CHAPTER 7

CFD MODELLING OF POLLUTANT FLOW DYNAMICS IN THE RIVER GANGA

7.1 INTRODUCTION

Computational Fluid Dynamics (CFD) is a helpful technique to model the pollutant flow pattern in the river. It will help to predict the dispersion of the pollutant at the different section of the river. This result will help industries and the environment at agencies to reduce effluent flow in the river to maintain a healthy environment. This chapter contains the methodology used for pollutant dynamics modelling, results obtained, and a summary discussed in the following section.

7.2 STUDY AREA

For modelling of the pollutant dynamics, a 10 km stretch near Varanasi was considered for the investigation. A detailed description of the study area has been provided in subsection 3.2.4 of chapter 3.

7.3 METHODOLOGY

The ANSYS software package is used for CFD modelling of pollutant flow dynamics in the river Ganga. Wastewater from the 'Assi Nallah' was taken as the pollutant source for river Ganga in this study. The ANSYS software works on the finite element method. The detail of the methodology is presented in Figure 7.1. The geometry profile of the river with pollutant drain was prepared. After this, meshing was done, and the Fluent interface was prepared. Then fluid properties and boundary conditions were entered. Through solver, property output results were obtained in terms of the flow pattern of pollutant in river Ganga. The details of the study area considered, mathematical formulation, data used, and data processing is presented in the following sub-sections.

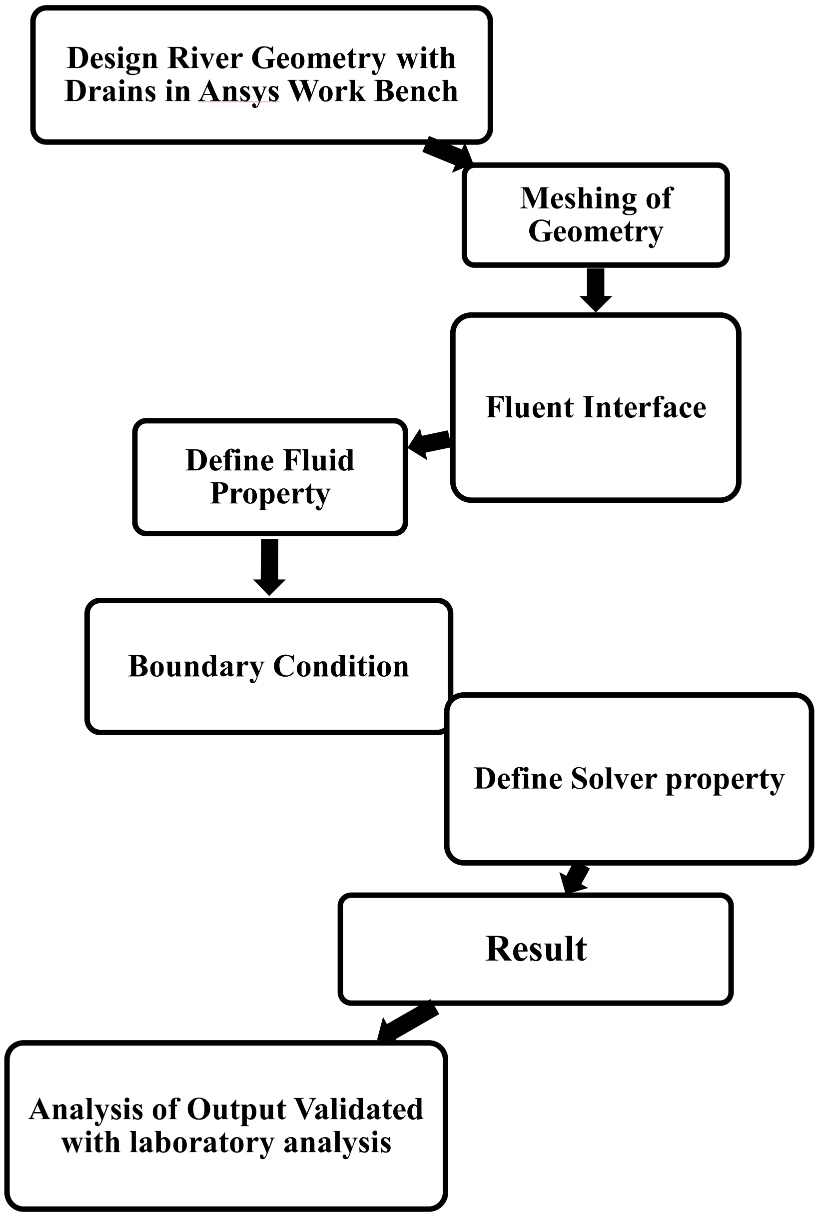

Figure 7.1 Flow chart of the methodology

7.4 MATHEMATICAL FORMULATION

7.4.1 ASSUMPTIONS

In this study, a multiphase system was considered, such as air, water and pollutant. Newtonian non-viscous condition considers for air and water. CFD was considered for the close visualization of pollutant flow in the river. The flow is unsteady, and the mixing of pollutant is also unsteady.

River Ganga near Varanasi is taken for the computational experiment. The length of the channel is 10 km, the width of the channel varies from 300 m to 1000 m, and the water depth from 3 m to 22 m.

7.4.2 GOVERNING EQUATIONS

The governing equations based upon the assumption can be written as follows:

$$\frac{\partial \rho}{\delta t} + \frac{\partial \rho U_i}{\partial x_i} = 0 \tag{7.1}$$

$$\frac{\partial \rho U_i}{\partial t} + U_i \frac{\partial \rho U_i}{\partial x_i} = \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial P}{\partial x_i} + \rho g_i + S_{i,s} \tag{7.2}$$

For the simulations of sharp fluid-fluid interfaces using a finite volume approach, the VOF model is very suitable. The idea with the VOF method is that the interface between two phases is tracked. This is done by a colour function (phase indicator function) that indicates the fractional amount of fluid at a particular position. This method can simulate flows, including the shape and evolution of the free surface.

In continuity Eq. (1), a single velocity field is shared by the two phases. It indicates that there is a continuous velocity of the phases across the interface. In momentum Eq. (2), the interaction between the phases is modelled by the surface tension $S_{i,s}$.

The tracking of the interface between the phases is done with the solution of the continuity Eq. (1) for the second phase. This interface is so calculated with the following equation:

$$\frac{\partial(\alpha_2\rho_2)}{\delta t} + \frac{\partial(\alpha_2\rho_2 U_i)}{\partial x_i} = s_2 \tag{7.3}$$

where,

s_2= the source of phase 2

s_2 is equal to zero in this work,

ρ_2 = the density of the secondary phase and

α_2 = the volume fraction of the secondary phase

$\alpha_2 = V_2/V$

V = the total volume of fluids ($V = V_1 + V_2$); $V1$ is the volume of phase 1, and V_2 is the volume of phase 2.

The volume fraction of the primary phase ($\alpha_1 = V_1/V$) is calculated by the constraining

$$\sum_{i=1}^{2} \alpha_i = 1 \tag{7.4}$$

The interface that is calculated with Eq. (3) was reconstructed with the Geo- Reconstruction scheme. The geometric reconstruction scheme uses a piecewise linear approach to rebuild the interface between the phases. It assumes that the interface between two phases has a linear slope within the cell. This linear shape is used to calculate the advection of the fluid through the cell faces. The introduction of the Reynolds stress tensor τ_{ij} In the momentum, Eq. (2) requires the use of a turbulent closure model. The standard $k - \epsilon$ turbulence model was chosen for its ability to model the turbulent features and to generate the least number of cells compared to other models. This model is based on the Boussinesq hypothesis, which assumes that the Reynolds stress is proportional to the mean velocity gradient, with the constant of proportionality being the turbulent viscosity. This quantity is given by:

$$\nu_2 = C_\mu \frac{k^2}{\epsilon} \tag{7.5}$$

where,

k = the kinetic energy,

ϵ = the dissipation rate and

C_μ = an empirical constant of the standard $k - \epsilon$ model.

The following equations give the kinetic energy and the dissipation rate:

$$\frac{\partial k}{\delta t} + \frac{\partial U_i k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right) + \frac{\partial(k)}{\partial x_i}\right] + P_k - \epsilon \tag{7.6}$$

$$\frac{\partial \epsilon}{\delta t} + \frac{\partial U_i \epsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\nu + \frac{\nu_t}{\sigma_\epsilon}\right) + \frac{\partial(\epsilon)}{\partial x_i}\right] + C_{1\epsilon} P_k \frac{\epsilon}{k} - C_{2\epsilon} \frac{\epsilon^2}{k} \tag{7.7}$$

where v is the viscosity of fluids and P_k is the production for turbulence given by:

$$P_k = -\overline{u_i' u_j'} \frac{\partial U_j}{\partial x_i} = \left[\nu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\delta\tau_{ij}\right] \frac{\partial U_j}{\partial x_i} \tag{7.8}$$

The default values of the involved empirical constants are as follows:

$\sigma_k = 1.0$, $\sigma_\epsilon = 1.3$, $C_{1\epsilon} = 1.44$, $C_{2\epsilon} = 1.92$, $C_\mu = 0.09$

The concentration equation for a dissolved pollutant in water is

$$\frac{\partial C}{\delta t} + \frac{\partial U_i C}{\partial x_i} = D \frac{\partial}{\partial x_i}\left(\frac{\partial C}{\partial x_i}\right) \tag{7.9}$$

where C is the concentration of the dissolved pollutant and D is the diffusion coefficient assumed constant.

7.4.3 BOUNDARY CONDITIONS

A proper boundary condition is necessary to solve the problem with the help of equation 1 to 9. There are two types of inlet surface. The first is water inlet, and the second is pollutant inlet. There is only one outlet. A three-phase system is considered for this problem, and each phase is defined. The water inlet and outlet boundary condition are considered pressure inlet and pressure outlet, respectively. But the pollutant inlet is considered as velocity inlet. The density of water is considered 1000 kg/m^3, and the density of pollutant is considered 670 kg/m^3. The density of pollutant water is less as compared to clean water.

The boundary conditions of k and ϵ, at the channel inlet, correspond to those found in Buil's experiment and are given as follows:

$$k_0 = 0.002.u_0^2 \tag{7.10}$$

$$\epsilon_0 = \frac{k^{3/2}}{0.3\frac{4HL}{(H+L)}} \tag{7.11}$$

where; L and H indicate the channel length and height, respectively.

7.5 DATA USED AND DATA PROCESSING

ANSYS Fluent was used for pollutant dynamics in ANSYS Workbench. First, to generate a fluent fluid flow analysis system in the ANSYS Workbench study area was selected for the model geometry development. The geometry was kept precisely similar to the actual condition, i.e. identical to the geometry stretch of river Ganga considered in this study. The geometry was created in ANSYS Design Modeler (SpaceClaim editor), as shown in Figure 7.2. The geometrical model of the river Ganga is drawn in ANSYS by scaling down the actual width and length of the river. The total length and width of the river in the geometry are 80 m and 0.6m, respectively. The width of Assi Nala is drawn as 0.5 m. Figure 7.2

shows the geometry of the river drawn in ANSYS.

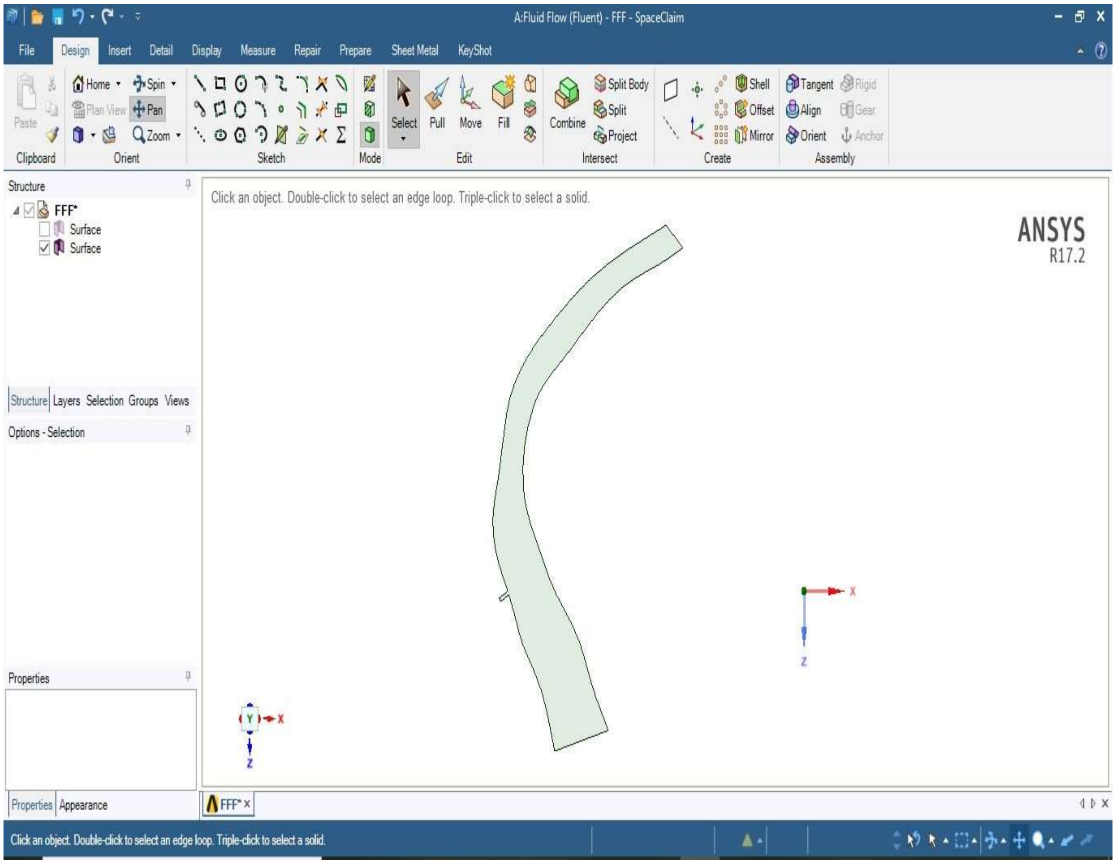

Figure 7.2 Geometry of river channel

After the development of the geometry file, the computational mesh was created for the geometry using ANSYS Meshing. Finite Element Analysis (FEA) is mainly used to calculate physical phenomena using different mathematical method. After creating meshing, it splits the domain in many numbers of the tiny elements. Figure 7.3 shows the meshing created in this study for the geometry constructed. These small elements contribute to solving complex problems. With the increase in the number of elements accuracy of results increases, but processing time also increases. The mesh generated in this study had a total 26,115 number of nodes, and the total number of elements were 130868.

Curvature selected for the size function, where the relevance centre was fine and smoothing was medium.

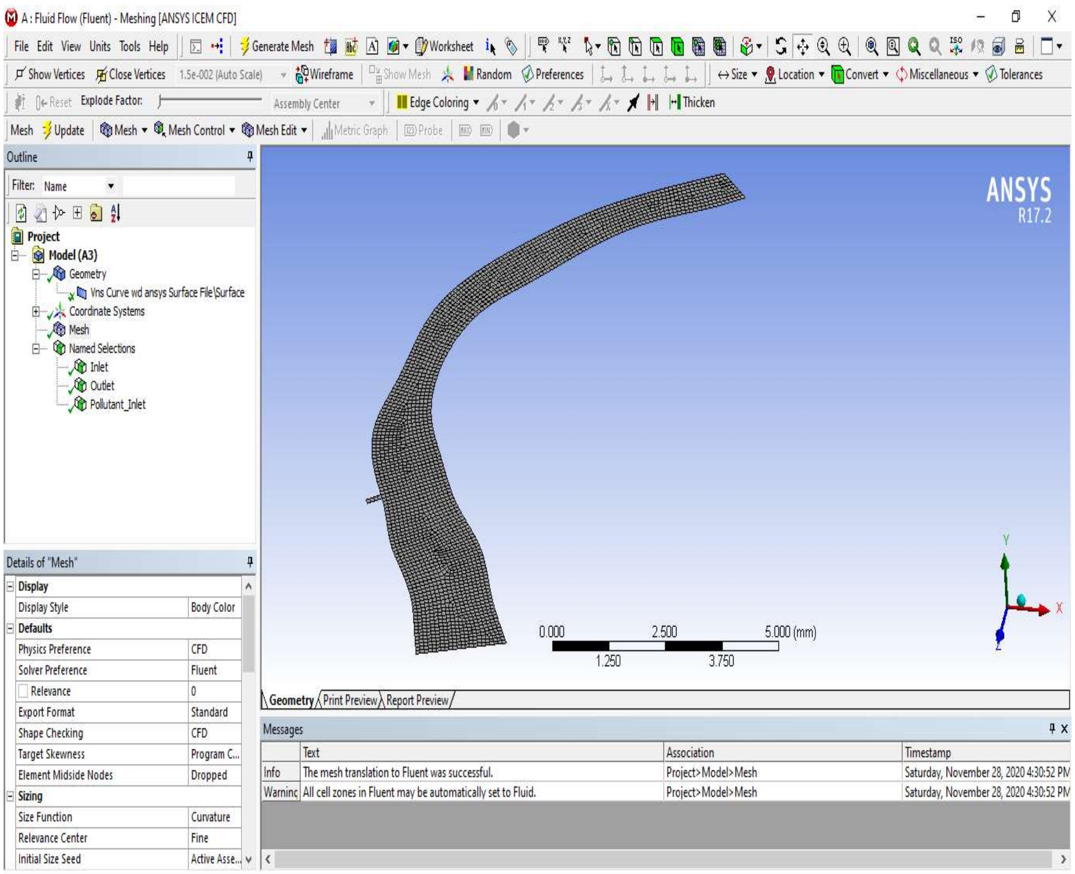

Figure 7.3 Mesh of channel

After generation of the mesh set up of the CFD simulation was created in Ansys workbench, as shown in Figure 7.3. There is a need for several parameters like general models, cell zone condition, boundary condition, dynamic mesh, and reference value for the setting up domain. Pressure–based solver, density-based solver and transient time selected in general parameter. A pressure-based solver applied for the incompressible flows and less compressible flows. A density-based solver is used for high-speed compressible flows. A pressure-based solver was selected for the modelling due to incompressible water flow in the river.

Four different options for modelling are available in ANSYS. The first one is Volume of Fluid (VOF), the second one is Mixture, the third one is Eulerian, and the fourth one is West Stream. But in the present study, VOF with the multiphase model was selected. The total Eulerian phase was three, the first phase was air, the second phase was water, and the third phase was pollutant (sewage water). The open channel model was selected in the VOF sub-model. It deals with free surface modelling, which is helpful for open channel modelling. It is based upon Eulerian methods. Hirt and Nichols (1981) proposed the VOF method for multiphase systems to simulate gas-liquid and liquid-liquid. Implicit was used as a volume fraction parameters. Here implicit body force used for the body force formulation. Figure 7.4 shows the processing of calculation in ANSYS of the model.

In the present study, turbulent flow condition was considered. K-epsilon (k-ε) turbulence model is widely used for mean flow characteristics of turbulent flow conditions in CFD. So, it was used for turbulent flow condition for river flow modelling. Three types of material were used in this model, namely air, water and pollutant. Property of the material entered on the basis of density of material in the database. There were four zones in the domain, and apply all boundary condition to the zones. All four zones were namely inlet water, outlet water, free surface and pollutant inlet. Pressure inlet was used for inlet water, pressure outlet for outlet water, pressure outlet for free surface and velocity inlet for the pollutant inlet. The selected inlet velocity of the pollutant was 0.3 m/s, and the density of sewage was 720 kilogram per cubic meter.

After the setup of the general setting, there is a need for selection for solution sets. There is two option available in solution initialization 1) Hybrid initialization and 2) Standard initialization. Hybrid initialization was selected for the solution. Hybrid initialization utilizes a given boundary condition and provides a solution with based upon the Euler problem. Calculate the result with the help of the run calculation tab. Total time step size 0.001 were selected with 500 number of time steps. The total numbers of iteration were 500 for calculation. After completion of the run calculation, we have plotted the result in the result tab of CFD fluent.

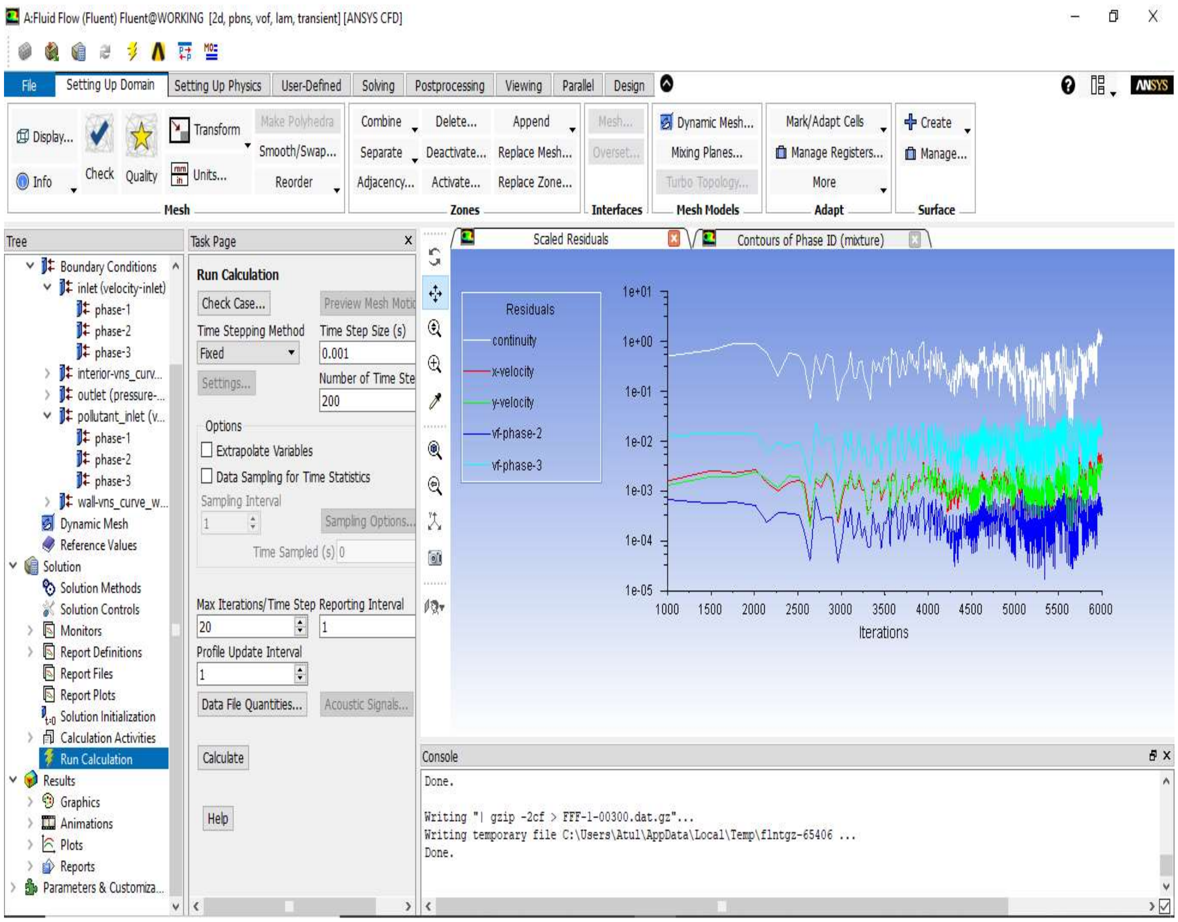

Figure 7.4 Calculation processing in CFD Fluent

7.6 RESULTS AND DISCUSSION

To study the flow of the pollutant in the river, the CFD model was prepared, and validation was done by a manual sampling process at different cross-sections, as discussed above. The ANSYS Fluent software package did the CFD model.

Figure 7.5 shows the diffusion behaviour of the pollutant obtained in the CFD model. Pollutants have started diffusing at the inlet level. The result shows that the pollutant's concentration is higher at a distance of 0.5-2 km from the inlet of the contaminant. The probable reason, the velocity at the inlet is higher than the velocity of the water in the channel. So, near to the inlet due to high-speed pollutant flows forward. Figure 7.6 shows the contour of the velocity. It shows that the velocity at the inlet is greater, while it is smaller at some distance from the pollutant inlet. Due to this, the contamination concentration at the inlet is lesser, while at some distance where the velocity is smaller, contamination is found higher.

Further, it can be observed that after a considerable distance impact of the pollutant become negligible. This happens due to the dilution of the pollutant. The concentration of the pollutant was also found maximum at the surface of the channel. Since the density of the pollutant is lower than the water, due to this reason, the concentration of the pollutant is higher at the surface. It can be further observed that the contamination covers 80% of the width of the channel at the section of highest contamination. Similarly, the length of the contamination zone is found approximately four times the width of the contamination. Since the direction of the flow of the contamination at the inlet is almost perpendicular to the river, the contamination area is significant.

The result obtained from the CFD model was compared with the results of the river. The volume fraction of the pollutant was compared for validation of the model. In the river, the contamination is more significant at a distance of about 2 km from the source of the contamination. This finding matches the model of the CFD. Similarly, the length of the contamination zone was found about four times the width of the contamination zone. This also supports the findings obtained from the CFD model. But further analysis is required by considering different parameters. The velocity streamline pattern is shown in Figure 7.7. It indicates the flow pattern of the river with the pollutant.

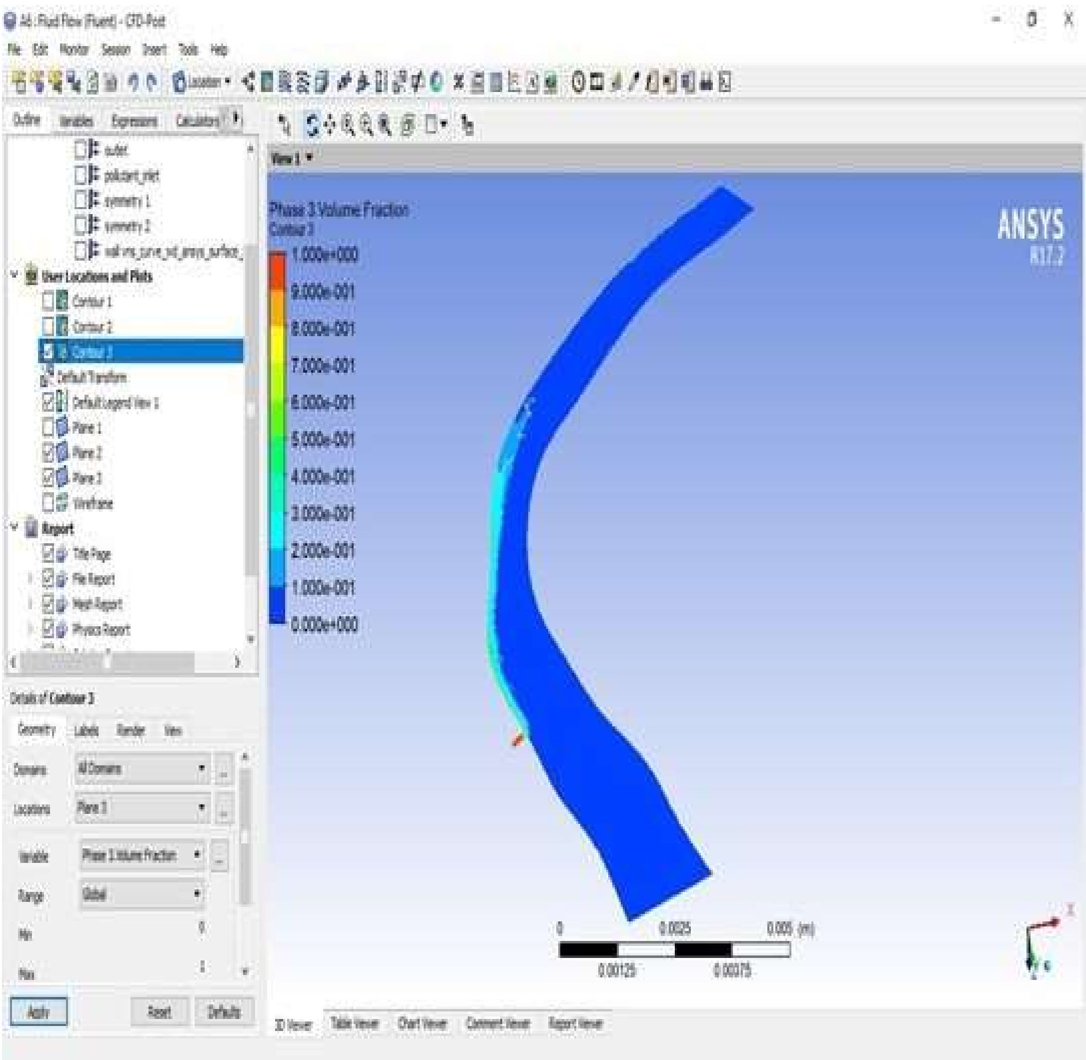

Figure 7.5 Contour profile of volume fraction

Ghats in the Varanasi is on the left bank of the river. Pilgrims take a bath in the water of Ganga at the Ghats. The pollutant from the Assi Nallah passes through the left bank, which can affect the pilgrims. Proper operation of the sewage treatment plant or discharge of such polluted water on the other side of the river could be the possible solutions.

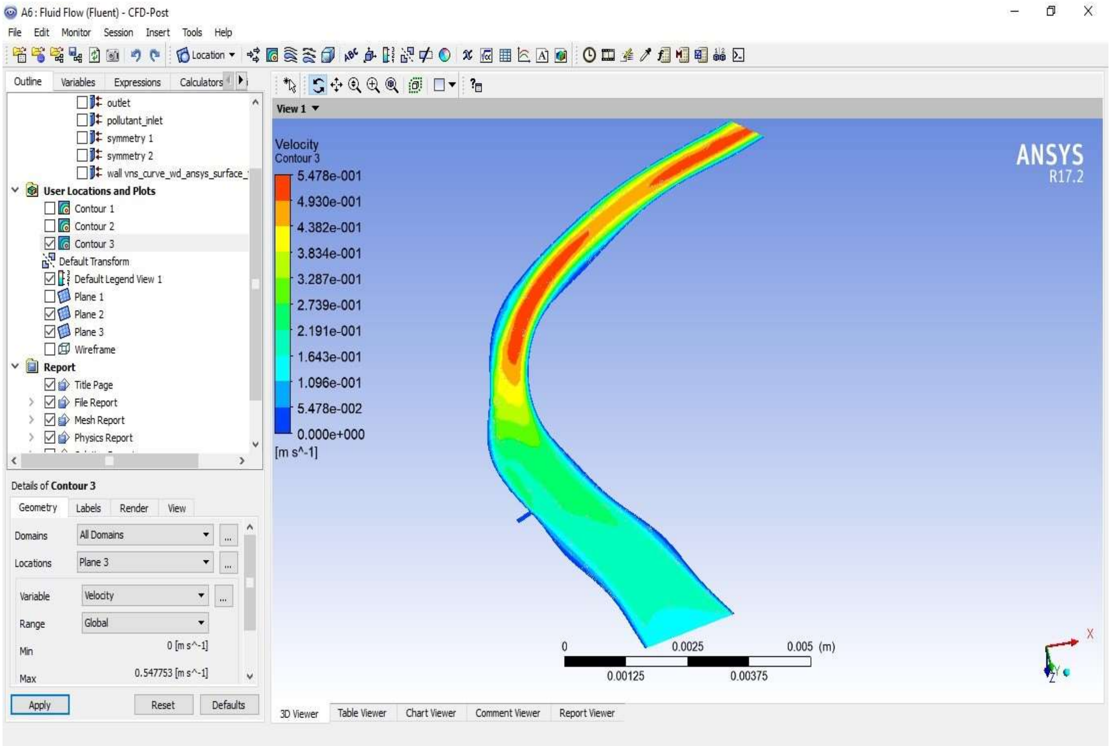

Figure 7.6 Isovels of river

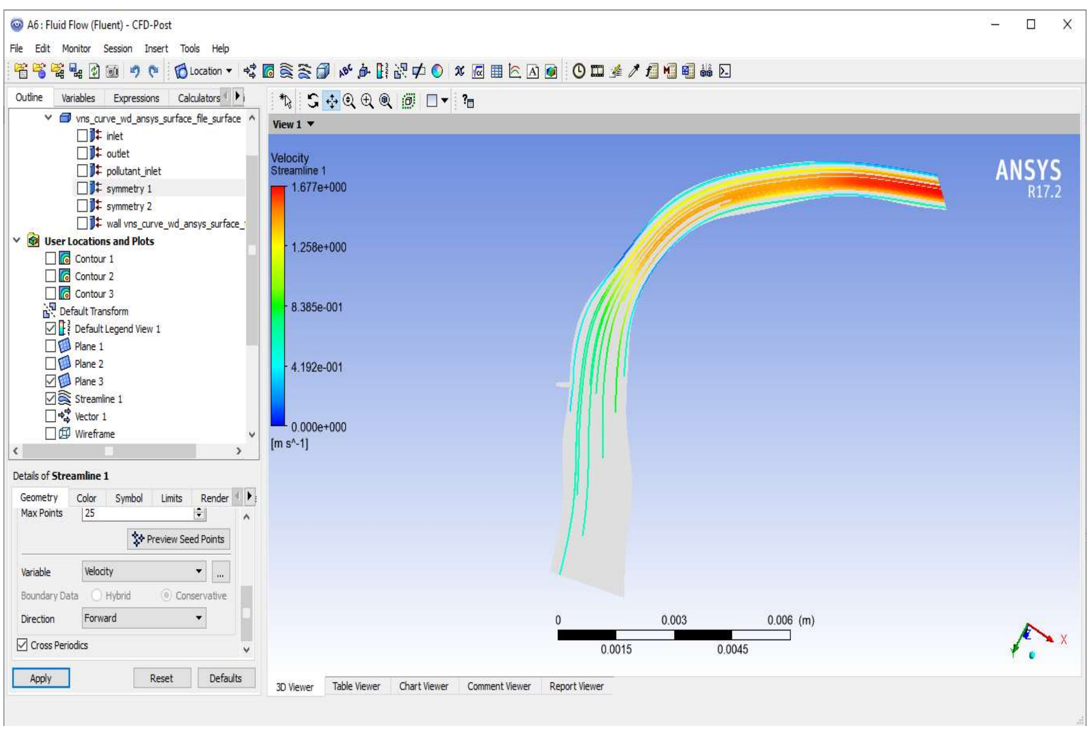

Figure 7.7 Velocity Streamline Pattern

7.7 VALIDATION

Validation of the output obtained from the modelling through ANSYS essential for confirmation of the methodology used and result obtained. In this study, validation was done by comparing the flow pattern obtained from the ANSYS modelling and the flow pattern of pollutant in the channel in the laboratory.

To obtain the flow pattern of the pollutant in the laboratory, a channel with an S-bend was used. The size of the channel used was 12 m x 0.5 m x 0.5 m with the central part of the channel S- bend with 180^{o} bend. The bed of the channel was prepared by the white cement collected from the river. Dye was used as a pollutant for testing. Dye was inserted in the water of the channel through the nozzle. A photographic view of the channel used in this study for validation is presented in Figure 7.8. The arrangement of the nozzle with dye is presented in Figure 7.9. The colour of the dye used was Blue.

Figure 7.8 Validation of Result in S- shape Laboratory Channel

Figure 7.9 Pollutant Flow Pattern in Lab Channel

(a) CFD result (b) Lab channel result

Figure 7.10 Comparison of CFD and laboratory channel result

Figure 7.10 shows the results obtained from the CFD modelling of ANSYS and the laboratory flume. In both cases, the pollutant was discharged on the left side of the flow in the concave bank. By observing the flow pattern of the pollutant, it can be seen that in both cases, the pollutant flows in the left bank (concave bank) of the river. This shows that CFD modelling of pollutant fluid dynamics have done with the help of ANSYS gives the precise result of the flow pattern.

7.8 CONCLUSION

Rivers are the main sources of fresh surface water systems that are important for human life and aquatic plants and animals. Assi river is one of the tributaries of River Ganga confluence at Varanashi, which has become a sewage drain due to the high rate of discharge of pollutants. Varanasi city located concave bank (Outer bank) of river Ganga and selected study area consisted of a 10-kilometre river section. The CFD model developed to study the flow dynamics of pollutant in the river. CFD mathematical model is based on momentum equations regulating the flow of fluid in all three different directions, i.e. vertical, longitudinal, and lateral mathematical processes jointly the continuity equation. CFD model was solved for Navier-Stokes equations with the help of k-epsilon turbulence resolution. The volume of fluid (VOF) model was used in this CFD program. The Eulerian free surface flow was used for model development. The model was tested with a meandering river cross-section. Pollutant flow dynamics have been tested for the best fissile location of this type of sewage drains for less impact on Varanasi ghats. The model was validated with a laboratory experiment. The laboratory experiment was conducted in the S-band channel, and the source of pollutant for this is the blue dye.

CFD is a powerful tool to analyse the river's pollutant flow dynamics pattern and article dispersion pattern. In this model, flow dynamics of pollutant show that the connection of this type of drain contributes to more pollution in the outer bank as the connection of sewage drain is more fissile in the inner bank resulting in less mixing of pollutant. The results obtained from this study show that the significant concentration from the pollutant point source is up to 2 km. The CFD model developed, and the data observed was in harmony. The CFD model was significantly able to predict the pollutants dynamics.