
Tutorial: Modeling Uniform Fluidization in 2D Fluidized Bed

Introduction

The prediction of pressure drop in an uniformly fluidized bed is a problem of long standing interest in the process industry. The Eulerian models in ANSYS 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 purpose of this tutorial is to study the hydrodynamics and bubble formation in a fluidized bed over a period of time. It also demonstrates how to customize a drag law for granular gas-solid flow.

This tutorial demonstrates how to do the following:

- Customize a drag law for granular gas-solid flow.
- Use the Eulerian models to predict the pressure drop in an uniformly fluidized bed.
- Solve the case using appropriate solver settings.
- Postprocess the resulting data.

Prerequisites

This tutorial is written with the assumption that you have completed [Tutorial 1](#) from the ANSYS FLUENT 12.0 Tutorial Guide, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

This tutorial will not cover the mechanics of using the Eulerian models. It will focus on the application of these models. For more information refer to Section [24.5 Setting Up the Eulerian Model](#) in the ANSYS FLUENT User's Guide. For information about user-defined functions (UDF) refer to the [ANSYS FLUENT UDF Manual](#).

Problem Description

The default drag law in ANSYS 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.

The default Syamlal-O'Brien is as follows: The fluid-solid exchange coefficient is

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

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

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

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 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 considered is a 1m x 0.15m fluidized bed as shown in Figure 1. The inlet air enters 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).

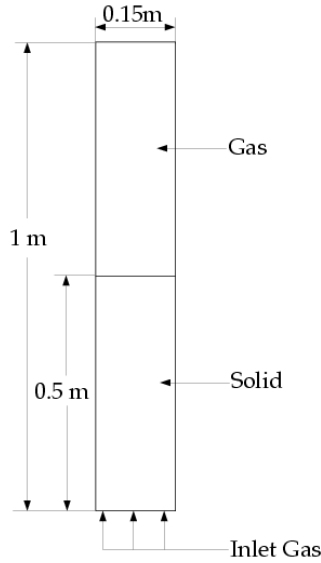


Figure 1: Problem Specification

Preparation

1. Copy the files, `bp.msh.gz` and `bp_drag.c` to the working folder.
2. Use FLUENT Launcher to start the 2D version of ANSYS FLUENT.

For more information about FLUENT Launcher see [Section 1.1.2 Starting ANSYS FLUENT Using FLUENT Launcher](#) in the ANSYS FLUENT 12.0 User's Guide.

3. Enable Double-Precision in the Options list.
4. Click the UDF Compiler tab and make sure that Setup Compilation Environment for UDF is enabled.

The path to the .bat file which is required to compile the UDF will be displayed as soon as you enable Setup Compilation Environment for UDF.

If the UDF Compiler tab does not appear in the FLUENT Launcher dialog box by default, click the Show Additional Options>> button to view the additional settings.

Note: *The Display Options are enabled by default. Therefore, after you read in the mesh, it will be displayed in the embedded graphics window.*

Setup and Solution

Note: *All entries in setting up this case are in SI units, unless otherwise specified.*

Step 1: Mesh

1. Read the mesh file `bp.msh.gz`.

File → Read → Mesh...

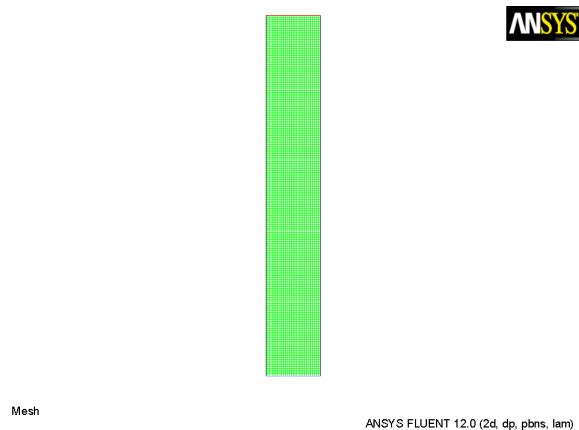


Figure 2: Graphics Display of the Mesh

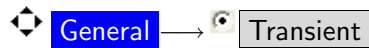
Step 2: General

1. Check the mesh.



ANSYS FLUENT will perform various checks on the mesh and will report the progress in the console. Ensure that the minimum volume reported is a positive number.

2. Enable the transient solver by selecting Transient from the Time list.



Step 3: Models

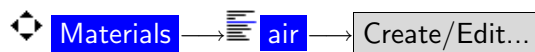
1. Select the Eulerian multiphase model.



- (a) Select Eulerian from the Model selection list.
- (b) Retain the default settings and close the Multiphase Model dialog box.

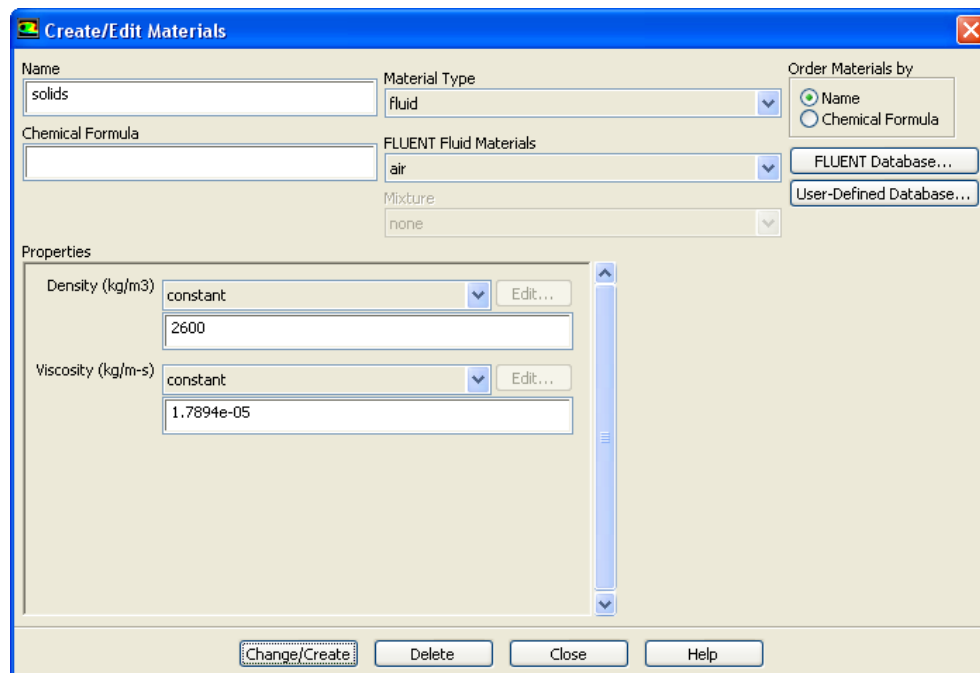
Step 4: Materials

1. Modify the properties for air.



- (a) Enter 1.2 kg/m³ for Density.
- (b) Enter 1.8e-05 kg/m-s for Viscosity.
- (c) Click Change/Create.

2. Define a material called solids.



- (a) Enter `solids` for Name.
- (b) Enter 2600 kg/m³ for Density and 1.7894e-05 kg/m-s for Viscosity.
- (c) Click Change/Create.

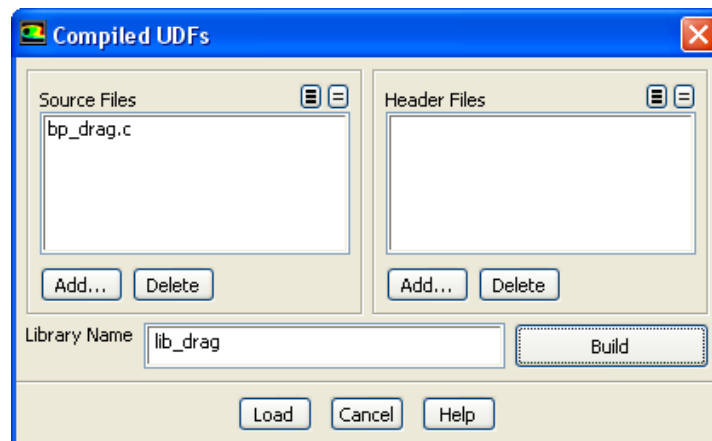
A Question dialog box will appear asking if you want to overwrite `air`. Click No.

3. Close the Create/Edit Materials dialog box.

Step 5: 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 dialog box 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.

Define → User-Defined → Functions → Compiled...



1. Click the Add... button in the Source Files section to open the Select File dialog.
2. Select the file `bp_drag.c`.
3. Enter `lib_drag` for Library Name.
4. Click Build.

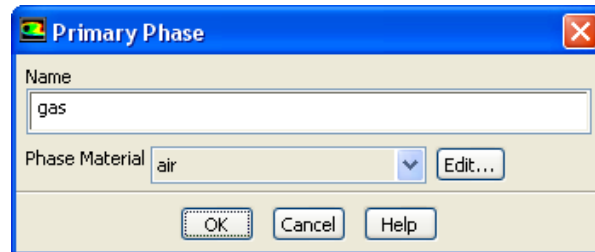
A Warning dialog box will appear, warning you to make sure that the UDF source files are in the same folder that contains the case and data files. Click OK to close the Warning dialog box.

You can view the compilation history in the log file that is saved in your working folder.

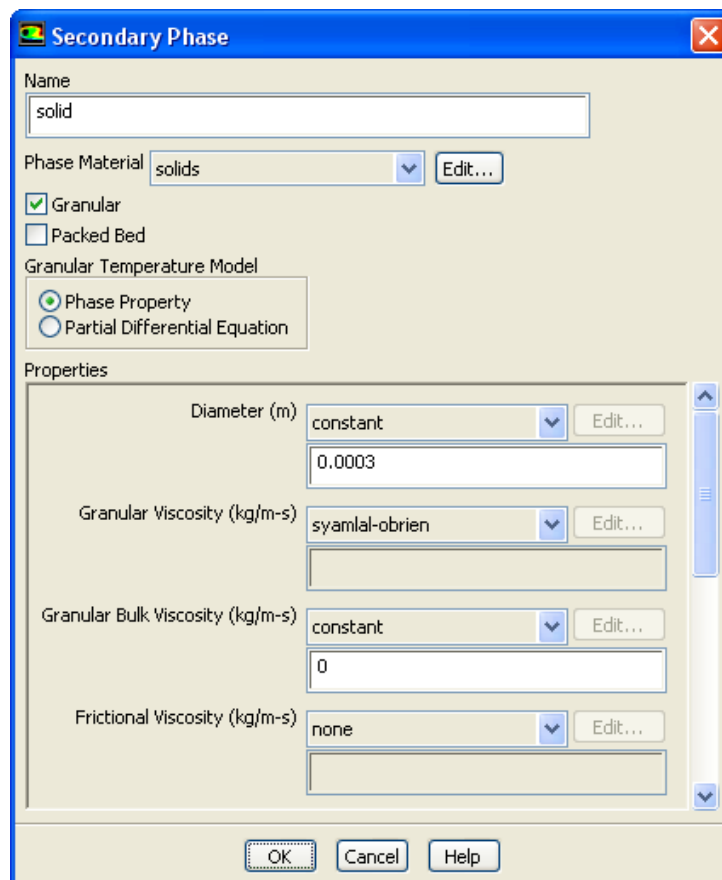
5. Click Load to load the library.

Step 6: Phases

1. Define primary phase.



- (a) Enter gas for Name.
 - (b) Ensure air is selected from the Phase Material drop-down list.
 - (c) Click OK to close the Primary Phase dialog box.
2. Define secondary phase.

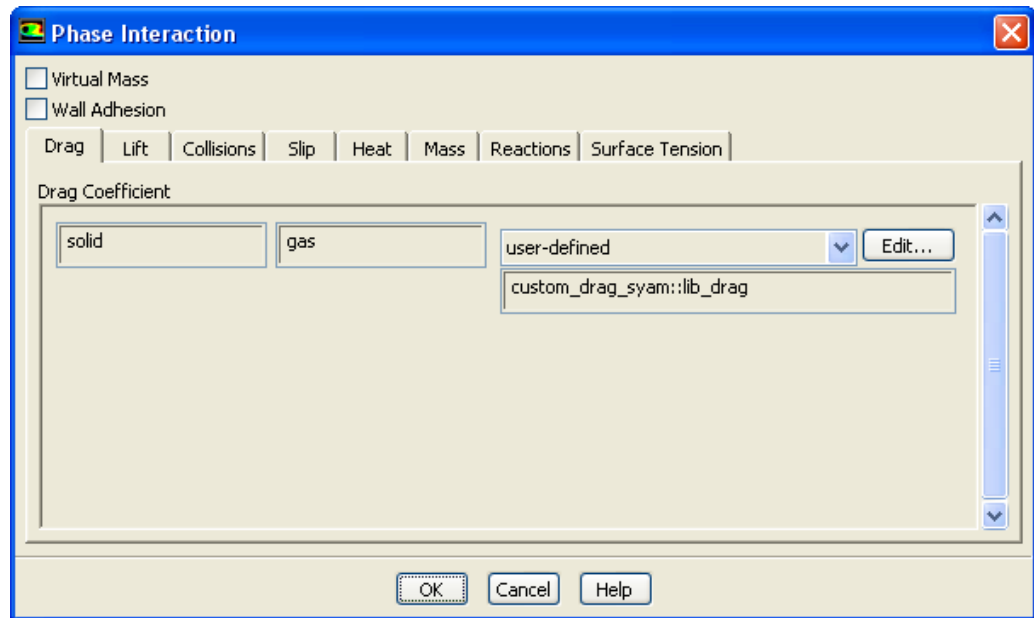


- (a) Enter **solid** for Name.
- (b) Select **solids** from the Phase Material drop-down list.
- (c) Enable **Granular**.
- (d) Enter 0.0003 m for Diameter, and select **syamlal-obrien** from the Granular Viscosity drop-down list.
- (e) Retain the default values for the other parameters.
- (f) Click **OK** to close the Secondary Phase dialog box.

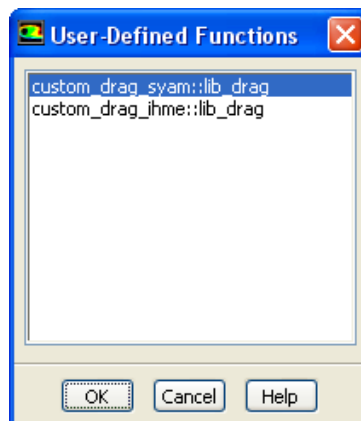
Check the column numbers where the two phases appear in the Phase Interaction dialog box. In this case solid and gas appear in the first and second columns respectively. These columns are used to specify the phase indices in the argument list for the UDF.

3. Set the drag coefficient.

- (a) Select **gas** from the Phases selection list and click the **Interaction...** button to open the Phase Interaction dialog box.



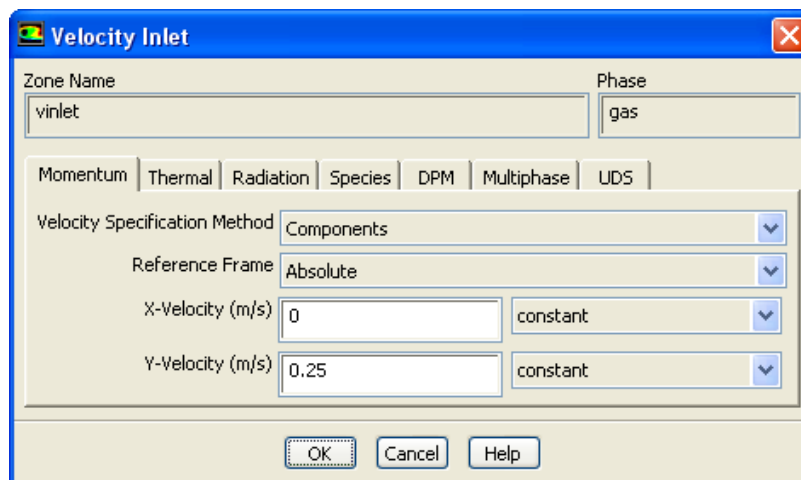
- i. Select **user-defined** from the drop-down list in the Drag Coefficient group box.



- A. Ensure that `custom_drag_syam::lib_drag` is selected.
- B. Click OK to close the User-Defined Functions dialog box
- ii. Click OK to close the Phase Interaction dialog box.
- (b) Similarly select the user defined function for solid (`custom_drag_syam::lib_drag`).

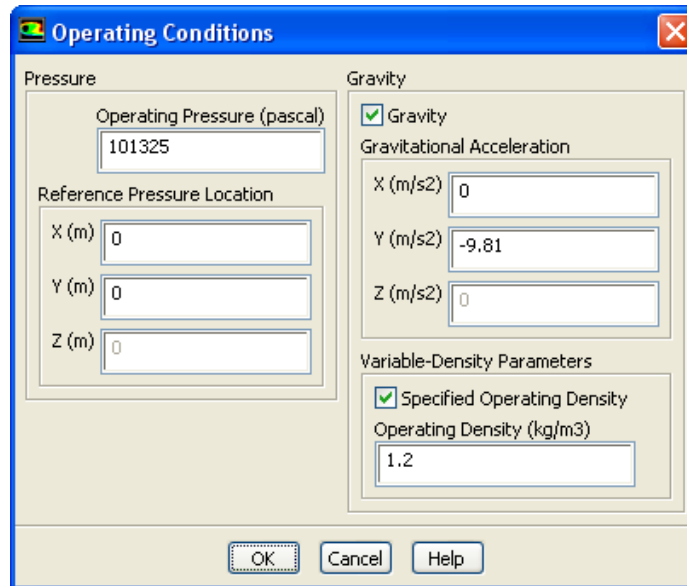
Step 7: Boundary Conditions

1. Set the boundary conditions for vinlet zone.



- (a) Select `gas` from the Phase drop-down list and click Edit....
 - i. Select `Components` from the Velocity Specification Method drop-down list.
 - ii. Enter 0.25 m/s for Y-Velocity.
 - iii. Click OK to close the Velocity Inlet dialog box.
- (b) Select `solid` from the Phase drop-down list and click the Edit... button to open the Velocity Inlet dialog box.
 - i. Click the Multiphase tab.
 - ii. Ensure that Volume Fraction is 0.
 - iii. Click OK to close the Velocity Inlet dialog box.

Step 8: Operating Conditions

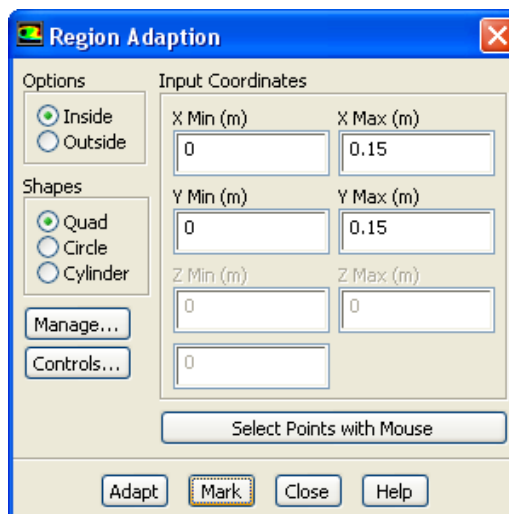


1. Enable Gravity and enter -9.81 m/s^2 for Gravitational Acceleration in the Y direction.
2. Enable Specified Operating Density, and enter 1.2 kg/m^3 for Operating Density.
3. Click OK to close the Operating Conditions dialog box.

Step 9: Solution

1. Mark a region for adaption.

Adapt → Region...



- (a) Enter 0.15 m for X Max and Y Max respectively in the Input Coordinates group box.
- (b) Click Mark to mark the cells for refinement.

Note: Click Adapt to perform the refinement immediately.

- (c) Close the Region Adaption dialog box.

2. Set the solution control parameters.



Solution Controls

Under-Relaxation Factors

Pressure
0.5

Density
1

Body Forces
1

Momentum
0.2

Volume Fraction
0.4

Default

Equations... Limits... Advanced...

Help

- (a) Enter 0.5 for Pressure in the Under-Relaxation Factors group box.
- (b) Enter 0.2 for Momentum.
- (c) Enter 0.4 for Volume Fraction.

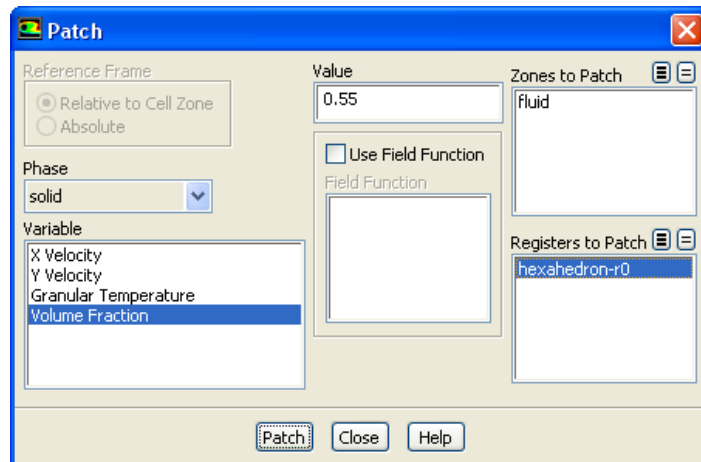
3. Initialize the flow with default values.



Solution Initialization → Initialize

4. Patch the solids volume fraction for hexahedron-r0.

❖ **Solution Initialization** → **Patch...**



- (a) Select **solid** from the Phase drop-down list.
- (b) Select **Volume Fraction** from the Variable selection list.
- (c) Enter **0.55** for Value .
- (d) Select **hexahedron-r0** from the Registers to Patch selection list.

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, enable the Use Field Function option and select the appropriate function in the Field Function list.

- (e) Click **Patch** and close the Patch dialog box.

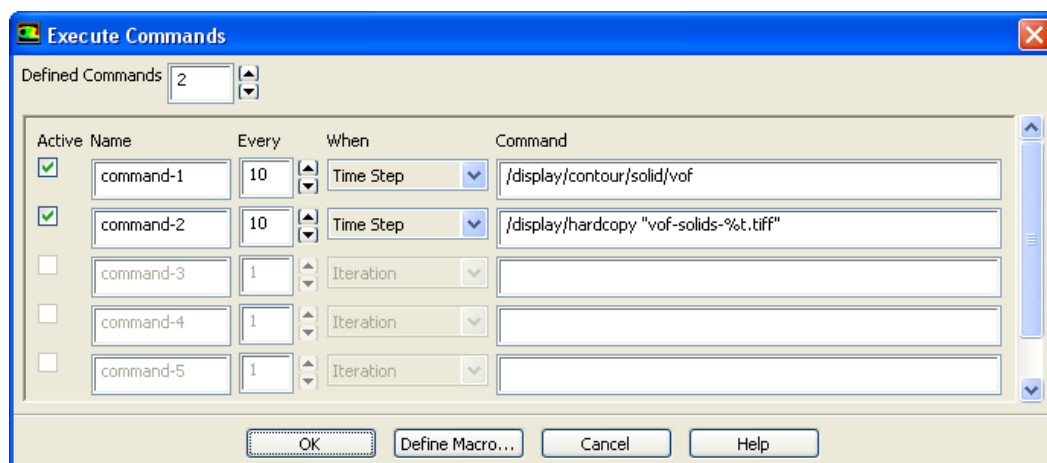
5. Enable autosaving of the data files for every 100 time steps.

❖ **Calculation Activities**

- (a) Enter **100** for Autosave Every (Time Steps).

6. Set up commands for animation.

❖ **Calculation Activities** (Execute Commands) → **Create/Edit...**



- (a) Set 2 for Defined Commands.
 - (b) Enable Active for both commands.
 - (c) Set 10 for Every for both commands.
 - (d) Select Time Step from the When drop-down list for both.
 - (e) Enter `/display/contour/solid/vof` for command-1.
 - (f) Enter `/display/hardcopy "vof-solids-%t.tiff"` for command-2.
 - (g) Click OK to close the Execute Commands dialog box.
7. Set the graphics hardcopy format.
 → Save Picture...
 - (a) Select TIFF from the Format list.
 - (b) Select Color from the Coloring list.
 - (c) Click Apply and close the Save Picture dialog box.
8. Set up the contours display.
 **Graphics and Animations** →  **Contours** → 
 - (a) Enable Filled from the Options group box.
 - (b) Select solid from the Phase drop-down list.
 - (c) Select Phases... and Volume Fraction from the Contours of drop-down lists.
 - (d) Click Display and close the Contours dialog box.
9. Save the case file (`bp.cas.gz`).
 →  → Case...
10. Start the calculation.
 **Run Calculation**.
 - (a) Enter 0.001 sec for Time Step Size.
 - (b) Enter 1400 for Number of Time Steps.
 - (c) Enable Extrapolate Variables.
 - (d) Click Calculate.
11. Save the data file (`bp.dat.gz`).
 →  → Data...

Step 10: Postprocessing

1. Display contours of volume fraction.

(a) Read the data file for the 200th time step (bp-1-00200.dat).

File → Read → Data...

(b) Display filled contours of volume fraction for solid, at 0.2 sec (Figure 3).

Graphics and Animations → Contours → Set Up...

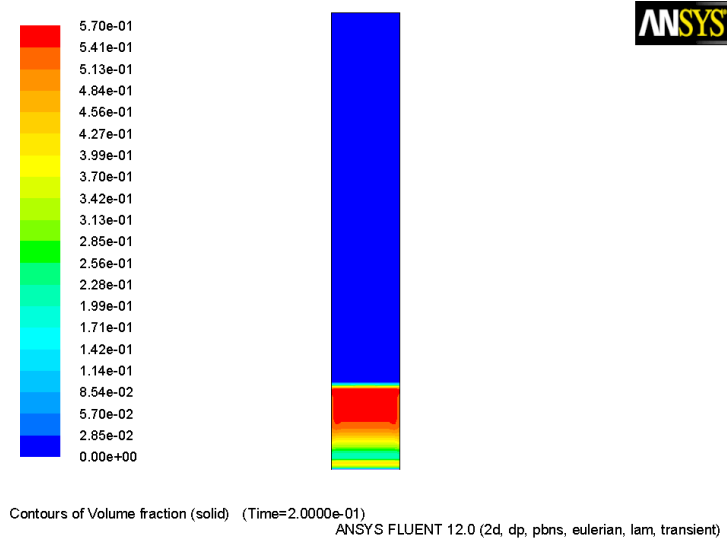


Figure 3: Contours of Volume Fraction of solid ($t = 0.2$ s)

(c) Similarly display contours at 0.9 sec (Figure 4), and 1.4 sec (Figure 5).

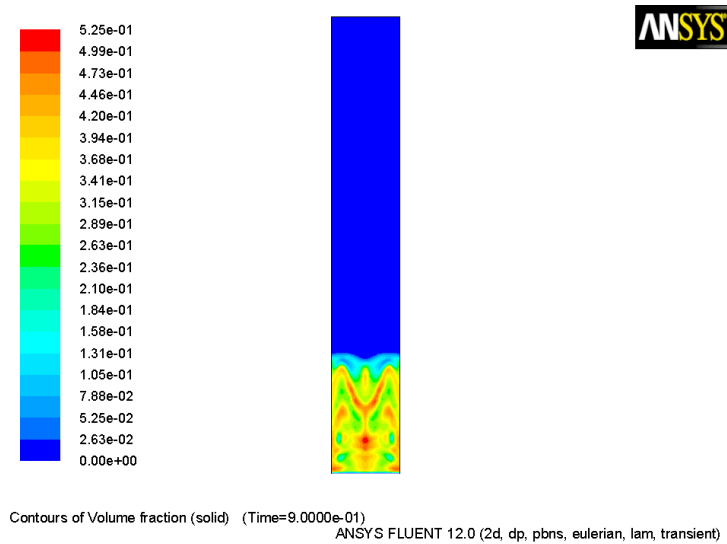


Figure 4: Contours of Volume Fraction of solid ($t = 0.9$ s)

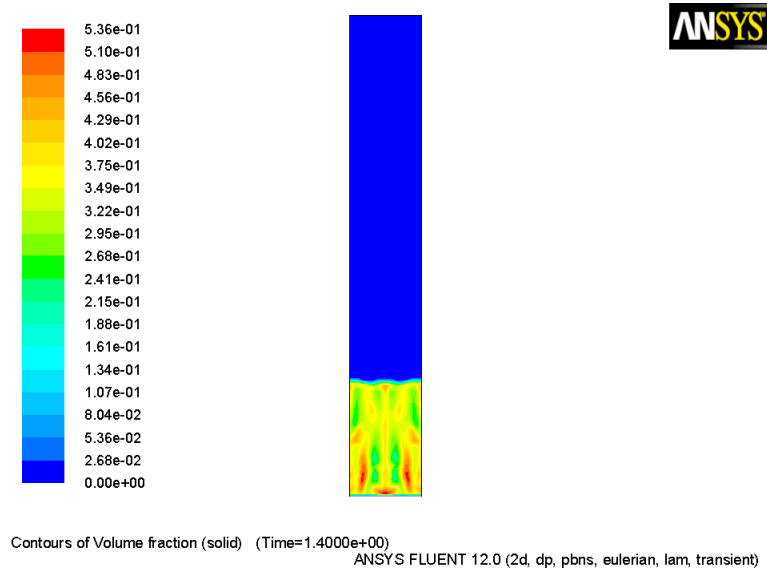


Figure 5: Contours of Volume Fraction of solid ($t = 1.4$ s)

2. View the animation for the fluidization process using the .tiff files.

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. If this is not done, the correct bubbling pattern will not be predicted, leading to incorrect predictions of pressure drop which is the most important objective of such simulations.