Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

USING CFD TO MODEL THE INTERACTION OF A HORIZONTAL FEED JET ON FLUIDIZED BED HYDRODYNAMICS

Jonathan Tyler and Philip Mees

Department of Chemical and Materials Engineering, University of Alberta, Edmonton, Alberta, CANADA
philip.mees@ualberta.ca

ABSTRACT

Fluidized beds are used in the North Albertan petroleum industry for processing the rich oil sands reserves. These very large fluidized bed cokers represent a major capital investment and modifications are costly. Advanced design tools and predictive methods such as CFD are desired to test case designs before implementation. At the University of Alberta, we are using CFX 4.2 from AEA Technologies to model the interaction of a horizontal gas feed jet with the fluidized bed hydrodynamics. Preliminary results of gas jet penetration into the bed compare favorably with correlations by Merry (1971). Model validation is being conducted against a smaller experimental fluidized bed to study the effect of discretization scheme (lower order schemes versus TVD schemes), grid refinement and model assumptions. Simulation results will be used to predict the optimal feed nozzle placement within the industrial fluidized beds. With this study we hope to be able to make quantitative comparisons between simulation results and experimental data.

Keywords: *fluidized bed, discretization schemes, horizontal feed jet, simulation*

NOMENCLATURE

BH initial bed height (m)
D_O equivalent nozzle diameter (m)
E compaction modulus
$\overline{F}_\alpha$ inter-phase non-drag forces
I nozzle insertion into reactor (m)
S distance of nozzle to distributor (m)
U_f fluidization gas velocity (m/s)
U_{jet} horizontal gas jet velocity (m/s)
XT reactor width (m)
YT reactor height (m)
ZT reactor depth (m)

$c_{\alpha\beta}$ inter-phase drag term
d_p particle diameter (m)
g gas phase
k multi-particle drag correction term (Gidaspow)
p pressure
s solids phase
$\overline{U}$ velocity vector (m/s)
$\overline{B}$ body force

α,β phase (gas or solid)
ε_α volume fraction phase α
ε^* compaction gas volume fraction
μ viscosity of phase α
ρ density (kg/m^3)

INTRODUCTION

Computational fluid dynamics (CFD) is gaining more favor as a design and testing tool for industrial applications. Research is being carried out to validate multiphase flow model capabilities with CFX 4.2 against experimental tests. Two of the world's largest fluidized bed reactors are located in Northern Alberta in Ft. McMurray. Each of these units represents a great deal of capital investment requiring high annual expenditures for maintenance and operation. The possible construction of an additional reactor is an expensive undertaking and as such requires the best available tools for optimal design.

The aim of our research at the University of Alberta is to use the two-fluid multiphase models available in CFX 4.2 from AEA Technology and to validate it against a companion two dimensional experimental fluidized bed at the University of Saskatchewan (UofS), Saskatoon, Canada. Results from these simulations and experiments are being used to learn about the strengths and weaknesses of current state of the art multiphase modeling with commercial packages.

Qualitative comparisons have been made between high speed videography of the experimental unit and simulation results. Performance factors such as gross overall bed movement, nozzle jet penetration depth, bubble formation and bed surface characteristics can be compared directly. Digital image analysis techniques are being investigated with the computer simulations to relieve some of the tedium of analyzing time-step by time-step output from the overall flow field.

MODEL DESCRIPTION

Model equations solved with CFX 4.2 are based on a finite volume approach to the integral equations of mass and momentum for the two phases (CFX 4.2 Solver Manual, 1997). Gas/solid phases are modeled using the two-fluid model with an equation of momentum for each of the phases. The flow is assumed to be laminar, incompressible and isothermal.

Governing Equations:

$$\frac{\partial}{\partial t}(\varepsilon_\alpha \rho_\alpha) + \nabla \cdot (\varepsilon_\alpha \rho_\alpha \overline{U}_\alpha) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}\left(\varepsilon_\alpha \rho_\alpha \overline{U}_\alpha\right)+\nabla\cdot\left(\varepsilon_\alpha\left(\rho_\alpha \overline{U}_\alpha \otimes \overline{U}_\alpha - \mu_\alpha\left(\nabla \overline{U}_\alpha + \left(\nabla \overline{U}_\alpha\right)^T\right)\right)\right)$$
$$= \varepsilon_\alpha\left(\overline{B}-\nabla p_\alpha\right)+\sum_{\beta=1}^{N_P} c_{\alpha\beta}{}^{(d)}\left(\overline{U}_\beta - \overline{U}_\alpha\right)+\overline{F}_\alpha \qquad (2)$$

Constitutive Relationships:

for $\varepsilon_g<0.8$

$$c_{\alpha\beta}=150\frac{(1-\varepsilon)^2}{\varepsilon\left(d_p\phi_s\right)^2}+1.75\frac{\rho_g\left|U_g-U_S\right|(1-\varepsilon)}{\phi_S d_p} \qquad (3)$$

for $\varepsilon_g>0.8$:

$$c_{\alpha\beta}{}^{(d)}=\frac{3}{4}\frac{C_D}{d}\varepsilon_\beta\rho_\alpha\left|U_\beta-U_\alpha\right| \qquad (4)$$

$$C_D=\frac{24}{\mathrm{Re}} \qquad 0\le \mathrm{Re}\le 0.2 \qquad (5)$$

$$C_D=\frac{24}{\mathrm{Re}}\left(1+0.15\,\mathrm{Re}^{0.687}\right) \quad 0\le \mathrm{Re}\le 500\text{-}1000$$

(Schiller and Nauman)

$$(6)$$

$$C_D=0.44 \qquad 500\text{-}1000\le \mathrm{Re}\le 1\text{-}2\mathrm{x}10^5 \qquad (7)$$

$$\mathrm{Re}=\frac{d_p\left|U_s-U_g\right|\rho_g}{\mu_g} \qquad (8)$$

Solids Pressure:

$$-\nabla P_S=-e^{E(\varepsilon_c-\varepsilon^*)}\nabla\varepsilon_g \qquad (9)$$

Boundary and Initial Conditions

The geometry is shown in Figure 1 and has dimensions of 1.2 x 1.2 x 0.1 m. All simulations use particles with a diameter of 370µm, ρ_p=1450kg/m³ and fluidization air with ρ_g=1.2kg/m³. Inlet boundary conditions (BC) were set at the nozzle tip and distributor along the low XZ plane with a gas normal velocity. A pressure BC was used for the high XZ plane with zero solids carry over. All walls are assumed to have gas no-slip and solids slip conditions. The bed was initially filled to a height of BH=0.71m and the gas phase was set to have a vertical velocity component equal to that of minimum fluidization. Simulations did not assume symmetry in the XY center plane.

An Algebraic Multigrid solver was used with Spalding's IPSA algorithm (Lo 1989) linking the phase momentum equations. A time step of 0.001 seconds was used.

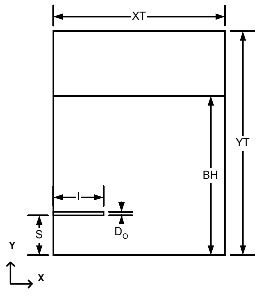


Figure 1: Schematic diagram of geometry.

RESULTS

Preliminary Investigations

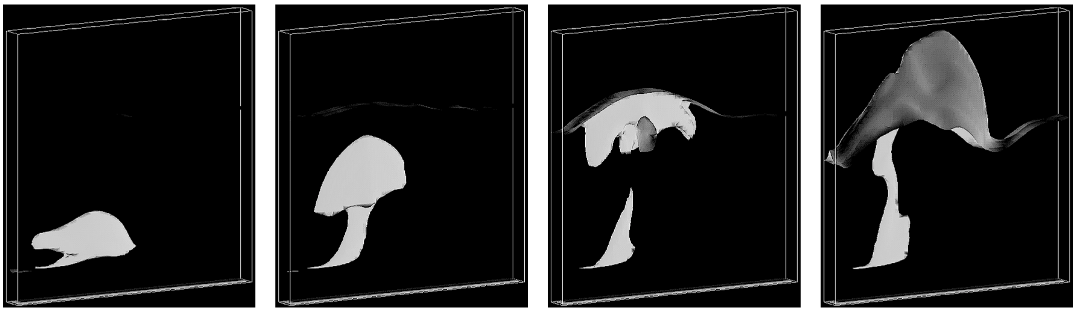
Two dimensional simulations of fast bubbles rising in a fluidized bed produced good agreement with studies by Bouillard et al. (1991) for the flow patterns, shape and size of the bubbles. However, the two dimensional assumption used for CFD should not be generalized for inherently three dimensional phenomena such as a horizontal gas feed jet. Simulations using a two dimensional mesh for the horizontal feed jet resulted in the defluidization of the bed above the jet plume because none of the fluidization gas was able to traverse around the plume to fluidize the top part of the bed. Although experimental setups such as the one at the UofS are generally considered to be two dimensional, CFD simulations require in this case a three dimensional domain. Experimental setups which are "two dimensional" might actually be referred to as being "pseudo-two dimensional" because of the possibility for fluidization gas to leak up either side of the gas jet.

Three different discretization schemes were investigated to see what impact they have on the simulation results for the horizontal gas feed jet (Table 1). All spatial terms are discretized using second order central differencing; advection terms were discretized using the following different schemes. In CFX the schemes were chosen as follows: Hybrid scheme which is first order accurate and overly diffusive, Minmod TVD scheme which is somewhat diffusive and Superbee (SB) TVD scheme which is overly compressive. Selected gas volume fraction iso-surface plots are shown in Figure 2 for the first second of simulation with U_{jet} = 250 m/s. The Hybrid scheme results in the formation of a large "mushroom" of gas being blown into the bed. Long fingers form from the jet plume and connect the jet plume to the surface of the bed. The Minmod scheme results in an initial mushroom pattern which soon breaks apart into a series of large bubbles which then move up the bed. Notice that the jet plume still forms a fairly large void within the bed. SB discretization results in the formation of a large number of bubbles within the fluidized bed and a more distinct jet plume issuing from the nozzle. Based upon discussions with our colleagues at the UofS, we decided to continue simulations using the SB discretization scheme. Being overcompressive, it tends to sharpen the boundaries between the gas and solid phases.

CFD Run	Scheme	U_f	D_o	S	I
U_{jet}=150 m/s					
AA01	Hybrid	U_{mf}	0.0099	0.145	0.1175
AA02	Minmod	U_{mf}	0.0099	0.145	0.1175
AA03	SB	U_{mf}	0.0099	0.145	0.1175
U_{jet}=250 m/s					
AB01	Hybrid	U_{mf}	0.0099	0.145	0.1175
AB02	Minmod	U_{mf}	0.0099	0.145	0.1175
AB03	SB	U_{mf}	0.0099	0.145	0.1175
U_{jet}=300 m/s					
AC01	Hybrid	U_{mf}	0.0099	0.145	0.1175
AC02	Minmod	U_{mf}	0.0099	0.145	0.1175
AC03	SB	U_{mf}	0.0099	0.145	0.1175

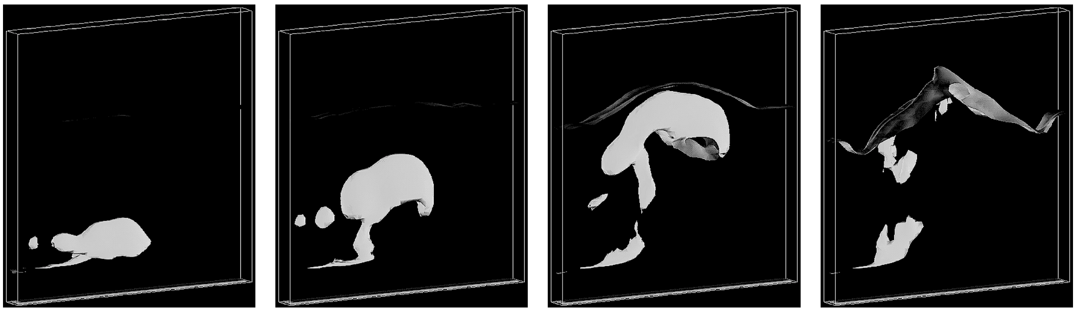
Table 1: Modeling conditions for comparison of discretization schemes.

a) Hybrid discretization scheme; run AB01



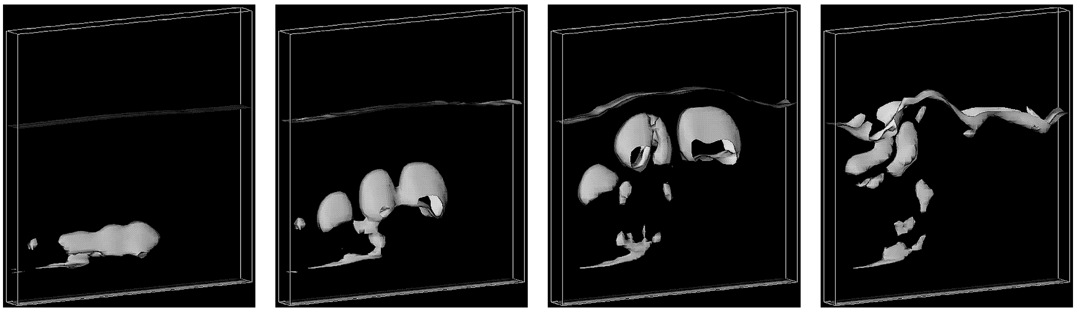
i) ii) iii) iv)

b) Minmod discretization scheme; run AB02



i) ii) iii) iv)

c) Superbee discretization scheme; run AB03



i) ii) iii) iv)

Figure 2: Qualitative comparison of different discretization schemes for a fluidized bed with horizontal gas feed jet located on the left hand side. Light coloured surfaces represent a gas volume fraction iso-surface with ε_g=0.8 approximating bubble and jet plume boundaries. U_{jet}=250 m/s. Real time: i) 0.25, ii) 0.50, iii) 0.75 and iv) 1.00 seconds.

Grid Densities

Current simulations have been based on a mesh with approximately 30,000 nodes. Ongoing investigations are addressing how the grid density at the nozzle inlet BC affects jet penetration, jet plume formation and bubble creation. A single 1x1 inlet BC specification for the nozzle is insufficient to satisfy BC specifications. For example, flow variables are stored at the centroid of the control voume (CV) and the required flux values at the CV faces are then interpolated. The single face of a CV (i.e. 1x1 inlet BC) will not be interpolated to the correct values for specifying the desired nozzle inlet flow rate. As such, it is necessary to increase the number of nodes defining the nozzle orifice BC. Investigations are underway to determine how many nodes are sufficient to satisfy BC requirements.

Grids were constructed using a fortran routine which used a geometric mesh size progression above and below the nozzle orifice. This was necessary to ensure that a smooth mesh size progression was realized over the entire computational domain in the Y direction. One problem with creating a grid for this simulation was ensuring that the grid was fine enough around the nozzle orifice without being overly fine outside of the nozzle region. As pointed out by Hong *et al.* (1997) decreases in the nozzle orifice size require a much finer grid which can lead to a significant increase in computational time. Grid spacing in the Z direction was based upon the nozzle orifice Z dimension and, where possible, a geometric grid spacing was used. Steps were taken to ensure that the grid spacing did not create control volumes that were too small to be applied to a two fluid continuum model, please see Fan and Zhu (1998).

CFD Run	Scheme	U_f	D_o			
U_{jet}=250 m/s			(m)	Orifice BC Nodes	Grid Type	Max. Nodes
BA01	SB	U_{mf}	0.0099	1x1	Coarse	10,000
BA02	SB	U_{mf}	0.0099	1x1	Med	60,480
BA03	SB	U_{mf}	0.0099	1x1	Fine	83,160
BA04	SB	U_{mf}	0.0099	2x2	Med	71,280
BA05	SB	U_{mf}	0.0099	2x2	Fine	98,010
BA06	SB	U_{mf}	0.0099	3x3	Med	86,670
BA07	SB	U_{mf}	0.0099	3x3	Fine	116,640

Table 2: Nozzle inlet BC node studies. I = 0.1175m, S = 0.145m.

Comparison with Experimental Results

The two dimensional bed at the University of Saskatchewan was used to gain insight into the physical operation of the fluidized bed. Figure 3 is a schematic of this apparatus. The bed measures 1.2m x 1.2m x 0.1m with an initial bed height of 0.71m. Front and back walls are made from transparent Perspex glass. Particles were coloured red, black and white so that flow patterns could be followed. Two cyclones were used to separate solids from the carryover and returned through the dip leg to the bed. Compressed air was split three ways: two feeds for the distributor and one feed for the jet air. The unique split distributor allows experimenters to change the fluidization gas flow on either side of the fluidized bed to set up a circulating bed. The bed is filled with polyethylene particles with D_p=370µm and ρ_p=950 kg/m³.

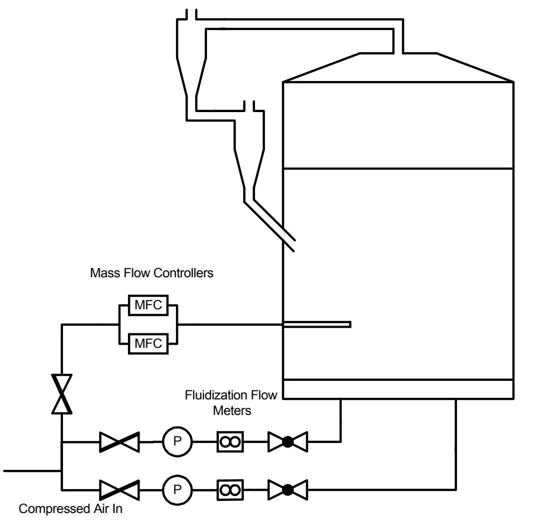


Figure 3: Experimental set up.

Measurements taken on the fluidized bed included high speed video pictures at 500 frames per second to capture pictures of bubble shape for comparison against the CFD work and measurements of jet penetration into the bed.

High speed video pictures of the fluidized bed show good qualitative agreement with selected frames from simulations using the Superbee discretization scheme under similar operating conditions.

One of the main problems with attempting to validate multiphase CFD simulations against experimental work is the difficulty to obtain a meaningful comparison. It appears that flow patterns must be compared qualitatively since it is very difficult to extract exact flow data from within a multiphase experiment.

Figures 4 and 5 show some results from the experimental setup and the CFD simulations respectively. Figure 4 shows the extent of the gas jet as it impacts on the side wall and forms bubbles which then move up through the fluidized bed. CFD simulations show the formation of the jet plume as it appears in a plane cutting directly through the fluidized bed. It has been observed from the high speed video that the end of the jet tends to "carve" the bed above the jet plume and form a void until the bed collapses back in upon itself. Behavior like this leads to the formation of bubbles at both the tip of the jet and slightly back from the maximum penetration depth, also exhibited by the CFD predictions.

Figure 4: Still photograph of fluidized bed with U_{jet}=327m/s and U_f=U_{mf}. Uprights represent approximately 0.3m separation. D_o=0.0036m.

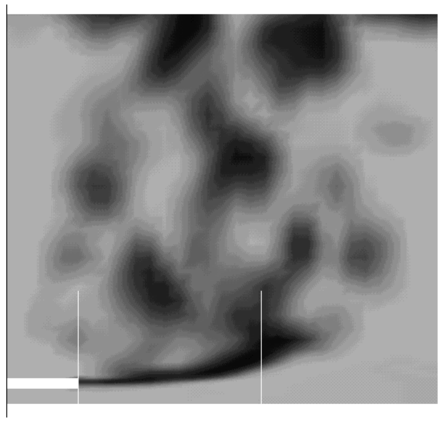


Figure 5: Volume fraction plot from CFD simulation with U_{jet}=300m/s and U_f=U_{mf}. Uprights represent 0.3m separation. Black areas represent higher gas volume fraction.

Digital Image Analysis

One of the problems facing CFD simulations of multiphase flow is the interpretation of the produced data. Single runs to produce 1 second of real time simulation results can have tens of thousands of time steps. It is not feasible to look at each individual time step to trace bubble sizes, bubble velocities, jet penetration results, etc… To remedy this situation we have been looking at using digital image analysis techniques (Jähne 1995) whereby the volume fraction data from a simulation is output as a bitmap file. This file is then input into MatLab for image processing. The 256 colour bitmap is filtered at a specific threshold value to produce a binary image; thresholding at ε_g=0.8 produces a binary image of bubbles and jet plumes. An eroded image is then subtracted from the binary image to produce the edges, or boundaries, outlining the phase boundaries. Once the edge information has been located, an edge tracing algorithm is used to isolate, count and classify individual bubbles within the computational domain. Our current efforts have been focussing on the counting and classification of bubble boundaries within a two dimensional plane.

Figure 6 shows CFD results for the gas volume fraction displayed on an XY plane. The data are thresholded and an edge enhancement operator is applied to extract boundary data for the bubbles and top of the fluidized bed.

Binary image data is an attractive option because it allows morphological data from the simulations to be saved in much less space than holding the entire flow domain data. Edge tracing routines can further reduce the storage space by saving only the data required to trace the outlines of the shapes.

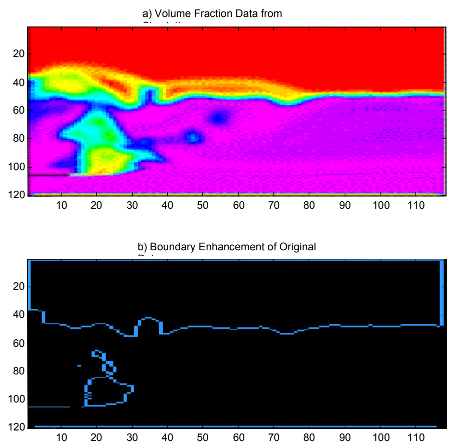


Figure 6: Example of use of image analysis tools to reduce data from volume fraction to surface boundary. a) Volume fraction data from simulation BA04. b) Boundary extraction at ε_g=0.8. Axes numbers represent the matrix indices for each pixel.

CONCLUSION

Based upon our preliminary research with CFD it appears that the higher order Superbee TVD scheme predicts horizontal gas jet and bubble behavior similar to experiments. The over compressive nature helps to sharpen phase boundaries resulting in a greater number of bubbles being predicted to form from the jet than either the Hybrid or Minmod schemes.

Horizontal gas jets should be modeled using a fully three dimensional computational domain. This avoids the problems associated with defluidizing the bed above the nozzle jet plume.

Digital image analysis can help reduce the amount of time required for manually examining CFD results and will be able to report qualitative data. These data could include bubble shape factors (sphericity, perimeters, areas and centroids), bubble counts and propagation rates. Binary image and edge tracing data helps reduce the amount of information required for storage.

Quantitative comparison is now needed to establish CFD codes available for development. Detailed initial and startup conditions used for experimental work need to be reported for use in simulations.

ACKNOWLEDGEMENTS

This work would not have been possible without the support of the Natural Sciences and Engineering Research Council (NSERC) of Canada. A special thanks to the UofA computer staff for their help. Experimental results were provided by Dr. Berutti and Jason Copan from the University of Saskatchewan, Saskatoon, Canada.

REFERENCES

BOUILLARD, J.X., GIDASPOW, D., and LYCZKOWSKI, R.W., (1991) "Hydrodynamics of fluidization: fast bubble simulation in a two-dimensional fluidized bed," *Powder Technology,* **66**, 107-118.

CFX-4.2: Solver Manual: (1997), AEA Technology, 342-345.

FAN, L.-S., and ZHU, C. (1998) "Principles of Gas-Solid Flows," *Cambridge University Press*. Cambridge, UK.

GIDASPOW, D., (1994) "Multiphase Flow and Fluidization: Continuum and Kinetic Theory Descriptions," *Academic Press*, 37, equation 2.16.

HONG, R., LI, H., LI, H,. and WANG, Y. (1997) "Studies on the Inclined Jet Penetration Length in a Gas-Solid Fluidized Bed," *Powder Technology*. **92**, 202-212.

JÄHNE, B., (1995), "Digital Image Processing: Concepts, Algorithms, and Scientific Applications", *Springer-Verlag*, New York.

LO, S.M., (1989), "Mathematical basis of a multi-phase flow model", *United Kingdom Atomic Energy Authority HARWELL,* AERE R 13432

MERRY, J.M.D., (1971), "Penetration of a Horizontal Gas Jet Into a Fluidised Bed", *Trans. Instn Chem. Engrs* **49**, 189-195.