Tutorial: Modeling the Effect of Sedimentation Concentration in a Secondary Clarifier using a UDF

Introduction

The purpose of this tutorial is to include a user-defined scalar equation (sediment concentration) in the calculation process. With the help of the user-defined functions (UDFs), a Boussinesq-like approximation is considered. The buoyancy effects due to density gradients are accounted for by a gravity source term in the vertical (axial) momentum equation. The damping effect of stratification on turbulence is introduced by means of source terms in the transport equations of turbulence kinetic energy (k) and its rate of dissipation (e). A rheology function is also considered to account for the increased viscosity of highly concentrated sludge mixtures.

This tutorial demonstrates how to do the following:

- Define an unsteady case with appropriate settings.
- Compile and link a compiled UDF.
- Set up qualitative and quantitative convergence monitors.
- Conduct alphanumeric and graphic postprocessing.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from 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.

Some knowledge of UDF structure, compilation and linking is desired. If you have not used UDFs, ANSYS FLUENT 12.0 UDF Manual will provide you with the necessary experience.

Problem Description

The problem considered is a circular, center-feed secondary clarifier with inclined bottom and central sludge withdrawal. The central inlet includes two baffles: a vertical inlet baffle that forces the inflow to enter the tank at a relatively low position; and a horizontal deflection plate that prevents short circuiting from the inlet to the sludge withdrawal. Axisymmetry is assumed, and the flow and settling processes are modelled in a radial section on a two-dimensional grid [1].

Setup and Solution

Preparation

1. Copy the files, (clarifier.msh.gz) and (clarifier.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* ANSYS FLUENT *12.0 User's Guide.*

Step 1: Mesh

1. Read the file, `clarifier.msh.gz`.

   File ⟶ Read ⟶Case...

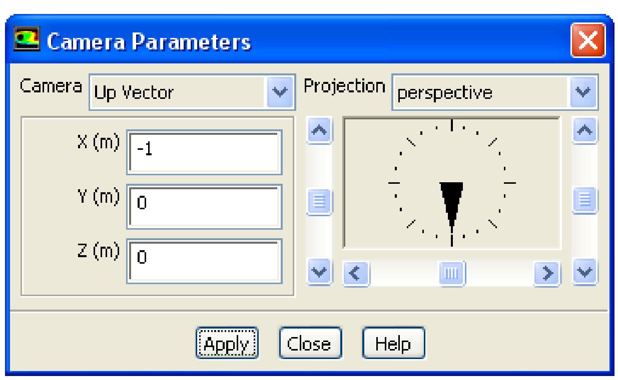
2. Orient the view with the camera tool.

   Display ⟶Views...

   (a) Click the Camera... button to open the Camera Parameters dialog box.

   (b) Select Up Vector from the Camera drop-down list.

   (c) Set X, Y, Z to `-1, 0, 0`.



(d) Click Apply and close the Camera Parameters dialog box.

(e) Click Apply and close the Views dialog box.

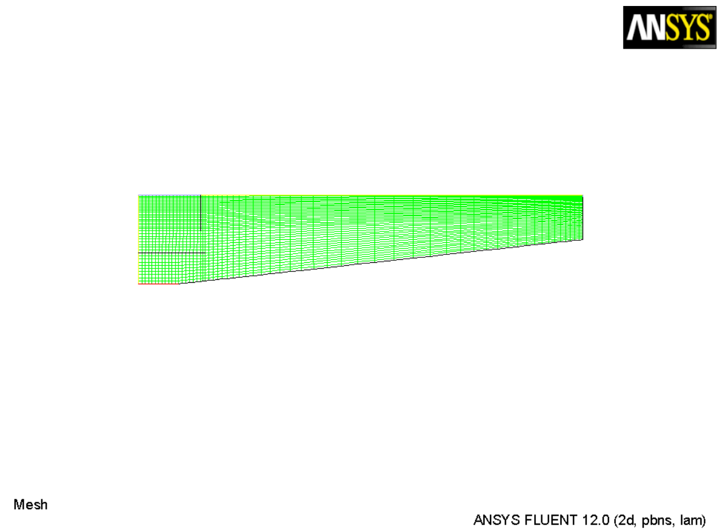


Figure 1: Mesh Display

Step 2: General Settings

1. Check the mesh.

   General ⟶ Check

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

2. Define the solver settings.

   General

   (a) Enable Transient.

   (b) Select Axisymmetric from the 2D Space list.

Step 3: Models

1. Enable the k-ϵ turbulence model.

   Models ⟶ Viscous ⟶ Edit...

2. Select k-ϵ (2 eqn) from the Model list.

3. Retain the default selection of Standard Wall Functions from the Near-Wall Treatment list.

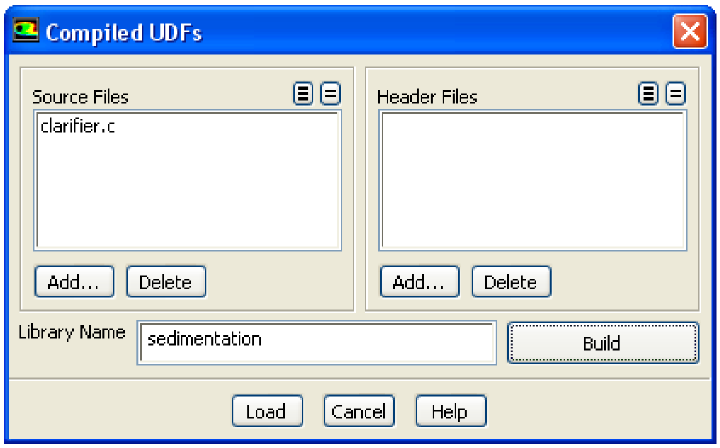
   Based on Reynolds number calculation at the inlet, the flow is expected to be turbulent, so an appropriate turbulence model is required.

4. Click OK to close the Viscous Model dialog box.

Step 4: Compile the UDF

Compile the user-defined function.

Define ⟶ User-Defined ⟶ Functions ⟶Compiled...



1. Click the Add... button in the Source Files section.

2. Select the `clarifier.c` file and click OK in the Select File dialog box.

3. Enter `sedimentation` in the Library Name text entry box

4. Click Build to compile the UDF code.

   ANSYS FLUENT *displays a* Warning *dialog box asking you to make sure that UDF source files are in the folder that contains the case and data files. Click* OK *in the* Warning *dialog box. All related files following the compilation will be stored in the sedimentation folder.*

5. Click Load so that all the macros are available in the graphical user interface.

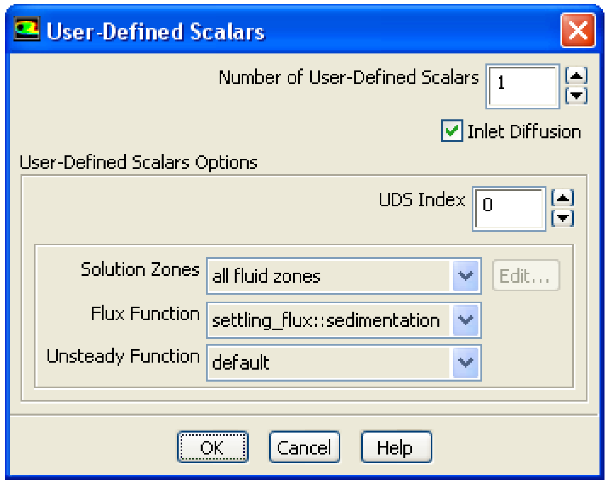
For more details regarding the resolved system of equations, refer to Appendix 2: Compiling the UDF.

Step 5: User-Defined Scalars

Besides the continuity, momentum, and turbulence transport equation, a new passive scalar equation is defined to determine local particle concentration (6). Although the standard GUI does not allow the direct activation of the concentration equation, you can introduce the computation of a general transport equation, whose terms are adapted to yield the final sediment concentration equation.

1. Enable the user defined scalar.

   Define ⟶ User-Defined ⟶Scalars...



(a) Increase the Number of User-Defined Scalars to 1.

(b) Select setting_flux::sedimentation from the Flux Function drop-down list.

(c) Retain the selection of default from the Unsteady Function drop-down list.

(d) Click OK to close the User-Defined Scalars dialog box.

ANSYS FLUENT *displays an* Information *dialog box informing that available material properties or methods have changed. Click* OK *in the* Information *dialog box.*

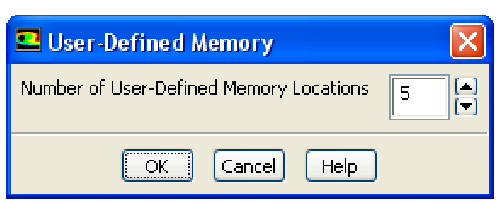
Step 6: User-Defined Memory

A user-defined memory (UDM) is a customized variable defined within the UDF code. UDMs can store any expression combining flow variables and user constants. They are easily accessed and can be used for postprocessing (contours, vectors, XY plots, etc.).

In this tutorial, five UDMs have been defined to store quantities as shown in the following table:

UDM	Description
0	Settling Velocity
1	UDS Diffusion Coefficient
2	Gravity Source Term
3	Turbulence Kinetic Rate Source Term
4	Turbulence Dissipation Rate Source Term

Define → User-Defined → Memory...



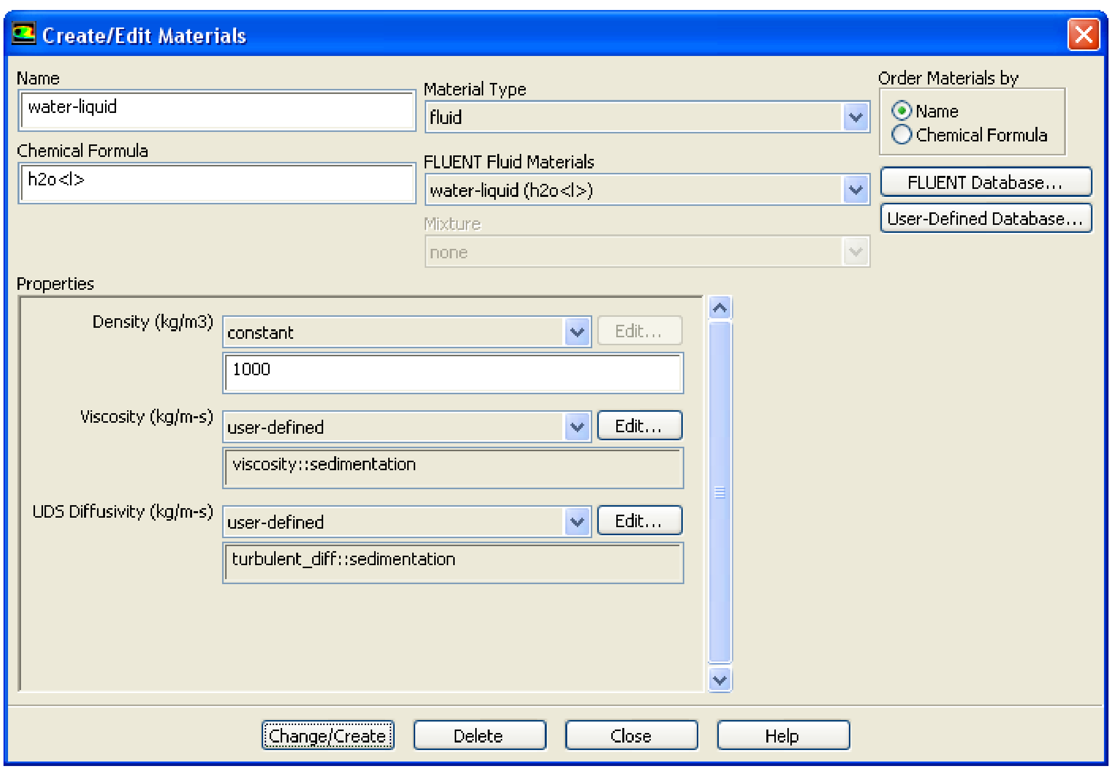
1. Increase the Number of User-Defined Memory Locations to 5.
2. Click OK to close the User-Defined Memory dialog box.

Step 7: Materials

Materials ⟶ Create/Edit...

The Create/Edit Materials *dialog box for* air *opens.*

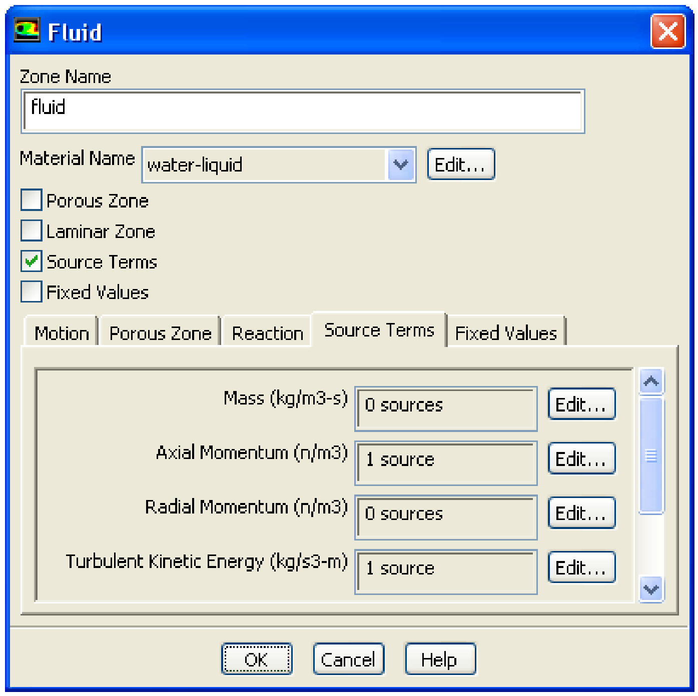
1. Select user-defined from the UDS Diffusivity drop-down list.
2. Select turbulent_diff::sedimentation and click OK to close the User-Defined Functions dialog box.
3. Click Change/Create.
4. Import the water-liquid material from the ANSYS FLUENT Materials Database.
5. Enter 1000 kg/m^3 for Density.
6. Select user-defined from the Viscosity drop-down list and select viscosity::sedimentation in the User-defined Functions dialog box.
7. Select user-defined from the UDS Diffusivity drop-down list and select turbulent_diff::sedimentation in the User-Defined Functions dialog box.



8. Click Change/Create and close the Materials dialog box.

Step 8: Cell Zone Conditions

Cell Zone Conditions ⟶ fluid ⟶ Edit...



1. Select water-liquid from the Material Name drop-down list.
2. Enable Source Terms.
3. Click on the Source Terms tab.
   (a) Click the Edit... button next to Axial Momentum to open the Axial momentum (n/m^3) sources dialog box.
      i. Set the Number of Axial Momentum (n/m^3) sources to 1.
      ii. Select udf X_mom_src::sedimentation from the drop-down list.
      iii. Click OK to close the Axial momentum (n/m^3) sources dialog box.
4. Similarly, enable the UDF source term udf turb_k_source::sedimentation for Turbulent Kinetic Energy.
5. Also enable udf turb_e_source::sedimentation for Turbulent Dissipation Rate.
6. Click OK to close the Fluid dialog box.

Step 9: Boundary Conditions

1. Set boundary conditions for inlet.

   Boundary Conditions ⟶ inlet ⟶ Edit...

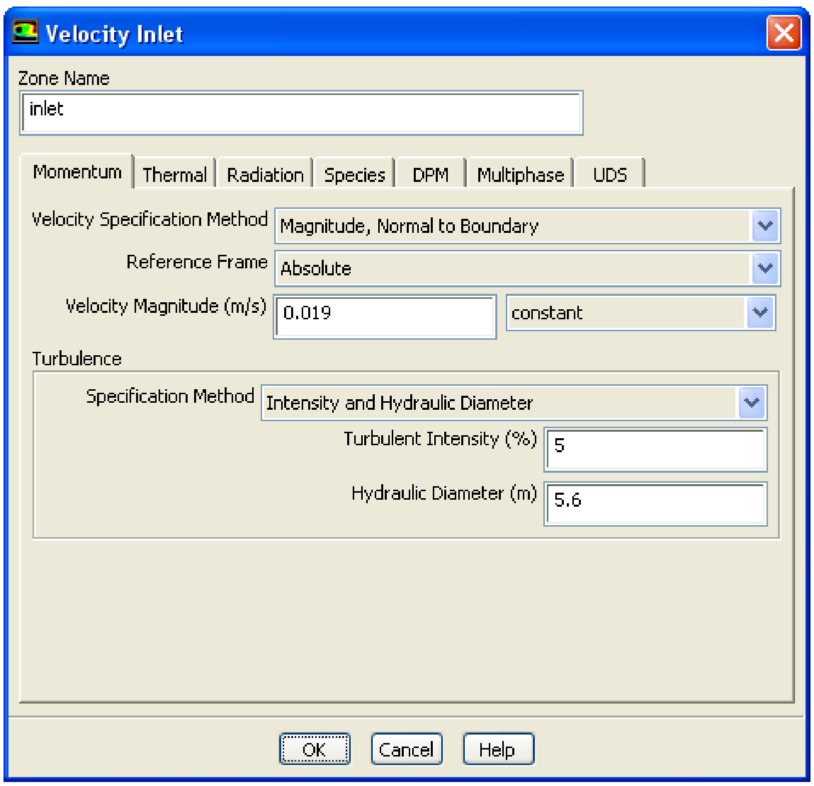
   (a) Enter 0.019 m/s for Velocity Magnitude.

   (b) Select Intensity and Hydraulic Diameter from the Specification Method in the Turbulence group box.

   (c) Enter 5% and 5.6 m for Turbulent Intensity and Hydraulic Diameter, respectively.

   (d) Click the UDS tab and select Specified Value from the User Scalar 0 drop-down list.

   (e) Enter 3.2 for User Scalar 0.



   (f) Click OK to close the Velocity Inlet dialog box.

2. Set boundary conditions for outlet_top.

   Boundary Conditions ⟶ outlet_top ⟶ Edit...

   (a) Enter -0.04 m/s for Velocity Magnitude.

   (b) Select Intensity and Hydraulic Diameter from the Specification Method in the Turbulence group box.

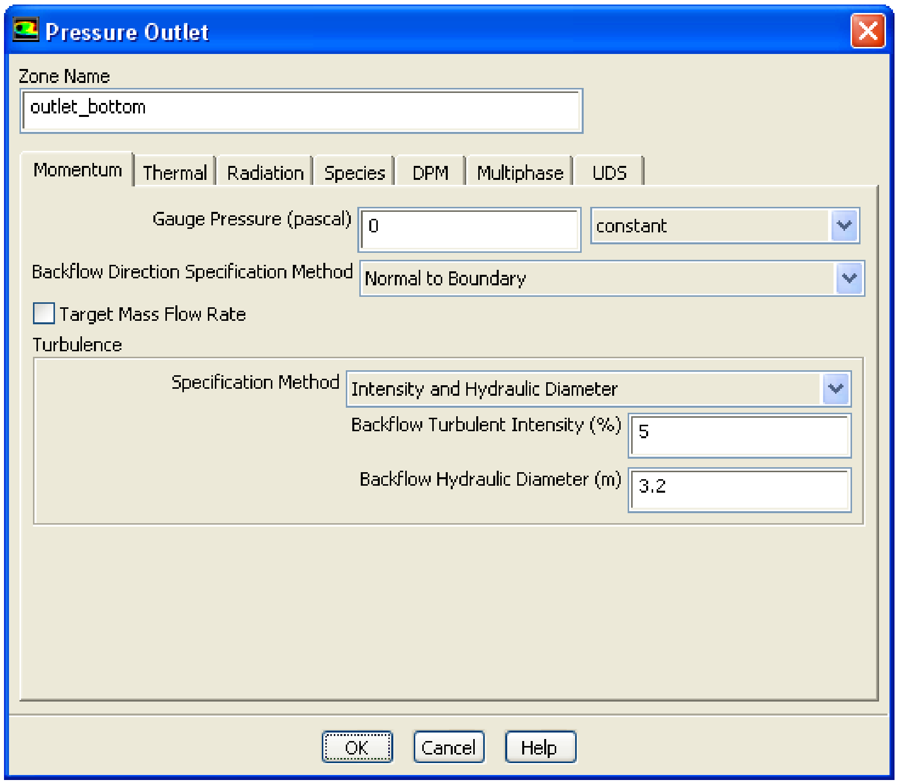
   (c) Enter 5% and 0.02 m for Turbulent Intensity and Hydraulic Diameter, respectively.

   (d) Retain the default parameters in the UDS tab.

(e) Click OK to close the Velocity Inlet dialog box.

3. Set boundary conditions for outlet_bottom.

Boundary Conditions ⟶ outlet_bottom ⟶ Edit...



(a) Select Intensity and Hydraulic Diameter from the Specification Method in the Turbulence group box.

(b) Enter 5% and 3.2 m for Backflow Turbulent Intensity and Backflow Hydraulic Diameter, respectively.

(c) Retain the default parameters in the UDS tab and click OK to close the Pressure Outlet dialog box.

4. Retain the default settings for other zones.

Step 10: Solution

1. Define the solution control parameters.

Solution Methods

(a) Select Green-Gauss Cell Based from the Gradient drop-down list.

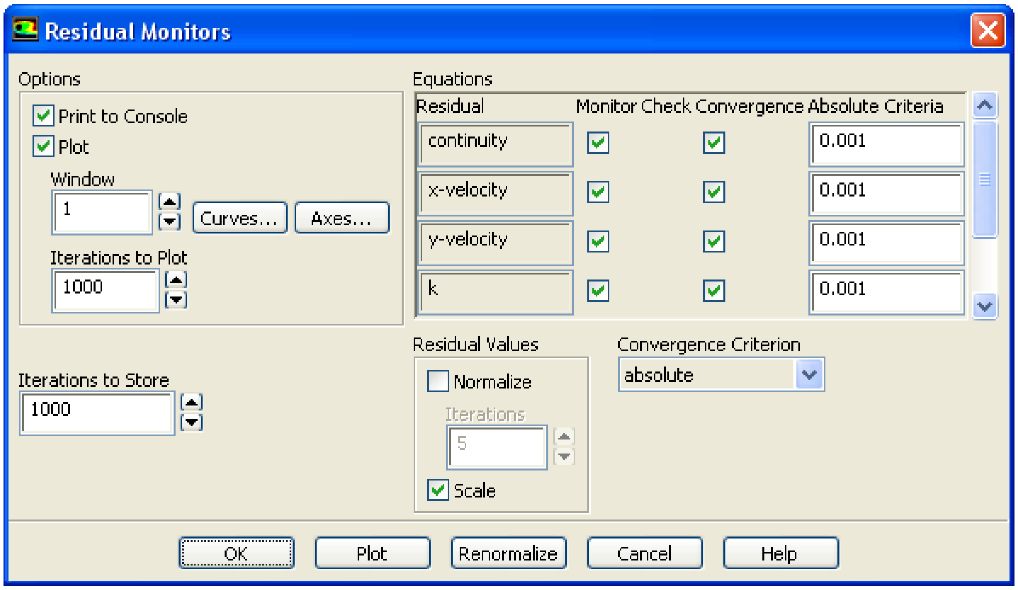
(b) Select PRESTO! from the Pressure drop-down list.

(c) Select QUICK for all the other parameters.

(d) Select Second Order Implicit from the Transient Formulation drop-down list.

2. Enable residual plotting.

   Monitors ⟶ Residuals ⟶ Edit...

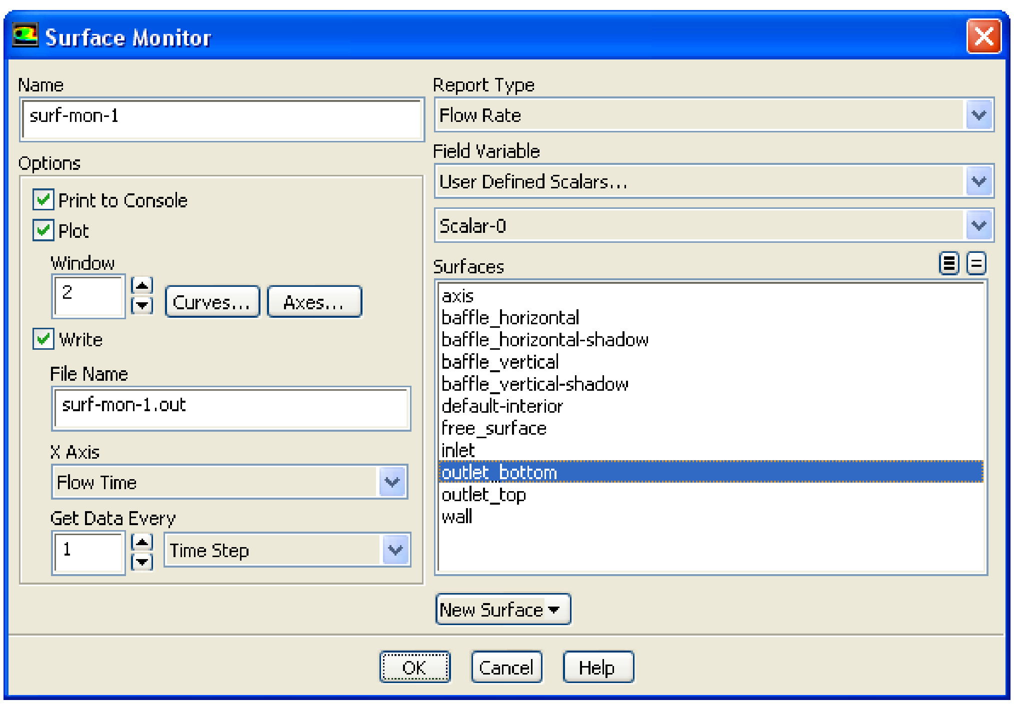


   (a) Ensure that Plot is enabled in the Options group box.

   (b) Click OK to close the Residual Monitors dialog box.

3. Define a surface monitor.

   Monitors (Surface Monitors) ⟶ Create...



(a) Enable Print to Console, Plot, and Write.

(b) Select Flow Time from the X Axis drop-down list.

(c) Select Time Step from Get Data Every drop-down list.

(d) Select Flow Rate from the Report Type drop-down list .

(e) Select User Defined Scalars... and Scalar-0 from the Field Variable drop-down lists.

(f) Select outlet_bottom from the Surfaces selection list.

(g) Click OK to close the Surface Monitor dialog box.

4. Initialize the solution.

   Solution Initialization

   (a) Select all-zones from the Compute From drop-down list.

   (b) Click Initialize.

5. Write the case and data files (`clarifier-t=0.cas.gz` and `clarifier-t=0.dat.gz`).

   File ⟶ Write ⟶ Case & Data...

6. Run the calculations.

   Run Calculations

   (a) Enter 10 for Time Step Size and 7300 for Number of Time Steps.

   (b) Click Calculate.

   The hydraulic residence time (hrt = ratio of volume to flowrate) of the sedimentation tank is approximately 7300 seconds. It is assumed that the steady-state solution is achieved for 10 times the hrt (conservative estimation).

   The size of the time step size is a compromise between numerical stability and computational time. A time step of 10 seconds is chosen which yields a total number of 7300 time steps to reach a steady-state solution.

   The outcome of the simulation is the steady-state clarifier behavior and intermediate time-dependent information will be neglected.

   Depending on computer resources, the simulation may take several hours to complete.

7. Write the case and data files (`clarifier-steady.cas.gz` and `clarifier-steady.cas.gz`).

   File ⟶ Write ⟶ Case & Data...

Step 11: Postprocessing

1. Display contours of particle concentration, User Scalar 0 (Figure 2).

   Graphics and Animations ⟶ Contours ⟶ Set Up...

2. Display velocity vectors colored by velocity magnitude (Figure 3).

   Graphics and Animations ⟶ Vectors ⟶ Set Up...

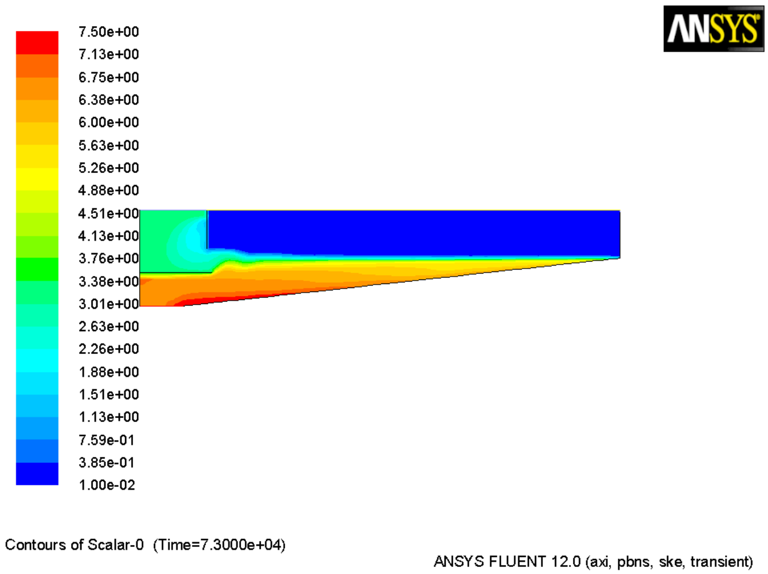


Figure 2: Contours of Particle Concentration

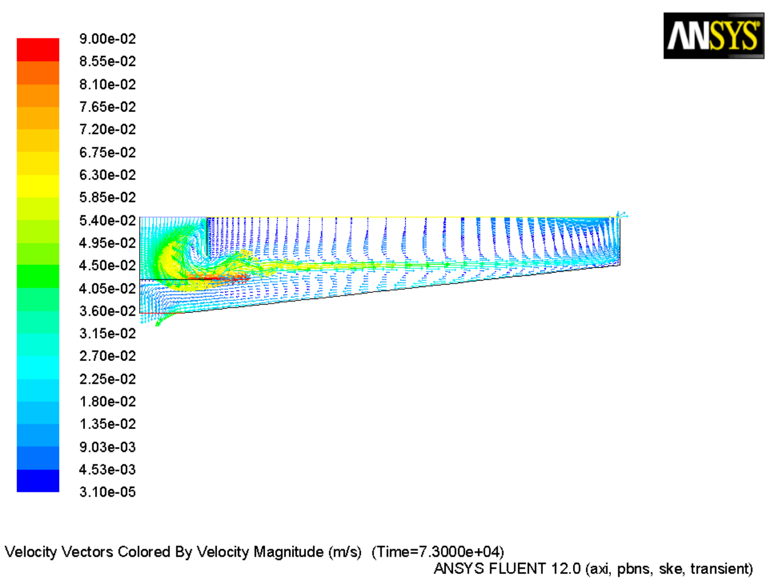


Figure 3: Velocity Vectors

(a) Increase Scale to 5 to improve vector viewing.

(b) Click Display and close the Vectors dialog box.

3. Display contours of settling velocity, User Memory 0 (Figure 4).

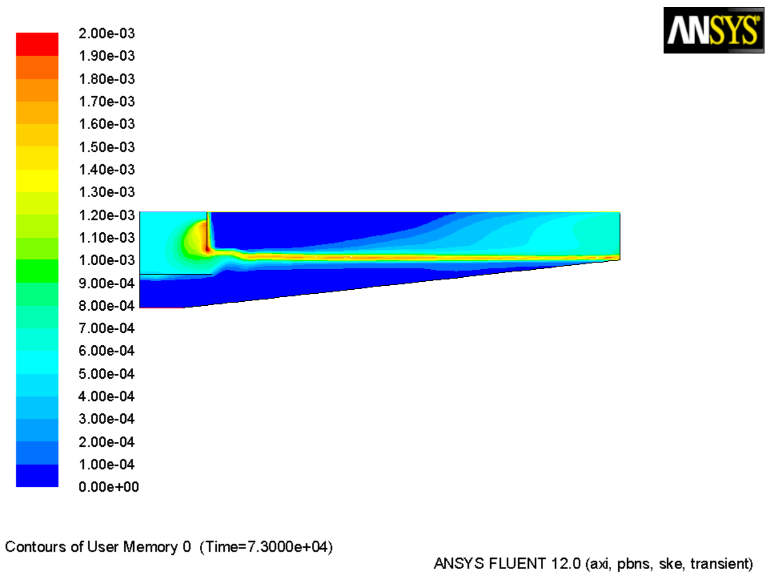
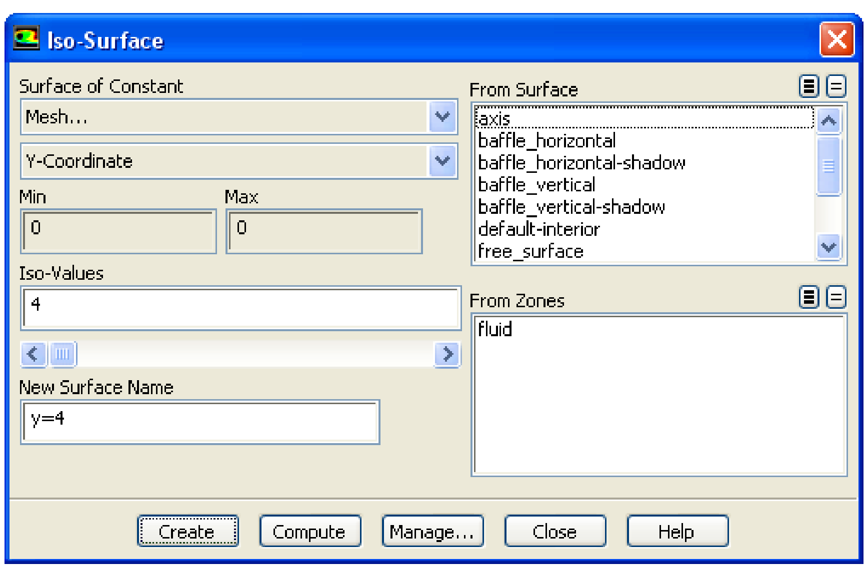


Figure 4: Contours of Settling Velocity

4. Create an isosurface at the radial coordinate, y=4 m.

Surface ⟶Iso-Surface...



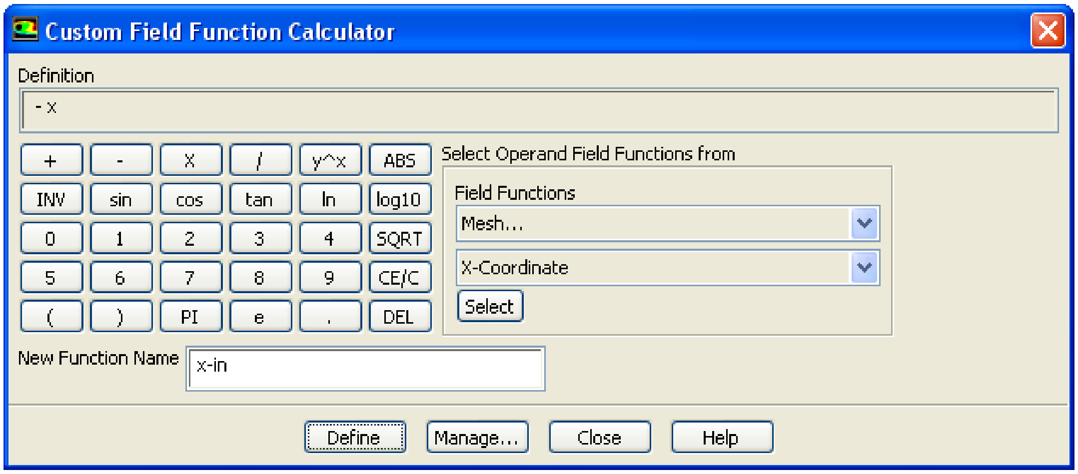
(a) Select Mesh... and Y-Coordinate from the Surface of Constant drop-down lists.

(b) Enter 4 for Iso-Values and y=4 for New Surface Name.

(c) Click Create and close the Iso-Surface dialog box.

5. Create two custom field functions.

   Define ⟶Custom-Field Functions...



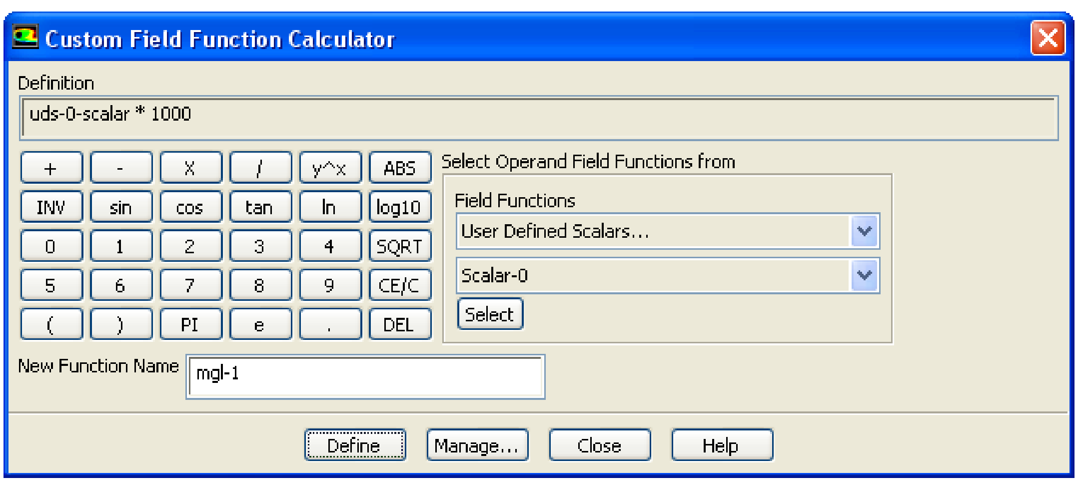
   (a) Click the - button on the keypad in the dialog box.

   (b) Select Mesh... and X-Coordinate from the Field Functions drop-down lists.

   (c) Click the Select button and enter `x-in` in the New Function Name text entry box.

   (d) Click Define.

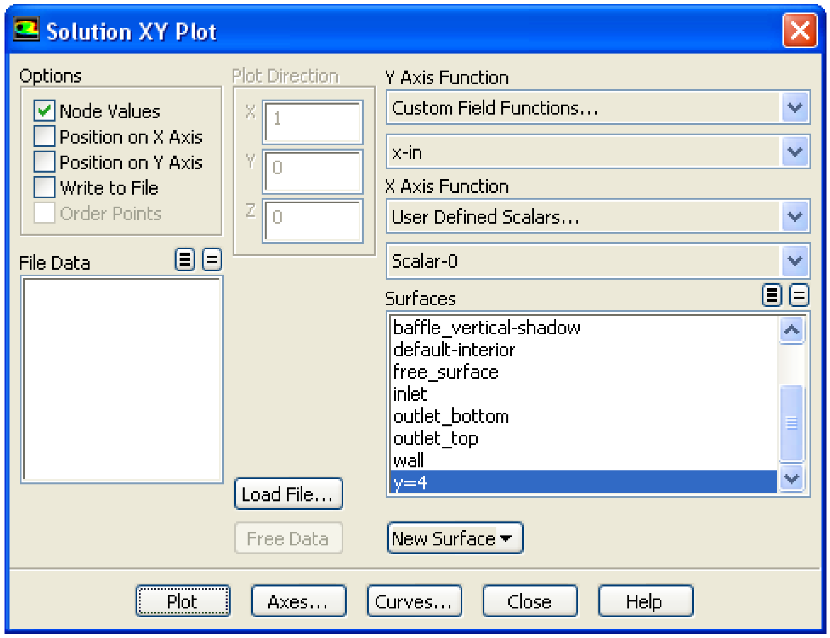
   (e) Similarly, create the second custom field function, mgl-1.



   (f) Close the Custom Field Function Calculator dialog box.

6. Create an XY plot to display concentration profiles at a radial position (Figure 5).

   Plots ⟶ XY Plot ⟶ Set Up...



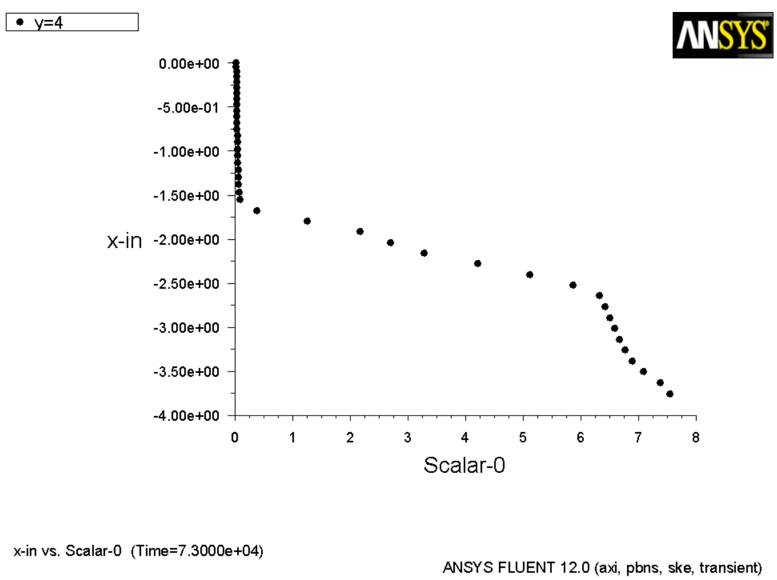
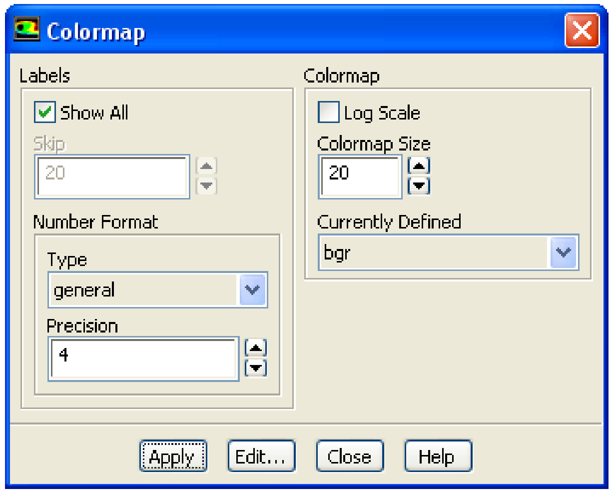


Figure 5: Concentration Profile at y = 4 m

7. Modify the number format.

   Display ⟶Colormap...

   Changing the number format (type and precision) eases results interpretation for a non-technical audience.



(a) Select general from the Type drop-down list.

(b) Enter 4 for Precision.

(c) Click Apply and close the Colormap dialog box.

8. Display concentration contours in mg/l units (custom field function and mgl-1).

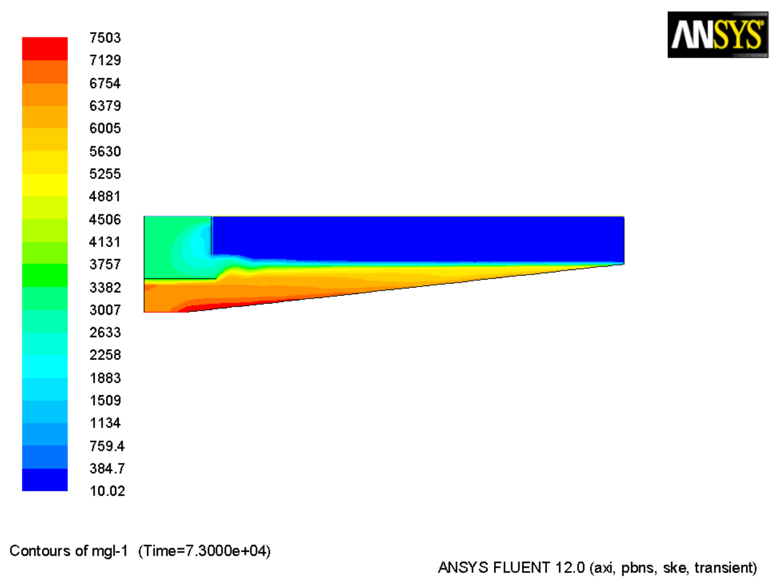


Figure 6: Contours of Particle Concentration (mg/l)

Appendix 1: Parameters

The loading conditions for this reference case can be found on the publication in [1]. For simplicity, some key parameters are summarized in the following table:

Parameter	Description	Value
U$_i n$	Inflow Velocity	0.019 m/s
C$_i n$	Inflow Particle Concentration	3.2 kg/m^3
R	Recycle Ratio	0.86
ρ_p	Dry Particle Density	1450 kg/m^3
ρ_w	Clear Water Density	1000 kg/m^3
σ_c	Schmidt Number	0.7
US0	Reference Settling Velocity	0.005 m/s
RH	Floc Settling Parameter	0.7 m^3/kg
RP	Colloidal Settling Parameter	5 m^3/kg
CMIN	Nonsettleable Concentration	0.01 kg/m^3
USMAX	Maximum Settling Velocity	0.002 m/s

Appendix 2: Compiling the UDF

The resolved system of equations is shown below. Some slight modifications are conducted on the general form of the transport equations to account for the complex physics of the clarification process. The sedimentation-related terms are marked and associated with the corresponding UDF macros.

1. Continuity Equation:

$$\rho\frac{\partial V_x}{\partial x} + \rho\frac{\partial V_y}{\partial y} + \frac{\rho V_y}{y} = 0 \tag{1}$$

2. X-Momentum Equation:

$$\rho\frac{\partial V_x}{\partial t}+\rho\frac{\partial V_x^2}{\partial x}+\frac{\partial (V_x V_y)}{\partial y} = -\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(2\mu_t\frac{\partial V_x}{\partial x}\right)+\frac{1}{y}\frac{\partial}{\partial y}\left[y\mu_t\left(\frac{\partial V_x}{\partial y}+\frac{\partial V_y}{\partial y}\right)\right]+\frac{gC(\rho_p-\rho_w)}{\rho_w} \tag{2}$$

where,

$\frac{gC(\rho_p-\rho_w)}{\rho_w}$: DEFINE_SOURCE(X_mom_src,c,tc,ds,eqn)

3. Y-Momentum Equation:

$$\rho\frac{\partial V_y}{\partial t}+\rho\frac{\partial V_y^2}{\partial x}+\rho\frac{\partial (V_x V_y)}{\partial x} = -\frac{\partial p}{\partial y}+\frac{1}{y}\frac{\partial}{\partial y}\left(2y\mu_t\frac{\partial V_x}{\partial y}\right)+\frac{\partial}{\partial x}\left[\mu_t\left(\frac{\partial V_x}{\partial y}+\frac{\partial V_y}{\partial x}\right)\right]-2\mu_t\frac{V_y}{y^2} \tag{3}$$

4. Turbulent Kinetic Energy (k) Equation:

$$\rho\frac{\partial k}{\partial t}+\rho\frac{\partial V_x k}{\partial x}+\rho\frac{\partial (V_y k)}{\partial y}=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x}\right]+\frac{1}{y}\frac{\partial}{\partial y}\left[y\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial y}\right]+G_k+G_b-\rho\epsilon \tag{4}$$

where,

G_b: DEFINE_SOURCE (turb_k_source,c,tc,ds,eqn)

$$G_b = \left(\frac{\rho_p-\rho_w}{\rho_p\rho_w}\right)g\left(\frac{\mu_t}{\sigma_c}\frac{\partial C}{\partial x}\right)$$

5. Turbulent Dissipation Rate (ϵ) Equation:

$$\rho\frac{\partial \epsilon}{\partial t}+\rho\frac{\partial V_x \epsilon}{\partial x}+\rho\frac{\partial (V_y \epsilon)}{\partial y}=\frac{\partial}{\partial x}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x}\right]+\frac{1}{y}\frac{\partial}{\partial y}\left[y\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial y}\right]+C_1\epsilon\frac{\epsilon}{k}(G_k-C_3\epsilon G_b)-\rho C_2\epsilon\frac{\epsilon^2}{k} \tag{5}$$

where,

$C_3\epsilon G_b$: DEFINE_SOURCE(turb_e_source,c,tc,ds,eqn)

6. Concentration Equation (Passive Scalar Equation):

$$\rho\frac{\partial C}{\partial t}+\rho\frac{\partial (V_x+V_s)C}{\partial x}+\rho\frac{\partial (V_y C)}{\partial y}=\frac{\partial}{\partial x}\left[\frac{\mu_t}{\sigma_c}\frac{\partial C}{\partial x}\right]+\frac{1}{y}\frac{\partial}{\partial y}\left[y\frac{\mu_t}{\sigma_c}\frac{\partial C}{\partial y}\right] \tag{6}$$

where,

$\rho\ \frac{\partial(V_x+V_s)}{\partial x}C+\rho\ \frac{\partial(V_y C)}{\partial y}$: DEFINE_UDS_FLUX(settling_flux,f,tf,i)

$\frac{\partial}{\partial x}\left[\frac{\mu_t}{\sigma_c}\frac{\partial C}{\partial x}\right]+\frac{1}{y}\frac{\partial}{\partial y}\left[y\frac{\mu_t}{\sigma_c}\frac{\partial C}{\partial y}\right]$: DEFINE_DIFFUSIVITY(turbulent_diff,f,tf,i)

References

[1]Computing Shear Flow and Sludge Blanket in Secondary Clarifiers
Djamel Lakehal, Peter Krebs, Johan Krijgsman, Wolfgang Rodi