# CHAPTER 4

# PRECIPITATION: THE WASHING AND CONCENTRATION OF MAGNETITE PARTICLES USING SEDIMENTATION FUNNELS

## 1. INTRODUCTION

As discussed in Chapter 2, a precipitate of magnetite in an aqueous salt solution is produced during the preparation of ferrofluid. For water-based ferrofluid production, concentration of the magnetite suspension is essential for the production of a fluid of suitable magnetic properties. Washing is also important to ensure stability of the fluid. Although concentration of the magnetite suspension is not vital in the production of hydrocarbon solvent-based ferrofluid (the magnetite is transferred to a specified volume of hydrocarbon carrier liquid), washing the precipitate will very likely improve the quality of the hydrocarbon solvent-based ferrofluid. The salts in solution may adversely affect the production process, hinder the final organic/aqueous phase separation and may increase the viscosity of the product. It was therefore decided to investigate a method for the washing and concentration of the magnetite particles.

Washing and concentrating the ultrafine magnetite particles is extremely difficult. By virtue of their size, the particles cannot be trapped in filter media. In addition, because of their size, they are colloidally stable and settle with difficulty. There is currently no commercially available system to perform this function on production scale. Washing of precipitate in ferrofluid manufacture is performed only on a laboratory scale and is a manual and laborious process.

To achieve the concentration and washing of the magnetite, a concept was developed which involves four cone-shaped vessels in series. These vessels were called sedimentation funnels which are plastic containers of approximately 20 litre volumetric capacity around which copper coils are wound. A direct current is passed through the coils thus generating a magnetic field in the interior of the funnel. The proposed layout of the funnels is given in Figure 4.1. [30]

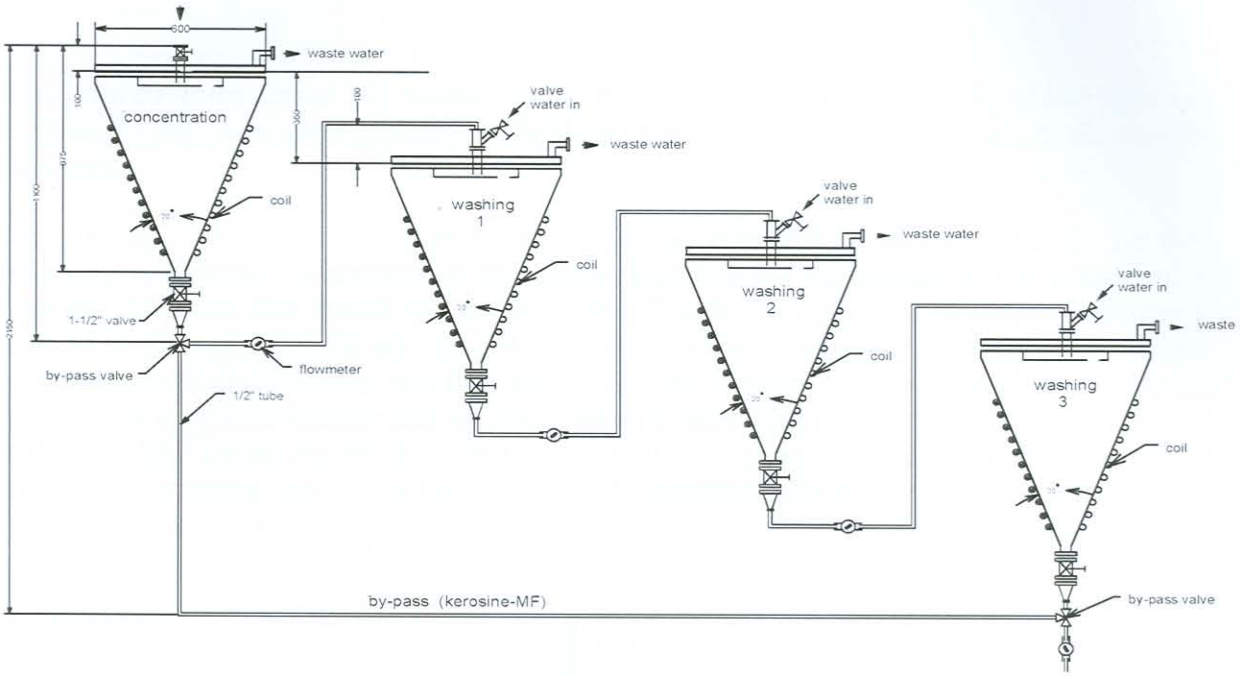


**Figure 4.1** Layout of sedimentation funnels

Preliminary modelling using 2D PC Opera (to display the magnetic field strength in one of the funnels) indicated that the magnetic field is strongest towards the base of the funnel as shown in Figure 4.2. [31] The values as given in Figure 4.2 are in Gauss or Oe.

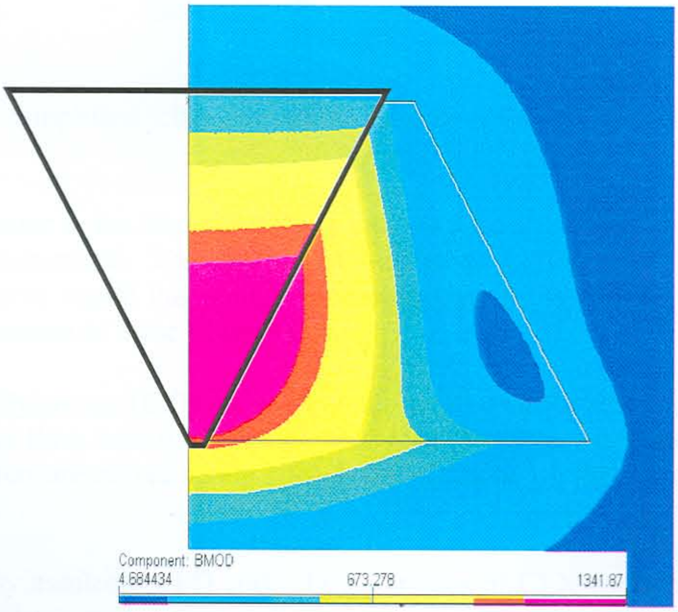


**Figure 4.2** Magnetic field strength in one of the sedimentation funnels

When placed in the funnel and exposed to the magnetic field, the magnetite should be attracted towards the region of greatest magnetic field and therefore become concentrated at the base of the funnel.

The aqueous salt solution containing the suspended magnetite particles is fed to the first sedimentation funnel. A volume of the carrier liquid and a percentage of the magnetite is removed from the first funnel via an overflow. This liquid reports to waste. In subsequent funnels, water is added to the mixture before its entry to these funnels. Depending on the volume of wastewater removed from the first funnel and the volume of water added to subsequent funnels, the magnetite fraction can be concentrated to the desired volume. The addition of water to the last three funnels serves to dilute the salts that are present in the salt solution. Wastewater leaves the remaining funnels via the overflow. A simplified schematic representation of one of the funnels is given in Figure 4.3.

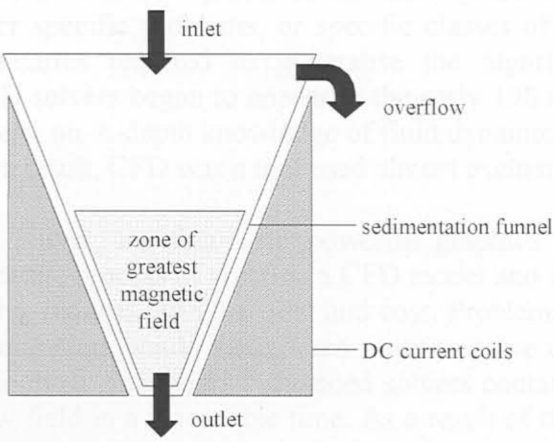


**Figure 4.3** Simplified schematic representation of the sedimentation funnel

The most important factor in the funnel design is that the total allowable loss of magnetite to the overflow should be minimised. The volume of magnetite reporting to the overflow will be affected by flow patterns inside the funnel. Modelling the flow of liquid through the funnels would provide an indication of these flow patterns.

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behaviour of systems involving fluid flow, heat transfer and other related physical processes. The equations of fluid flow over a region are solved using specified conditions on the boundary of the specific region of interest.

CFX is a commercially available CFD code. Two versions of CFX commonly in use are CFX-4.3 and CFX-5.4.1. These codes vary mainly in the method in which the geometry to be modelled is meshed. CFX-Build (the pre-processor) is used for geometry creation and meshing while CFX-Visualise or CFX-Analyse is used for visualisation of results (post-processing).

In the CFD study, initially two base case funnels consisting of only water flowing into the funnel were modelled using CFX-4.3. For comparison purposes, the same geometries were meshed in CFX-5.4 which makes use of a different method of meshing (this is explained in subsequent sections). Using multiphase flow, a solid phase was introduced into the funnel to obtain an idea as to the flow of magnetite particles and to determine whether they would be lost in the overflow. The final step would be to model the funnel with the effect of the magnetic field but this was not performed in this investigation.

# 2. COMPUTATIONAL FLUID DYNAMICS

## 2.1 The history of Computational Fluid Dynamics

Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problem. From the mid-1970s the complex mathematics required to generalise the algorithms began to be understood. General purpose CFD solvers began to appear in the early 1980s and required what were then very powerful computers, an in-depth knowledge of fluid dynamics and large amounts of time to set up simulations. As a result, CFD was a tool used almost exclusively in research.

Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models mean that the process of creating a CFD model and analysing the results is much less labour-intensive. This reduces both the time and cost. Problems that can be solved in a few seconds with current computers would have taken years to solve using computational methods and computers available thirty years ago. Advanced solvers contain algorithms which enable robust solution of the flow field in a reasonable time. As a result of these factors, CFD is now an established industrial design tool, helping to reduce design timescales and improving processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing, with variations on the simulation being performed quickly. CFD has become an important tool for use in experiment and pure theory. Certain problems with complex physics can be addressed using this technique as long as there is a governing equation describing the physics that can be solved numerically. [32]

## 2.2 The mathematics of CFD

CFD is the numerical solution of the fluid flow of partial differential equations describing certain models. The equations which describe the processes of mass, momentum and heat transfer are known as the Navier-Stokes equations. These are partial differential equations which were derived in the early nineteenth century. They have no known general analytical solution but can be discretised and solved numerically. The following equations give the Navier-Stokes equations for the conservation of mass (eq. (4.1)), momentum (eq. (4.2)) and energy (eq. (4.3)) respectively. [32]

$$\frac{D\rho}{Dt} + \rho\vec{\nabla} \bullet \mathbf{v} = 0 \tag{4.1}$$

$$\rho\frac{D\mathbf{v}}{Dt} = -\vec{\nabla}P + \rho\mathbf{g} + \mu\vec{\nabla}^2\mathbf{v} \tag{4.2 a}$$

In expanded form, (4.2 a) becomes:

$$\begin{aligned}
\rho(\frac{\partial v_x}{\partial t} + v_x\frac{\partial v_x}{\partial x} + v_y\frac{\partial v_x}{\partial y} + v_z\frac{\partial v_x}{\partial z}) &= -\frac{\partial P}{\partial x} + \rho g_x + \mu(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2} + \frac{\partial^2 v_x}{\partial z^2}) \\
\rho(\frac{\partial v_y}{\partial t} + v_x\frac{\partial v_y}{\partial x} + v_y\frac{\partial v_y}{\partial y} + v_z\frac{\partial v_y}{\partial z}) &= -\frac{\partial P}{\partial y} + \rho g_y + \mu(\frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2} + \frac{\partial^2 v_y}{\partial z^2}) \\
\rho(\frac{\partial v_z}{\partial t} + v_x\frac{\partial v_z}{\partial x} + v_y\frac{\partial v_z}{\partial y} + v_z\frac{\partial v_z}{\partial z}) &= -\frac{\partial P}{\partial z} + \rho g_z + \mu(\frac{\partial^2 v_z}{\partial x^2} + \frac{\partial^2 v_z}{\partial y^2} + \frac{\partial^2 v_z}{\partial z^2})
\end{aligned} \tag{4.2 b}$$

$$\frac{\partial}{\partial t}(\rho E) + \vec{\nabla} \bullet (\rho\mathbf{v}E) = \vec{\nabla} \bullet (k\vec{\nabla}T) + \vec{\nabla} \bullet (\bar{\bar{\sigma}} \bullet \mathbf{v}) + W_f + q_H \tag{4.3}$$

There are a number of different solution methods which are used in CFD codes. The most common is known as the finite volume technique. In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discretised and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one derives a full picture of the behaviour of the flow. [32]

The following equations give an example of the discretisation of the first and second derivatives where $O$ indicates the accuracy of the approximation. [32]

$$\begin{aligned}
\frac{\partial f}{\partial x} &= \frac{f(x+\Delta x) - f(x-\Delta x)}{2\Delta x} + O(\Delta x)^2 = \frac{f_{i+1} - f_{i-1}}{2\Delta x} + O(\Delta x)^2 \\
\frac{\partial^2 f}{\partial x^2} &= \frac{f(x+\Delta x) - 2f(x) + f(x-\Delta x)}{\Delta x^2} + O(\Delta x)^2 = \frac{f_{i+1} - 2f_i + f_{i-1}}{\Delta x^2} + O(\Delta x)^2
\end{aligned} \tag{4.4}$$

## 2.3 Uses of CFD

CFD is used by engineers and scientists in a wide range of fields. Typical applications include:

- Process industry e.g. mixing vessels, chemical reactors
- Building services e.g. ventilation of buildings, such as atria
- Health and safety e.g. investigating the effects of fire and smoke
- Motor industry e.g. combustion modelling, car aerodynamics
- Electronics e.g. heat transfer within and around circuit boards
- Environmental e.g. dispersion of pollutants in air or water
- Power and energy e.g. optimisation of combustion processes

- Medical e.g. blood flow through grafted blood vessels

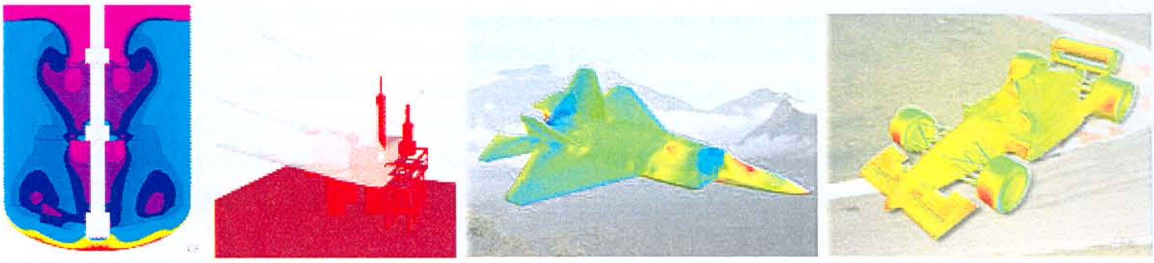

**Figure 4.4** Applications of CFD to a stirred tank, to investigate the effects of fire and smoke on an oilrig and in the areas of aeronautical and automotive engineering

## 2.4 Performing a CFD simulation

In many commercial CFD packages, the process of performing a CFD simulation is split into three stages: the pre-processing, the solving and the post-processing.

### *2.4.1 The Pre-processor*

The pre-processor is the component used to create the input for the solver. Pre-processing involves:

- Defining the geometry of the region of interest
- Selecting the physical models which are to be included in the simulation
- Specifying the properties of the fluid
- Specifying the boundary conditions
- Creating a mesh of control volumes

Pre-processing operations are interactive but are becoming increasingly automated e.g. in some commercial CFD packages, geometry can be imported from a CAD package and the mesh of control volumes is generated automatically. [33]

### *2.4.2 The Solver*

The solver is the non-interactive component which solves the CFD problem and produces the results. The solver proceeds as follows:

- The partial differential equations are integrated over all the control volumes in the region of interest. This is equivalent to applying a basic conservation law (e.g. for mass or momentum) to each control volume.
- These integral equations are converted to a system of algebraic equations by generating a set of approximations for the terms in the integral equations.

- The algebraic equations are solved iteratively. An iterative approach is required because of the non-linear nature of the equations. As the solution approaches the exact solution it is said to converge. For each iteration, an error, or residual, is reported as a measure of the overall conservation of the flow properties. How close the final solution is to the exact solution depends on a number of factors, including the size and shape of the control volumes and the size of the final residuals. Complex physical processes, such as combustion and turbulence are often modelled using empirical relationships, and the approximations inherent in these models also contribute to differences between the CFD solution and the real flow.
- The solver produces a file of results which is then passed to the post-processor. [33]

### *2.4.3 The Post-processor*

The post-processor is the interactive component used to analyse and present the results. Post-processing includes anything from obtaining point values to complex animated sequences.

Examples of some important features of post-processors are:

- Visualisation of the geometry and control volumes
- Vector plots showing the direction and magnitude of the flow
- Visualisation of the variation of scalar variables (such as temperature) through the domain
- Quantitative numerical output
- Hardcopy output [33]

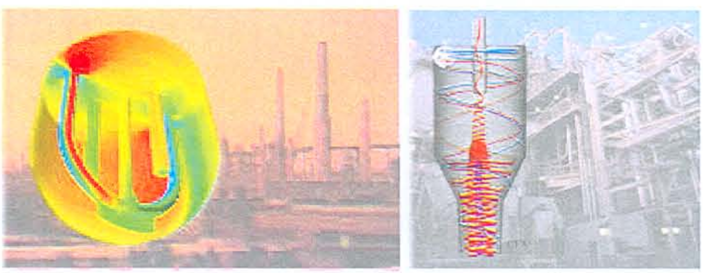

**Figure 4.5** Visualisation of CFD modelling using the post-processor

# 3. APPLICATION OF THE CFD CODE

## 3.1 CFX-4.3

This section describes the modelling of the sedimentation funnels performed in CFX-4.

### *3.1.1 CFX-Build*

CFX-Build, the pre-processor of CFX, is the main user interface module. It is an interactive program used to specify the CFD problem for input to the Solver. In CFX-Build, a parameterised model geometry must be created, meaning that when the meshlines are constructed on the geometry they "match up" and create a four-sided element. Figure 4.6 shows an example of a simple circle with unparametrised geometry. This geometry cannot be meshed as the corner elements would only have three sides. The addition of the central element (highlighted in the figure on the right in Figure 4.6) allows the geometry to be parametrised.

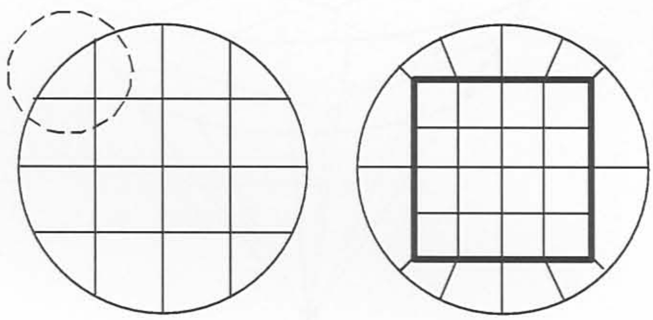

**Figure 4.6** Unparametrised circle which can be parametrised with the addition of a square element inside the circle

In CFX-4, structured meshes are used to mesh geometries. The structure refers to an ordered arrangement of the grid cells in the physical domain. Structured meshes consist of rectangular-shaped elements in 2D and cubic-shaped meshes in 3D. In CFX, the user performs the surface meshing on the geometry while the pre-processor performs the volume meshing.

Figure 4.7 gives a schematic of the initial geometry (Geometry 1) of the sedimentation funnel. The funnel cone was 400 mm high with a 450 mm diameter base and inlet, outlet and overflow diameters of 12.7 mm diameter. The overflow is positioned 100 mm from the cone base.

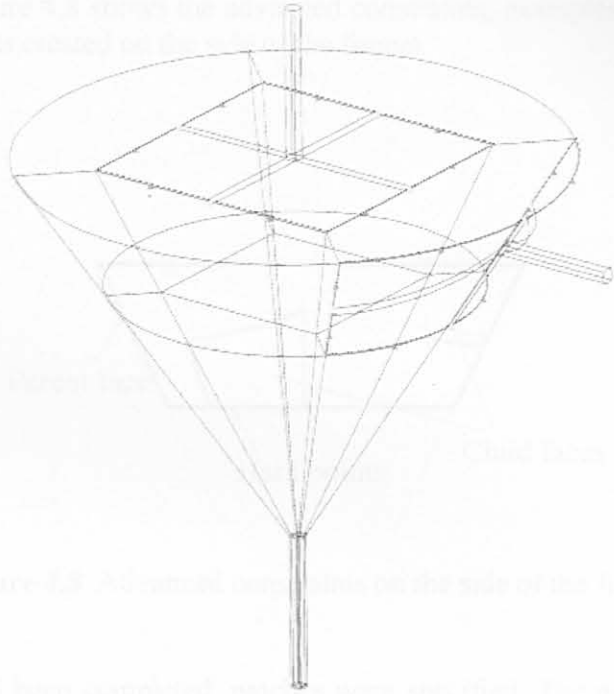

**Figure 4.7** Schematic of the initial geometry

The geometry was created in a number of stages. The outlet pipe was created from a square surface inside a circle. This surface was extruded to form the outlet pipe and then further extruded to form the body of the funnel. The reason that this inner rectangular pipe is required is so that the geometry can be topologically correct. The inlet pipe was created by transforming the outlet pipe. Once the inlet pipe was attached to the funnel upper surface, additional surfaces could be created to ensure the geometry was topologically correct. To complete the geometry, a surface from the outlet was transformed and then projected onto the main body. The transformed and the projected surfaces were joined.

In CFX-4.3, if a smaller face, known as the child face, is attached to a larger face known as the parent face, constraints must be used to define this relationship. For the funnel geometry, constraints were required as more than one face from the inlet and overflow pipes constrained the surface face.

To create the constraints, a set of subordinate surfaces completely covering the parent face were created. This meant that areas on the parent face were defined as surfaces. What are termed hard curves and hard points were then created at the intersection of the edge of each subordinate surface and the parent face. These hard points are used by CFX-Build to describe the geometric association between two curves which are coincident but of different lengths. The hard points force a mesh node to occur at each hard point when the surfaces are meshed. (During the meshing which is described below, it is required, when using constraints, to firstly mesh all the subordinate surfaces, then the parent faces and then the other surfaces in the model.) For the funnel application, Figure 4.8 shows the advanced constraints, examples of the child and parent faces and the hard points created on the side of the funnel.

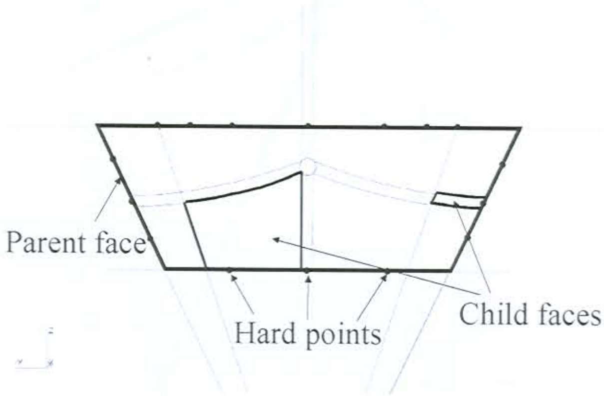


**Figure 4.8** Advanced constraints on the side of the funnel

Once the geometry had been completed, patches were specified. The patches are used to name and to specify the location of any surface on which to apply boundary conditions e.g. an inlet, mass flow boundary and solid region. The location, name and type of patch is specified in CFX-Build. In the CFX-4 command file, the boundary conditions e.g. velocity at an inlet applied to each patch are specified. Patches are applied to whole surfaces or whole geometric solids to ensure that they are grid-independent. Patches were specified for the inlet, outlet and overflow. The remainder of the surfaces can either be specified, or the default is taken as a wall.

Once patches have been specified, the geometry can be meshed. CFX-4.3 uses the multi-block grid methodology. The method involves the use of an unstructured set of blocks which are attached to one another. On each of these blocks there is a structured grid. Meshes are prepared in two stages. Firstly, CFX-Build is used to generate surface meshes. This is an interactive process. (This is where, for example, mesh seeds can be specified and the constraint surfaces meshed.) The non-interactive program VOLMSH is then used to generate grids in 3D regions. For the funnel application, mesh seeds were used as shown in Figure 4.9 for various parts of the geometry. Mesh seeds were used to prevent too many cells from being formed, to ensure that the aspect ratio (cell height to width) of cells is relatively uniform (as close as possible to 1) and to ensure that the mesh is created correctly.

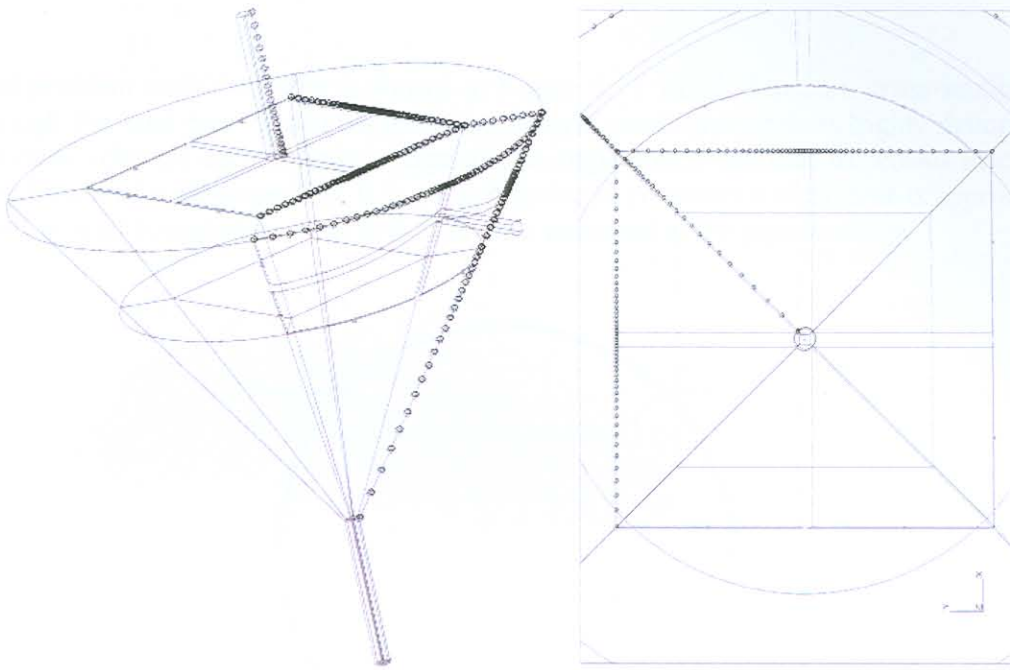

**Figure 4.9** Mesh seeds in the geometry

Once Geometry 1 had been meshed, the limitations of using structured meshing became apparent. Because of the ratio of the outlet pipe to the funnel size, the mesh became very fine towards the base of the funnel as the cells were propagated downwards. This effect is shown in Figure 4.10. This results in an extremely fine mesh in relation to other areas of the geometry. If the coarser areas of the mesh at the top of the funnel were to be refined, this would result in a large number of cells with an over refinement in areas where it would not be required. (The aspect ratio is not close to 1.)

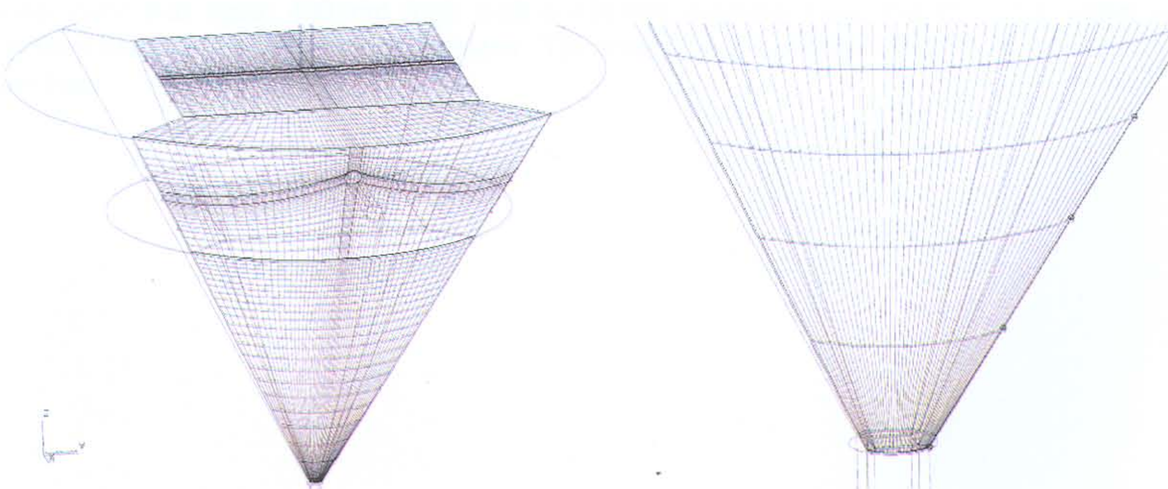

**Figure 4.10** Poor mesh created at base of the funnel

A second problem with the mesh is shown in Figure 4.11 which depicts a cross-sectional grid slice through the inlet pipe. It can be seen that the grid inside the circle is highly distorted. The corner control volumes each contain a large obtuse angle which can lead to serious inaccuracies in the prediction of wall properties. It is good practice to construct a mesh that is approximately orthogonal at wall boundaries. (This distortion also occurred in the pipe overflow.)

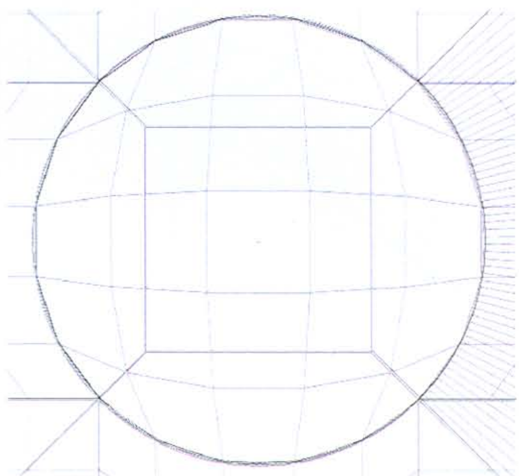

**Figure 4.11** Obtuse angles created inside the inlet and overflow pipes

The results obtained from such a grid may not be accurate. It was decided to investigate this geometry using larger pipe diameters. (The original geometry is investigated later in this chapter using unstructured grids in CFX-5.3.)

A schematic of the new geometry (Geometry 2) is given in Figure 4.12. In this configuration, the funnel cone was again 400 mm high with a 450 mm diameter base, but the inlet, outlet and overflow diameters are 75 mm in diameter. The overflow is again positioned 100 mm from the cone base.

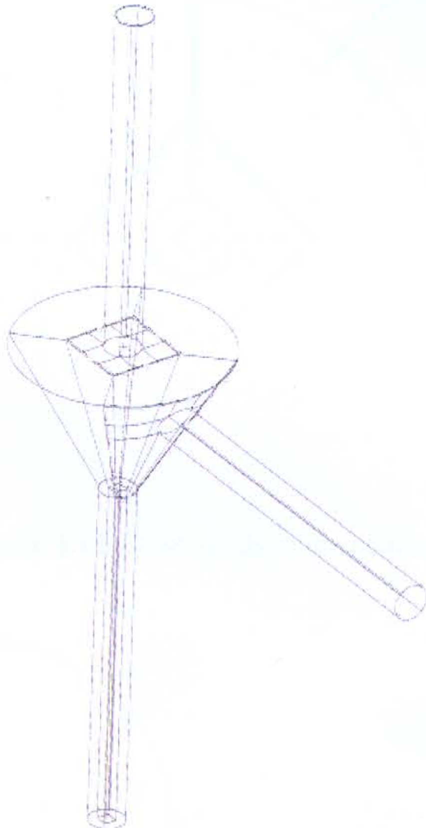

**Figure 4.12** Schematic of the second geometry dimensions

The geometry was constructed in a similar manner, constraints and patches were specified, mesh seeds created (see Figure 4.13) and the geometry meshed. The mesh appeared improved at the areas of concern as shown in Figure 4.14 although there is still some distortion on the circular inlet.

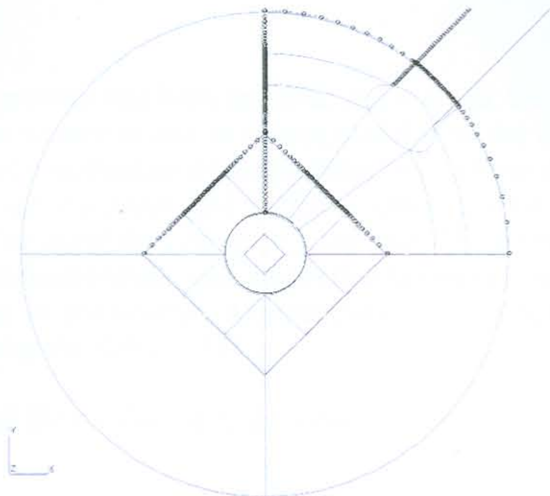

**Figure 4.13** Mesh seeds in the geometry

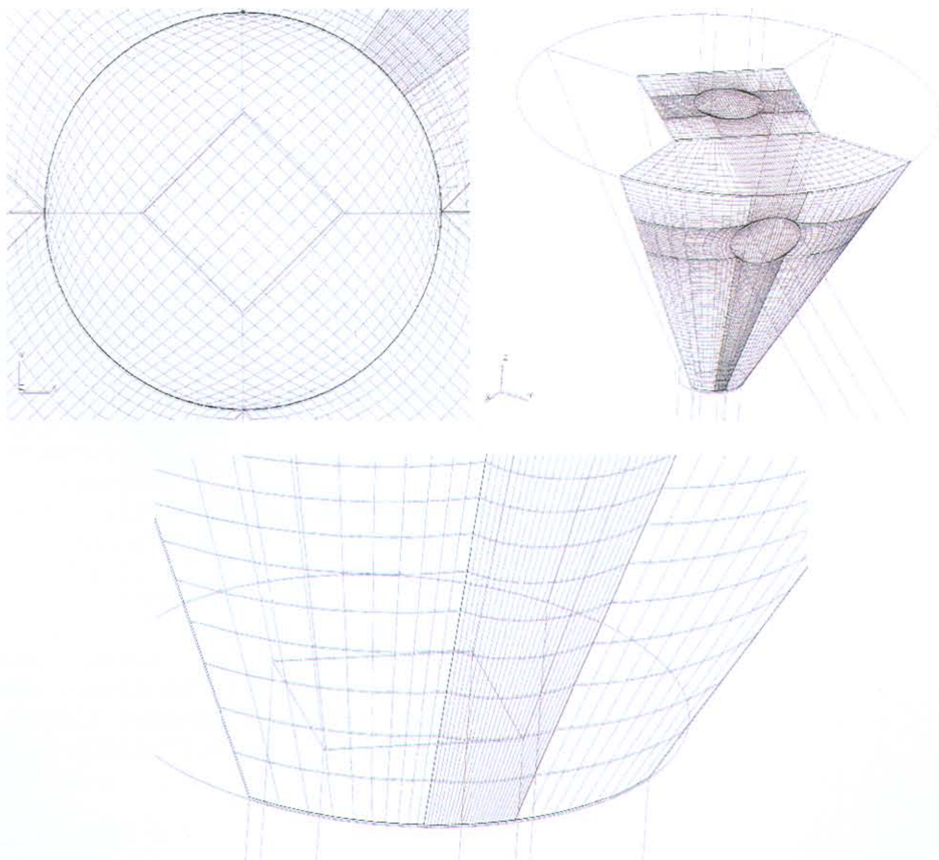

**Figure 4.14** Improved mesh of Geometry 2

### *3.1.2 Solver*

Once the geometry of the problem has been specified and meshed, the details of the topology and the grid coordinates can be written to disk in a form readable by the Frontend of the Solver. The Frontend converts the input specification of the problem from a form convenient for the user into a form designed for efficient execution and performs detailed error checking. The problem is specified in a single data file using the Command Language. The Command Language is a set of English-like commands, subcommands, and associated keywords. In the Interactive Frontend, CFX-Setup, this data file is constructed automatically via a graphical user interface. This command file is then used by the Solver. [33]

An example of a Command file for Geometry 2 follows:

```
>>CFX4
  >>OPTIONS
    THREE DIMENSIONS
    BODY FITTED GRID
    CARTESIAN COORDINATES
    TURBULENT FLOW
    ISOTHERMAL FLOW
    INCOMPRESSIBLE FLOW
    STEADY STATE
    USE DATABASE
>>MODEL DATA
  >>MATERIALS DATABASE
    >>SOURCE OF DATA
      PCP
    >>FLUID DATA
      FLUID 'WATER'
      MATERIAL TEMPERATURE 2.7300E+02
      MATERIAL PHASE 'LIQUID'
  >>TITLE
    PROBLEM TITLE 'FUNNELEX4'
  >>PHYSICAL PROPERTIES
    >>STANDARD FLUID
      FLUID 'WATER'
      STANDARD FLUID REFERENCE TEMPERATURE 2.7300E+02
    >>FLUID PARAMETERS
      VISCOSITY 1.0000E-03
      DENSITY 1.0000E+03
>>SOLVER DATA
  >>PROGRAM CONTROL
    MAXIMUM NUMBER OF ITERATIONS 400
    MASS SOURCE TOLERANCE 1.0000E-03
  >>DEFERRED CORRECTION
    K START 0
    K END 200
    EPSILON START 0
    EPSILON END 200
>>MODEL BOUNDARY CONDITIONS
  >>INLET BOUNDARIES
    PATCH NAME 'INLET'
    NORMAL VELOCITY 1.0000E+00
    TURBULENCE INTENSITY 3.7000E-02
```

```
    DISSIPATION LENGTH SCALE 7.5000E-02
  >>PRESSURE BOUNDARIES
    PATCH NAME 'PRESS_OVERFLOW'
    PRESSURE 0.0000E+00
  >>PRESSURE BOUNDARIES
    PATCH NAME 'PRESS_OUTLET'
    PRESSURE 0.0000E+00
  >>WALL BOUNDARIES
    PATCH NAME 'WALL'
>>STOP
```

### *3.1.3 CFX-Analyse*

CFX-Analyse is used to produce the main graphics output. The Solver writes the results to disk files, and these are read by CFX-Analyse. Figure 4.15 shows the output in CFX-Analyse for vector plots of the flow. From this figure it can be seen that the flow entering the funnel may be detrimental as it may disturb the settling of the magnetite particles in the magnetic field.

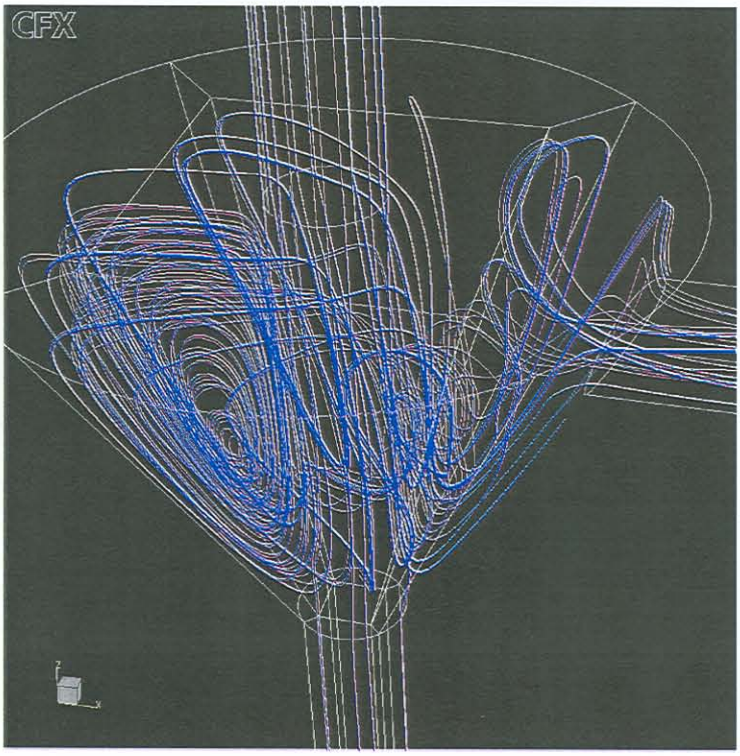


**Figure 4.15** Output in CFX-Analyse for vector plots of the flow

## 3.2 CFX-5.4.1

Because of the limitations in terms of meshing geometries in CFX-4, the CFD study of the sedimentation funnels was performed in CFX-5.

### *3.2.1 CFX-Build*

In CFX-5, unstructured meshes are used to mesh geometries. The unstructured meshes consist of triangular-shaped elements in 2D and tetrahedral-shaped elements in 3D. The advantages of using unstructured meshes are that complex shapes can be meshed relatively easily and the mesh can be easily adapted to give a good solution. CFX-5 is much simpler than CFX-4 in that it is not necessary to construct simple or advanced constraints. Because the mesh structure used by CFX-5 is unstructured, the definition of the position of mesh nodes is not necessary.

In CFX-Build, the mesh is prepared in two stages. Firstly, a surface mesh of triangular elements is generated and secondly a volume mesh of tetrahedral elements (also possibly using prismatic and pyramidal elements) is generated from the surface mesh during the creation of the CFX-5 Definition File. If tetrahedra are used, then a fine surface mesh may be required to avoid generating highly distorted tetrahedral elements at the surface. The mesher overcomes this problem by using prisms to create a mesh that is finely resolved normal to the wall, but coarse parallel to it. This allows the user to grow a series of prismatic elements from triangular elements at the surface to produce a good solution at the model wall where velocity gradients are large normal to the surface. This is termed inflation. This mesh arrangement is beneficial for cost effective analysis. The thickness of the inflation is controlled by the number of layers, the maximum thickness specification, the local element edge length and the inflation thickness multiplier. [33]

Mesh Controls are used to refine the surface and volume mesh in specific regions of the model. Mesh Controls can be:

- a point (region of influence defined by a sphere),
- line (region of influence defined by the cylindrical volume between two spheres),
- triangle (region of influence defined by the prismatic region between three spheres) or
- surface (region of influence defined by the prismatic region between four or more spheres).

Each type of control has certain attributes that should be specified: length scale, radius and expansion factor. [33]

Figures 4.16 and 4.17 show the solids and the meshes that were created for the two geometries in CFX-5. As can be seen from Figure 4.16, the problems that were encountered when structured meshes were used in CFX-4 have been eliminated. The transition from the large funnel to the small outlet pipe results in smooth and even elements being propagated into the outlet pipe. The outlet pipe mesh should be further refined to ensure that the outlet pipe is circular.

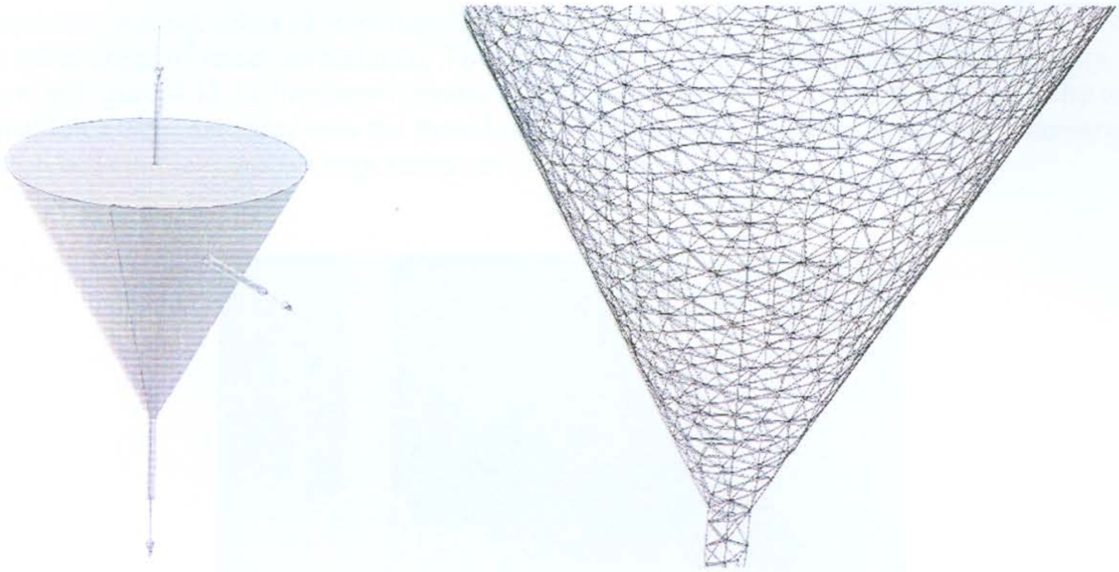

**Figure 4.16** Unstructured mesh of Geometry 1

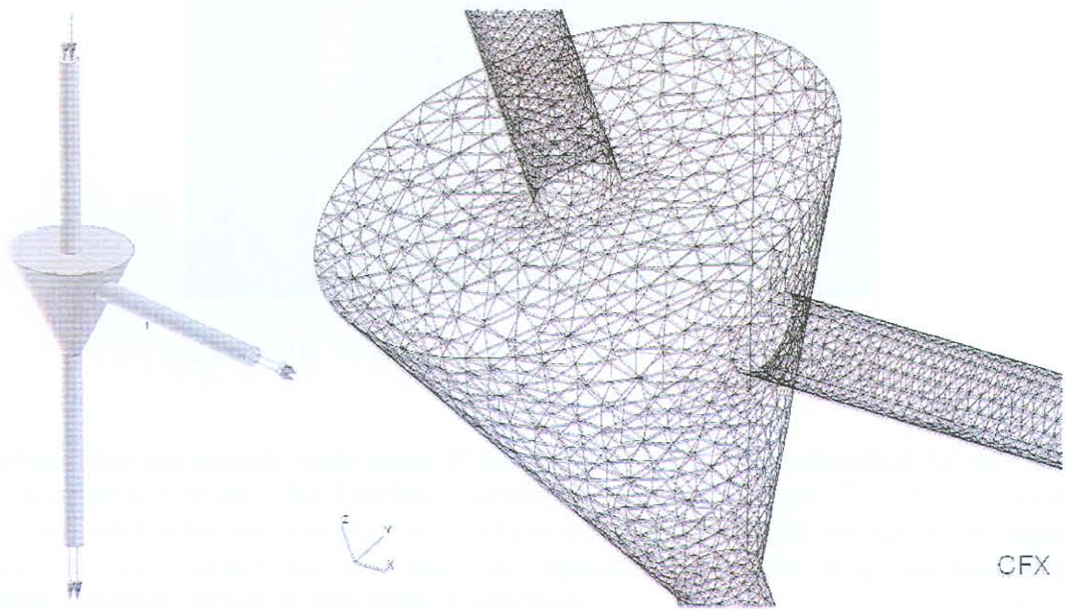


**Figure 4.17** Unstructured mesh of Geometry 2

A multi-phase flow case was studied using Geometry 2 in CFX-5.4.1. Initially, to obtain an indication as to whether convergence would be obtained for the multi-phase model, simple flow of particles in a pipe was modelled. Once convergence had been obtained for the multi-phase flow in a pipe, the parameters used for the pipe flow were used to solve the multi-phase flow through the funnel.

Using mesh control, areas of importance that were to be more defined were meshed more finely than other areas of lesser importance. The advantage of using mesh controls for Geometry 2 is shown in Figure 4.18. A line mesh control was used at the inlet, outlet and overflow. The mesh control lines were extended into the funnel to more clearly define these areas. The concentration of mesh cells in the region of importance are visible in Figure 4.18.

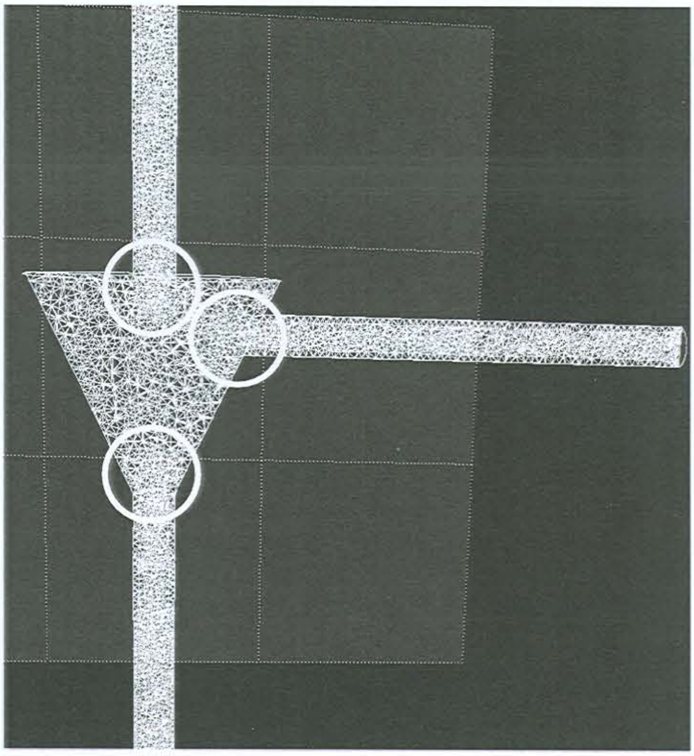

**Figure 4.18** Visualisation of mesh control for Geometry 2

The particle size and density were some of the parameters that were specified for the magnetite for the multi-phase model. The Schiller-Naumann model was selected. This model should only be used for solid spherical particles, or fluid particles that are small enough to be considered spherical as was assumed for the magnetite. Specification of the drag coefficient was an alternative selection, however, this value is unknown.

### *3.2.2 Solver*

The solver proceeds as for the CFX-4 solver. Residuals are plotted and the solution proceeds either until convergence is reached or stops if the solution becomes unstable.

### *3.2.3 CFX-Visualise*

Initially, obtaining convergence using the multi-phase flow proved to be difficult. The magnetite particle size was specified as 10 nm. Particles of such a size would not behave differently from the fluid in which they were contained. The correct solution to this problem would only be obtained once the magnetic field were incorporated into the model as the particles would then be acted upon by an additional external body force. To obtain an indication of the particle movement, it was decided to model the multi-phase flow using magnetite particles of 1 mm in size.

Figure 4.19 shows the water and magnetite velocities obtained using a mesh with 123 906 elements and mesh control. With an increase in the number of mesh cells, the solution obtained improves and so does the definition of the velocity plots. It is impractical, however, to increase the number of elements indefinitely as the computational time increases. A compromise must be reached between accuracy of solution and mesh refinement.



Figure 4.19: Flow visualisation for multi-phase flow in the funnel with mesh control

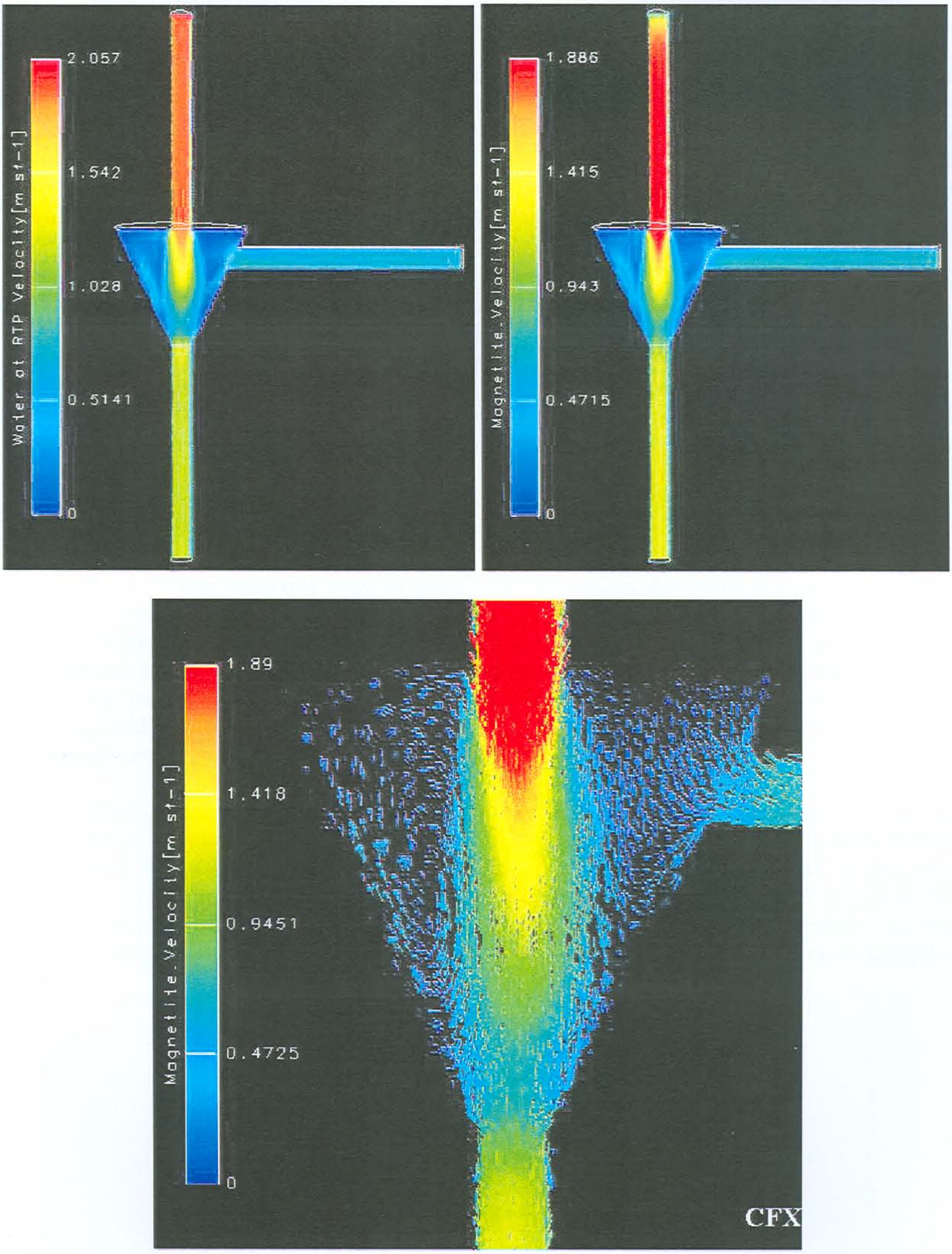


**Figure 4.19** Flow visualisation for multi-phase flow in the funnel with mesh control

The results obtained for the multi-phase flow problem in CFX-5 seem reasonable. The vector plots of velocity show the magnetite particles entering the funnel at a high speed and slowing down at the outlet. The vector plots again show, however, that the contents of the funnel are mixed by the inflowing liquid as was seen in CFX-4. It is noted that the magnetite velocity is lower than the water velocity. This may be owing to drag forces experienced by the particles (these results were obtained for the large particle size).

## 4. CONCLUSIONS AND RECOMMENDATIONS

In this CFD study, a base case funnel containing only water was modelled using CFX-4.3. It was found that because structured mesh creation requires a geometry that is topologically correct, this may complicate simple geometries. Modelling of complex geometries is difficult in CFX-4 as the creation of structured meshes is not easy. From preliminary results obtained from CFX-4, it appears that the inflowing liquid churns up material that may have settled at the base of the funnel. A possible method to eliminate this would be to insert a baffle plate at the funnel entrance.

In CFX-5.4.1, it was relatively easy to mesh both geometries that were attempted in CFX-4. Simple and complex geometries can be solved in CFX-5, largely owing to the method of mesh creation. Unstructured grids are more suitable for difficult geometries. The streamline plots of the solution obtained in CFX-5 confirmed the results obtained in CFX-4. It was realised, when performing the multi-phase flow modelling that it would be necessary to incorporate the magnetic field if the correct solid particle size is to be used. Without the additional body force, the nanometre size particles follow the flow of the main fluid.

It was initially recommended that the funnel geometry be modified such that the flow into the funnel does not disturb the particles at the base of the funnel. The magnetic field could then be incorporated into the model and the four funnels in series be modelled as a unit once convergence on an optimised funnel has been obtained. However, a new concept was proposed as an alternative to the sedimentation funnels and the CFD study was suspended. The new concept is called the wet high intensity magnetic separator (WHIMS) and will be discussed in Chapter 5.