted on particles due to high gas velocity in the central area of the bed. However, particles in the annulus of this area do not move very noticeably in the vertical direction. This is mainly due to the low gas velocity in this region. The gas velocity in this region is low for two reasons. First, the gas is not introduced into the bed from near the walls. Second, wall effects induce a velocity boundary layer in which the gas velocity is low near the wall.
Over time by particles movement from distributor, the void fraction in that region increase and the gas velocity decrees. Void fraction increasing in distributor center is more than the distributor corner. So reduction in the gas velocity in the distributor center holes is more than distributor corners. The gas velocity affects particles motion that caused to occurring two peaks in particles motion. If the simulation has been permitted to continue for a longer time, the transient state of the bed would have been vanished and a better solid flow pattern would have been obtained. Since the time required for the calculation is very high (around 1 month), this results has not been obtained yet.

Fig. 2 shows velocity vector and density contour of the gas in the bed. The bed is divided into four sections that are perpendicular to each other to able to show the behavior of the flow field in 3D cylindrical bed. However, this figure shows only two sections of the bed.

For low Mach number flows, changes in density are low, but as it can be seen in these figures, with upcoming the particles the density of fluid before them increases. Moreover, the velocity vectors in center of the cylinder are high. This flow pattern is in accordance with a gas flow pattern in a cylinder with boundary layer. According to the continuity relation, while the gas velocity increases, the density of gas decreases. This result shows that the presented model and the implemented method can predict compressibility behavior of the gas very well.
CONCLUSIONS
A combination of CFD and DEM was used to study the behavior of the high velocity fluidized bed reactor. The conservative form of Navier-Stocks equations in curvilinear coordinate was derived and used to obtain flow field of the gas in the bed. As described before, flux decomposition method is more compatible with high Reynolds numbers and high order solving methods than SIMPLE method; therefore in this work flux decomposition method was used for modeling high Reynolds fluidized bed. It must be emphasized that solving high order Navier-Stokes equations caused to seeing the physical behavior of the bed much better.

NOTATION

d    amount of wave for each primitive Navier-Stokes equation
D    amount of wave for each conservative Navier-Stokes equations
E    inviscid flux vector ($\xi$ direction)
E_v  viscous flux vector ($\zeta$ direction)
f_c  contact force, N
f_d  Damping force, N
f_f  particle-fluid interaction force, N
f_g  acceleration force, N
F    inviscid flux vector ($\eta$ direction)
F_v  viscous flux vector (η direction)
G  inviscid flux vector (ζ direction)
G_v viscous flux vector (ζ direction)
I  moment of inertia, kg m^2
J  Jacobian of the coordinate transformation,
L  amount of each characteristic wave
M  conversion matrix
q  vector of primitive variables
Q  vector of conservative variables
U  contra-variant velocity(x direction), m s^{-1}
V  contra-variant velocity(y direction), m s^{-1}
W  contra-variant velocity(z direction), m s^{-1}

REFERENCES