
Tutorial: Modeling Uniform Fluidization in a Two Dimensional Fluidized Bed

Purpose

The purpose of this tutorial is to study the bubble formation and the hydrodynamics of the bed over long times. It also demonstrates how to customize a drag law for granular gas-solid flow. The default drag law in FLUENT is the Syamlal-O'Brien drag law. This law works for a large variety of problems, but has to be tuned properly for predicting the minimum fluidization conditions accurately.

Prerequisites

This tutorial assumes that you are familiar with the FLUENT interface, and have a good understanding of basic setup and solution procedures. This tutorial will not cover the mechanics of using the models; instead, it will focus on the application of the models. If you have not used these models before, the FLUENT Tutorial Guide will provide you with the necessary experience.

Problem Description

The prediction of pressure drop in uniformly fluidized bed is a problem of long standing interest in the process industry. The Eulerian models in FLUENT provide an important modeling tool for studying dense phase particulate flow involving complex inter-phase momentum transfer.

Despite rigorous mathematical modeling of the associated physics, the drag laws used in the model continue to be semi-empirical in nature. Therefore, it is crucial to use a drag law that correctly predicts the incipient or minimum fluidization conditions where the bed of particles is essentially in a state of suspension as a result of the balance between interfacial drag and body forces. The default Syamlal-O'Brien will be as follows:

The fluid-solid exchange coefficient is

$$K_{sl} = \frac{3\alpha_s \alpha_l \rho_l}{4v_{r,s}^2 d_s} C_D \left(\frac{Re_s}{v_{r,s}} \right) |\vec{v}_s - \vec{v}_l|$$

where $v_{r,s}^2$ is the terminal velocity coefficient for the solid phase.

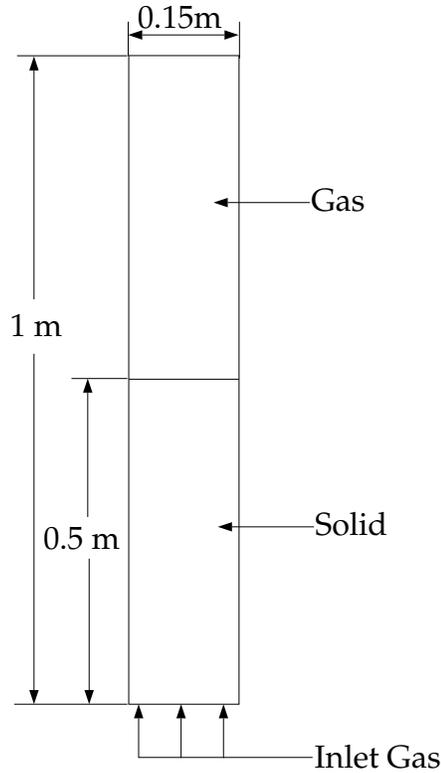


Figure 1: Problem Specification

$$v_{r,s} = 0.5 \left(A - 0.06Re_s + \sqrt{(0.06Re_s)^2 + 0.12Re_s(2B - A) + A^2} \right)$$

with $A = \alpha_l^{4.14}$ and $B = 0.8\alpha_l^{1.28}$ for $\alpha_l \leq 0.85$ and with $B = \alpha_l^{2.65}$ for $\alpha_l > 0.85$

The default constants of 0.8 and 2.65 predict a minimum fluidization of 21 cm/s. The experimentally observed minimum fluidization for this particular case is 8 cm/s. Therefore, by changing the above constants we can tune the drag law to predict minimum fluidization at 8 cm/s. After some mathematical manipulation, these constants come out to be 0.281632 and 9.07696 respectively. Therefore these values have to be used to predict the correct bed behavior and are passed to the code through User Defined Functions.

The problem to be considered is shown schematically below. It shows a (1m x 0.15m) fluidized bed. The inlet air comes in at 0.25 m/s and the top is modeled as a pressure outlet. The bed is packed with granular solids at 0.55 volume fraction (close to packing).

Setup and Solution

Step 1: Grid

1. Start the 2ddp version of FLUENT.
2. Read in the grid file bp.msh.
3. Check and display the grid.

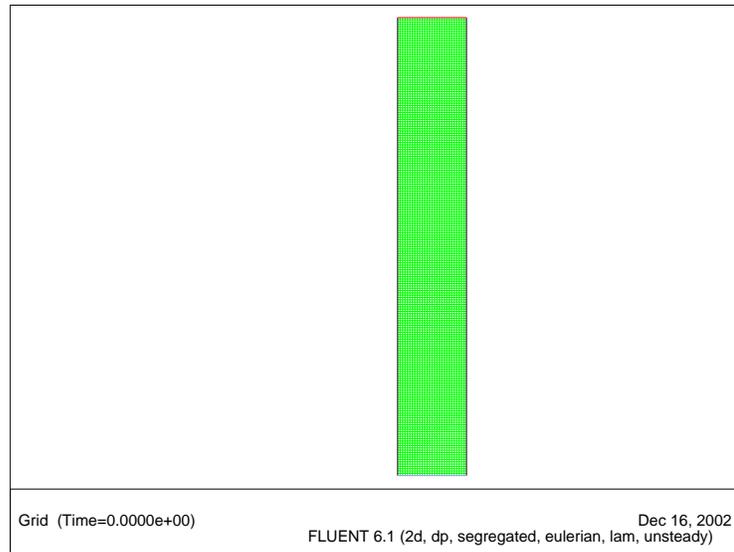


Figure 2: Graphics Display of the Grid

Step 2: Models

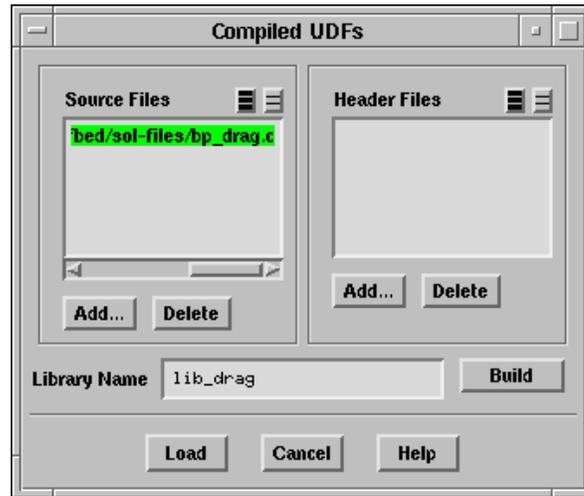
1. Define Segregated solver with 2D space and Unsteady time condition.
2. Define the Eulerian multiphase model.

Step 3: Materials

1. Modify the properties for air. Set the values for Density and Viscosity to 1.2 and $1.8e^{-05}$ respectively.
2. Define a material called solids. Set the values for Density and Viscosity to 2600 and $1.7894 e^{-5}$ respectively.

Step 4: Compile the UDF

The UDF contains two arguments `s_col` and `f_col`. These refer to the indices of the phases appearing in the second and first columns of the table in the interaction panel respectively. Therefore in this case `s_col` refers to the index of gas phase which is 0 and `f_col` refers to the index for solids which is equal to 1.



1. Compile the UDF using the Compiled UDFs panel.
2. Add the C function (`bp_drag.c`) under Source Files.
3. Enter `lib_drag` as the Library Name.
4. Click Build to build the UDF library.
5. Click Load to load the UDF and close the panel.

Step 5: Phases

Define → Phases...

1. Set the phase material of Primary Phase (Phase 1) to air. Rename it as gas.
2. Similarly set the Secondary Phase material to Solids and rename it to solid.
 - (a) Turn on Granular.
 - (b) Set the value of Diameter to 0.0003 m, and select `syamlal-obrien` from the Granular Viscosity pull-down list
 - (c) Retain the default values for the other parameters.

Check the column numbers in the Interaction panel under which the two phases appear. In this case solid and gas appear in the first and second columns respectively. These columns are used to specify the phase indices in the argument list for the UDF.

3. Set the drag coefficient.
 - (a) Select gas and click on Interaction.... The Phase Interaction panel opens.
 - (b) Under Drag Coefficient, select user-defined and select custom_drag_syam.

Step 6: Operating Conditions

1. Turn on Gravity and under Gravitational Acceleration, enter value of Y as -9.81 .
2. Turn on Specified Operating Density, and set the value of Operating Density to 1.2.

Step 7: Boundary Condition

1. Set the boundary conditions for vintlet zone.
 - (a) In the boundary conditions panel, under Phase select gas and enter a value of 0.25 for Y-Velocity in the velocity inlet panel. You will first need to select Components as the Velocity Specification Method.
 - (b) For solid phase, set the value of Volume Fraction to 0.

Step 8: Solution

1. Mark a region for adaption
 - (a) Enter minimum value of 0 for X and Y coordinates
 - (b) Enter maximum values of 0.15 and 0.5 for X and Y coordinates respectively.
 - (c) Click Mark to mark the cells for refinement.
2. Initialize the flow with default values.
3. Patch the solids volume fraction for hexahedron-r0.
 - (a) Select solid Volume Fraction for Variable and set the Value to 0.55.

If you wish to patch a constant value, enter that value in the Value field. If you want to patch a previously-defined field function, turn on the Use Field Function option and select the appropriate function in the Field Function list.
 - (b) Click on Patch to update flow field data.

Note: Click Adapt to perform the refinement immediately.

4. Set the parameters that control the solution.
 - (a) Retain the default selected equations (all of them).
 - (b) Under Under-Relaxation Factors, set the values of Pressure to 0.5, Momentum to 0.2 and Volume Fraction to 0.4.
 - (c) Retain the values for other parameters.
5. Enable autosaving of the data files for every 100 time steps.
6. Start the calculation by requesting 1400 iterations and set the Time Step Size to 0.001 sec.

Step 9: Commands for Animations

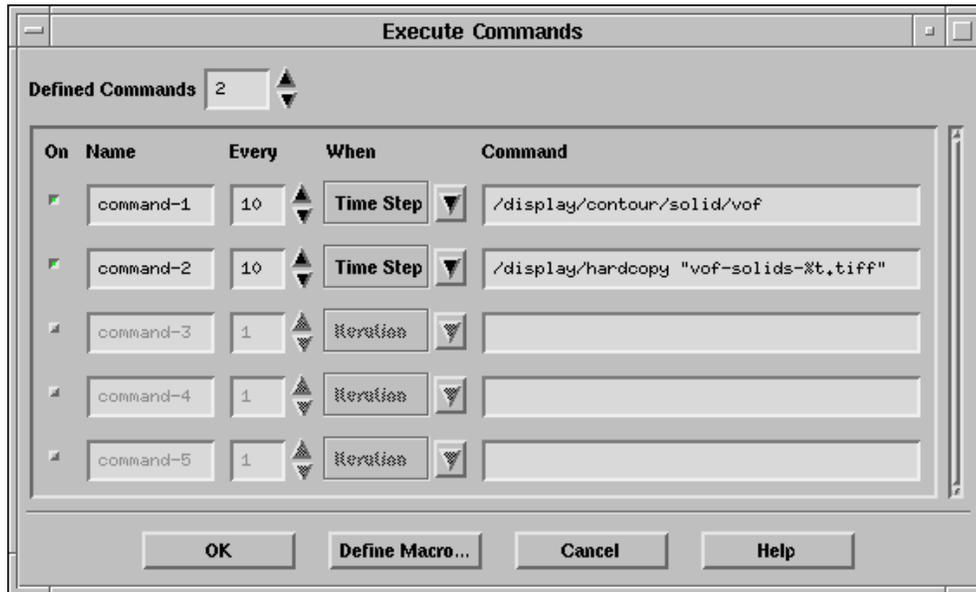
If required, set up commands to write out tiff files for animation.

1. Define the commands for animation.

Solve → Execute Commands...

The Execute Commands panel is displayed.

- (a) Define the commands as shown in the following panel.



2. In the Graphics Hardcopy panel, set the format to TIFF, Coloring to Color, disable the Reverse Foreground/Background option and click Apply.

Step 10: Postprocessing

1. Display contours of volume fraction of solid ($t = 0.2$ sec).

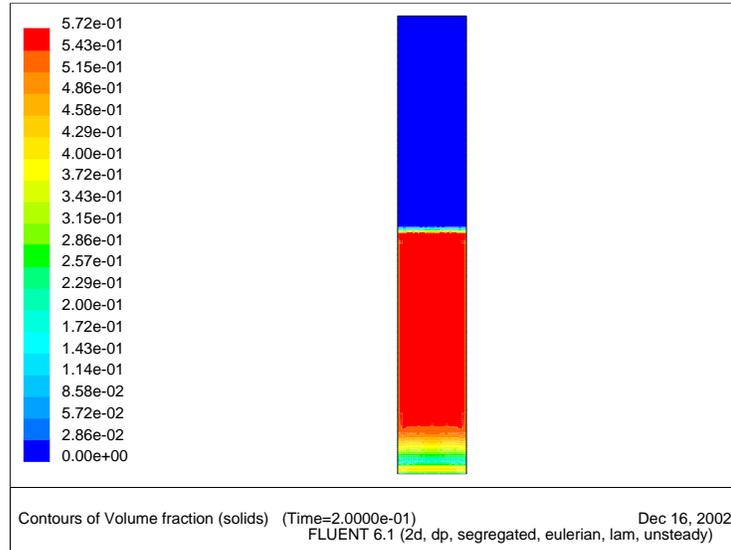


Figure 3: Contours of Volume Fraction of solid

2. Display contours of volume fraction of solid ($t = 0.9$ sec).

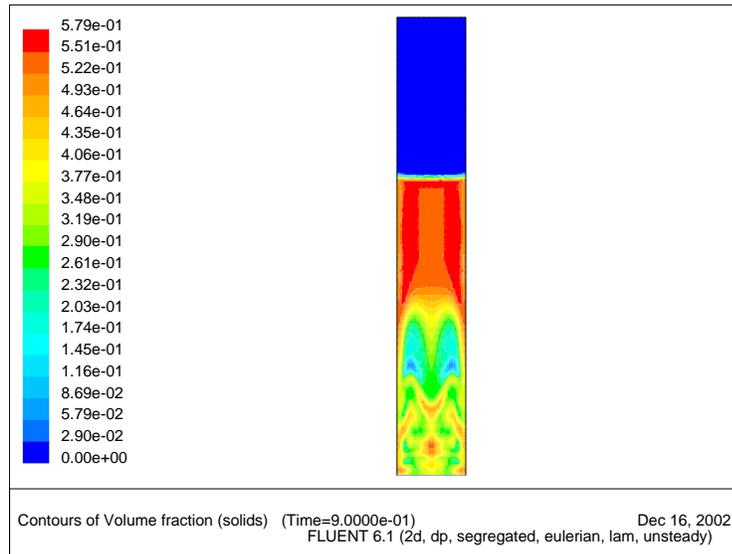


Figure 4: Contours of Volume Fraction of solid

Results

Typically, the constants set to 0.8 and 2.65 in the default drag law have to be modified to balance the interfacial drag with the weight of the bed at minimum fluidization. Otherwise, the correct bubbling pattern is severely under-predicted leading to incorrect prediction of pressure drop which is what most important objective of such simulations.