Trabajo Final de Máster
Máster en Mecánica de Fluidos Computacional

Optimización del diseño de las estructuras de entrada y salida de un decantador secundario mediante simulaciones CFD.

Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

Presentado por: Gómez Rey, Alberto
Director/a: Martorell Masip, Benjamí
Fecha: 23 de Julio de 2019
Contents

1. Introduction: ...................................................................................................................... 8
1.1. Background.................................................................................................................... 8
1.2. Literature review.......................................................................................................... 9
1.3. Objectives.................................................................................................................... 10

2. Secondary clarification.................................................................................................. 11
2.1. Sedimentation theory.................................................................................................. 11
2.2. Solid flux theory........................................................................................................... 12
2.3. Clarifier design approach............................................................................................ 15
2.4. General characteristics and clarifier types.................................................................. 15

3. Sludge rheology.............................................................................................................. 17
3.1. Bingham model............................................................................................................ 17

4. Mathematical model of two phase flow....................................................................... 19
4.1. Drift-flux model.......................................................................................................... 20

5. Physical model and simulation set up........................................................................... 23
5.1. Secondary clarifier design.......................................................................................... 23
5.2. Experiment matrix for simulations.............................................................................. 25

6. Modelling........................................................................................................................ 27
6.1. Software..................................................................................................................... 27
6.2. Initial and Boundary conditions (B.C.) implementation.............................................. 27
6.3. Equations discretization schemes................................................................................. 31
6.4. Mesh implementation and mesh independence analysis.......................................... 31

7. Results and analysis...................................................................................................... 36
7.1. Cases were steady state was not reached:................................................................... 37
7.2. Steady state simulation results.................................................................................... 39
7.3. Sludge transport failure: hypotheses verifications.........................................................48

8. Summary and conclusions.............................................................................................55
8.1. Future work..................................................................................................................56

9. Bibliography:..................................................................................................................57

10. ANNEX I: Results..........................................................................................................59

11. ANNEX II: Mass balance calculation...........................................................................78

12. ANNEX III: Geometry and meshing example.............................................................79
# Table of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>12</td>
</tr>
<tr>
<td>2.2</td>
<td>13</td>
</tr>
<tr>
<td>2.3</td>
<td>14</td>
</tr>
<tr>
<td>2.4</td>
<td>14</td>
</tr>
<tr>
<td>2.5</td>
<td>16</td>
</tr>
<tr>
<td>3.1</td>
<td>17</td>
</tr>
<tr>
<td>3.2</td>
<td>18</td>
</tr>
<tr>
<td>5.1</td>
<td>25</td>
</tr>
<tr>
<td>5.2</td>
<td>25</td>
</tr>
<tr>
<td>6.1</td>
<td>28</td>
</tr>
<tr>
<td>6.2</td>
<td>29</td>
</tr>
<tr>
<td>6.3</td>
<td>32</td>
</tr>
<tr>
<td>6.4</td>
<td>33</td>
</tr>
<tr>
<td>6.5</td>
<td>34</td>
</tr>
<tr>
<td>6.6</td>
<td>34</td>
</tr>
<tr>
<td>6.7</td>
<td>35</td>
</tr>
<tr>
<td>6.8</td>
<td>35</td>
</tr>
<tr>
<td>7.1</td>
<td>37</td>
</tr>
<tr>
<td>7.2</td>
<td>38</td>
</tr>
<tr>
<td>7.3</td>
<td>39</td>
</tr>
<tr>
<td>7.4</td>
<td>40</td>
</tr>
<tr>
<td>7.5</td>
<td>41</td>
</tr>
</tbody>
</table>
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
<table>
<thead>
<tr>
<th>Figure Reference</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 7.31</td>
<td>54</td>
</tr>
<tr>
<td>Figure 7.32</td>
<td>54</td>
</tr>
<tr>
<td>Figure 7.33</td>
<td>54</td>
</tr>
<tr>
<td>Figure 7.34</td>
<td>54</td>
</tr>
<tr>
<td>Figure 10.1</td>
<td>59</td>
</tr>
<tr>
<td>Figure 10.2</td>
<td>59</td>
</tr>
<tr>
<td>Figure 10.3</td>
<td>60</td>
</tr>
<tr>
<td>Figure 10.4</td>
<td>60</td>
</tr>
<tr>
<td>Figure 10.5</td>
<td>61</td>
</tr>
<tr>
<td>Figure 10.6</td>
<td>61</td>
</tr>
<tr>
<td>Figure 10.7</td>
<td>62</td>
</tr>
<tr>
<td>Figure 10.8</td>
<td>62</td>
</tr>
<tr>
<td>Figure 10.9</td>
<td>63</td>
</tr>
<tr>
<td>Figure 10.10</td>
<td>63</td>
</tr>
<tr>
<td>Figure 10.11</td>
<td>64</td>
</tr>
<tr>
<td>Figure 10.12</td>
<td>64</td>
</tr>
<tr>
<td>Figure 10.13</td>
<td>65</td>
</tr>
<tr>
<td>Figure 10.14</td>
<td>65</td>
</tr>
<tr>
<td>Figure 10.15</td>
<td>66</td>
</tr>
<tr>
<td>Figure 10.16</td>
<td>66</td>
</tr>
<tr>
<td>Figure 10.17</td>
<td>67</td>
</tr>
<tr>
<td>Figure 10.18</td>
<td>67</td>
</tr>
<tr>
<td>Figure 10.19</td>
<td>68</td>
</tr>
<tr>
<td>Figure 10.20</td>
<td>68</td>
</tr>
<tr>
<td>Figure 10.21</td>
<td>69</td>
</tr>
</tbody>
</table>

Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
# Index of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 2.1</td>
<td>16</td>
</tr>
<tr>
<td>Table 5.1</td>
<td>24</td>
</tr>
<tr>
<td>Table 5.2</td>
<td>26</td>
</tr>
<tr>
<td>Table 6.1</td>
<td>30</td>
</tr>
<tr>
<td>Table 6.2</td>
<td>31</td>
</tr>
<tr>
<td>Table 6.3</td>
<td>34</td>
</tr>
<tr>
<td>Table 7.1</td>
<td>36</td>
</tr>
<tr>
<td>Table 7.2</td>
<td>49</td>
</tr>
<tr>
<td>Table 7.3</td>
<td>50</td>
</tr>
</tbody>
</table>

Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Resumen

El presente trabajo estudia la influencia del diseño de las estructuras de entrada del fango activo (campana deflectora) y salida de agua tratada (posición y forma del vertedero de salida), en el rendimiento de un decantador secundario para el tratamiento de aguas residuales. El estudio se ha llevado a cabo a través de simulaciones CFD con el software openFoam, en las que se han analizado diferentes combinaciones de estructuras de entrada y salida.

Los resultados obtenidos muestran una gran influencia de estas estructuras en los campos de velocidad y la distribución de sólidos dentro del decantador, influyendo por tanto de manera decisiva sobre el rendimiento del decantador, tanto en lo referente a la extracción de fangos, como en lo que se refiere al contenido de sólidos en suspensión del efluente. Así mismo, se ha observado una gran influencia de los parámetros reológicos del fango y su densidad sobre los campos de velocidad obtenidos en las simulaciones.

**Palabras Clave:** CFD, openFoam, Decantación secundaria

Abstract

The present work studies the influence of the design of active sludge inlet structure (influent well) and treated water outlet structure (position and shape of effluent weir), in the performance of a secondary clarifier for waste water treatment. The study has been carried out through CFD simulations with the openFoam software, in which different combinations of input and output structures have been analysed.

The obtained results show a great influence of these structures in the velocity fields and the solids distribution inside the clarifier. These structures have shown to be a key factor regarding clarifier performance, both in transport and extraction of sludge, as well as in suspended solids content in the effluent. Results have shown a great dependency of flow fields on sludge density and rheologic properties.

**Keywords:** CFD, openFoam, Secondary clarification
1. **Introduction:**

1.1. **Background**

The goal of water and waste-water treatment, is to remove the pollutants from water to achieve the optimal conditions required for the intended purposes or to be released to the environment with the minimum impact. A big percentage of the pollutants in water and waste-water (WW) are in form of suspended solids, or can be converted from dissolved state into suspended solids by the use of chemicals, or by micro-organisms action.

The removal of the suspended solids from water is based on one of the following physical principles: density differences or particle interception.

Based on density differences, two main processes are used: sedimentation/clarification and flotation. For particle interception two main techniques are used: surface filtration and depth filtration.

The unitary process of clarification consists on the separation of suspended solids from water by gravity differences. This operation is normally performed in open tanks were the water is introduced, usually in an continuous mode, and maintained in laminar flow condition, reducing its velocity in order to allow suspended solids going to the bottom of the tank. In many cases suspended solids concentration and removal is also an important performance feature of the system, as it influences not only clarification performance, but also downstream sludge treatment.

In water treatment engineering, clarification is a very common operation used in several treatment stages of the process [1]:

- Drinking water treatment: clarification is used to remove residual suspended solids usually with the previous addition of chemicals (coagulant and floculant).

- Primary treatment (WW): clarification is used to remove suspended solids in raw waste-water.

- Secondary treatment (WW): clarification is used to remove activated sludge from water, usually this is the last stage of waste-water treatment, previous to the release into water bodies.

- Tertiary treatment (WW): clarification can be used to remove residual suspended solids usually with the previous addition of chemicals (coagulant and floculant).
Due to its spread use in water and waste-water treatment, clarification performance is a key point for the fulfilment of effluent water quality standards, and its correct design and operation can condition the proper performance of all the water treatment plant.

The previous reason joined to the lack of a deep knowledge of the performance of this equipments has motivated the present work. The availability of a such a powerful tool, as CFD can contribute to a better understanding of the flow fields involved in clarification processes.

1.2. Literature review

Several authors have studied clarifier performance by the use of CFD [2]. Kerbs et al. [3] suggested some inlet structures to minimize density currents effects, these authors recommended to introduce the mixture sludge water close to the bottom, for secondary clarifiers. They also recommended to implement an extra element, as interception bars or concentric slotted baffles, to reduce the potential energy. Krebs [4] also suggested that the installation of a baffle vertically in the bottom, at the middle of the clarifier length, would improve clarifier performance. Weiss et al. [5] implemented an asymmetric clarification model in the commercial solver Fluent 6. This model was used to study the flow field inside a circular clarifier, with special focus on the prediction of the TSS profile versus heigh. Sludge withdrawal was done through almost all the bottom of the clarifier, simulating an hydraulic suction type system. Dahl et al. [6] also presented a mathematical model used to simulate the flow field and TSS profiles inside a secondary clarifier.

Brennan developed the code settlingFoam; this code is the predecessor of driftFluxFoam used in the present work. In his thesis [7] Brennan explained the equations that forms the mathematical model and also gave some details about solver algorithm. Apart from model details, the author studied the influence of the rheological properties on the CFD simulations, and compared the simulation results with data obtained in lock exchange experiments. He also made several simulations of 2D and 3D rectangular settling tank, and compared the results with field data.

In the first part of his thesis, J De Clercq [8] made a review of clarifier simulation models, computational methods, and experimental techniques available for field data collection, including among others: velocity measurements, suspended solids, settling velocity, rheology parameters measurement, and particle size distribution. In a second part the author simulated the behaviour of a circular secondary settling tank with Fluent software by the implementation of a 2D axil-symmetric domain. The CFD results were confronted with field
data, TSS and Tracer test measurements. In his simulations J De Clercq found some difficulties with the scrapper simulation. Some other aspects as particle size distribution in the clarifier and sludge rheology properties were studied in his thesis. The author made an interesting experimental study regarding rheology of activated sludge, proposing a modified Herschel-Bulkley model for sludge simulation. He also studied the influence of rheological model on simulation results.

In [9] J De Clercq et al., proposed a new 1D clarifier model. The partial differential equations were converted into a system of ODE, by the division of the clarifier into a number of layers. This model included one dispersion term, and a settling velocity function based on the Cho function [10].

Burt [11], in his thesis, studied the improvement of several circular secondary clarifier performance by the implementation of a center well lower baffle and a peripheral baffle. He used a 2D axis-symmetric model based on a drift-flux model. The author studied the influences of several variables, as turbulence model, sludge rheological properties, sludge transport in the clarifier floor, among others. He simulated a tracer test within the CFD model, and compared the results with data obtained in field test, showing good correlations. He also studied the influence of rheological model on simulations and compared them with the lock exchange test results over a synthetic latex sludge manufactured by himself. Finally he compared the CFD simulation results with experimental data, obtained at field before and after improvement solutions implementation.

### 1.3 Objectives

The objectives of the present work can be summarized as follows:

- Obtain a detailed insight of secondary clarifier performance, through the use of computational fluid dynamics (CFD) simulations.

- Study the influence of inlet baffle configuration on flow field and clarifier performance.

- Study the influence of effluent weir position and baffle configuration on flow field and clarifier performance.

- Analyse the obtained results in order to derive design recommendations for inlet and outlet internal structures.
2. **Secondary clarification**

2.1. **Sedimentation theory**

The objective of particulate matter sedimentation theory is to explain the movement of suspended solids dispersed in a fluid (continuous media), forced by gravity action due to density differences. The characteristics of this behaviour depends on the nature of suspended solids and the fluid within which they are moving. In this work we have considered water as the fluid and activated sludge as the dispersed phase.

Depending on suspended solids concentration and their physiochemical characteristics it can be established the following four different particle settling behaviours in waste-water treatment [1, 12]:

- **Type I**, discrete non-flocculent particle settling: dilute suspension of particles settle in an independent way with no interaction between them. It happens in low suspended solids concentration suspension as in sand and grit removal. Settling velocity is described by stokes law.

- **Type II**, flocculate settling: a dilute suspension of particles, due to its physiochemical nature tend to join with each other (floculate) to form bigger particles that settle faster due to increased mass.

- **Type III**, hindered or zone settling: intermediate concentration suspension of particles, that interact with each other and form an structured matrix that settle together in a relative fixed position.

- **Type IV**, compression settling: a high concentrated solids suspension that has settled together in a structured matrix, concentrate further due to compression of the structure by the weight of the particles above them.

In secondary settling tanks, although type II, III and IV can happen, sedimentation type III governs the design [12]. In Type III settling, total suspended solids concentration (TSS) is much higher than in type I and II. Suspended solids, settle as a unit with uniform velocity, maintaining the relative position between them until the sludge blanket reach the bottom. At that time the sludge blanket start compression, increasing TSS, therefore reducing the volume occupied by the sludge.
2.2. Solid flux theory

Kynch [13], proposed the solid flux theory, based on the hypothesis that “at any point in a dispersion the velocity of fall of a particle depends only on the local concentration of particles”, until there is an interaction between them, that leads to a density increase (see Figure 2.1). In this way, sedimentation velocity can be calculated from batch experiments at different initial concentration. Data obtained from theses experiments can be used to built a curve for zone settling velocity ($v_s$) versus suspended solids concentration ($X$), this is, a settling velocity function (see Figure 2.3).

![Settling curve of an activated sludge suspension (from [18])](image)

Regarding the shape of this function, between the different proposal [7, 10, 14] the openFoam model used in the present work is based on the work of Takács [14].

2.2.1. Settling velocity:

Takács model is represented by the following formula:

$$v_s = v_0 e^{-r_h X^*} - v_0 e^{-r_p X^*} ; \text{ with } 0 \leq v_s \leq v'_0$$  \hspace{1cm} (2.1)

where

- $v_s$ = settling velocity of the suspension,
- $v_0$ = maximum settling velocity,
- $X^* = X - X_{\text{min}}$; $X_{\text{min}}$ minimum attainable suspended solids concentration;
- $r_h$ = settling parameter characteristic of the hindered settling zone;
- $r_p$ = settling parameter characteristic of low solids concentration.

As it can be seen in Figure 2.2 Takács model distinguish between 4 different settling behaviors, function of suspended solids concentration:

- Region I: starting from 0 up to $X_{\text{min}}$ TSS, settling velocity is zero, this means that an small solid fraction is not able to settle so it will go away with the clarified effluent water.
- Region II: once TSS increases over $X_{\text{min}}$, the settling velocity increases as TSS increases.

- Region III: settling velocity has reached a maximum, that is maintained in a TSS interval.

- Region IV: after the maximum, settling velocity decays with increasing TSS, until it reaches zero, at high concentration. In this region the settling behavior is identical to Vesiling model.

In openFoam 6, two settling models are available: simple model and general model (see Figure 2.3). General model is similar to Táckacs model, but it doesn’t limit the maximum settling velocity [10, 15]. The code used can be located in the following route of the users installation:

```
“applications/solvers/multiphase/driftFluxFoam/relativeVelocityModels/general/ general.c
```

The code used in the model for drift velocity of dispersed phase is represented as follows:

\[
U_{\text{dm}} = (\rho_{c}/\rho) \cdot V_{0} \cdot \left[ \exp\left(-a \cdot \max(\alpha_{d} - \text{residualAlpha}, \text{scalar}(0))\right) \right. \\
\left. \quad - \exp\left(-a_{1} \cdot \max(\alpha_{d} - \text{residualAlpha}, \text{scalar}(0))\right) \right]
\] (2.2)

where:
- $U_{\text{dm}}$ //relative velocity between phases
- $\rho_{c}$ //continuous phase density
- $\rho$ //mixture density: $\rho = \alpha_{1} \cdot \rho_{d} + \alpha_{2} \cdot \rho_{c}$
- $a$ //Velocity exponent for region IV in Figure 2.2
- $a_{1}$ //Velocity exponent for region II in Figure 2.2
- residualAlpha //Non settleable suspended solids volumen fraction
- $V_{0}$ //maximum sedimentation velocity

In reference [6] experimental data shows that settling velocity match Simple model shape. On the other hand in reference [16] sedimentation velocity is reduced at low concentration, so general model seems to be more suitable to describe settling velocity.
Experimental works show the hypothesis of the reduction in settling velocity at low concentration [9-10, 17-21].

2.2.2. Correlation between SVI and settling velocity

Sludge Volume Index, referred as SVI, is the volume occupied by a gram of sludge after 30 min of sedimentation, measured as ml/g [22]. Several authors [18], have studied the correlation of settling velocity of activated sludge with SVI in order to find a fast and easy way to find sludge settling characteristics. In Figure 2.4 it is shown several settling velocity curves calculated from different authors correlations, as indicated by Härtel and Pöpel [18], for the same SVI. The curves presented in Figure 2.4 are calculated according to Takács model, with $r_c=0.1*r_h$. 

Figure 2.3
Settling velocity models available in openFoam 6

Figure 2.4
Different settling velocities correlations calculated for an activated sludge with SVI=110 ml/g
For the correlations showed before, the one of Härtel and Pöpel [18], will be used in the present study in order to calculate settling velocity of an activated sludge with SVI=110 ml/g.

2.3. Clarifier design approach

Regarding clarifier design procedures, two main streams exist: one based on solid flux theory followed mainly in English language speaking countries (UK, USA), and the other based on empirical correlations from field studies, whose main representatives are Germany and Netherlands, with their corresponding standard ATV and STORA.

The first stream uses solid flux theory to determine the required surface to avoid solid overload. Although it is based on a theoretical description of clarifier performances, that considers also the settling velocities as function of sludge quality, the obtained information is reduced to required area. Return sludge concentration can be also determined, without considering any dilution effect generated by the extraction mechanism or density currents generated. In any case this theory does not give any information about water deep, inlet configuration, baffle arrangements, weir locations, etc… Information regarding effluent suspended solids is not supplied either [23].

German approach (ATV standard) on the other hand is based on empirical relationships that also consider the sludge quality. With this standard is possible to calculate required area clarifier water deep, as well as return sludge concentration including possible diluting effect of sludge collection mechanism. It also provides guidelines about bottom slopes and scraper velocities. All this guidelines considers an effluent suspended solid concentration below 20 mg/l [24].

2.4. General characteristics and clarifier types

In previous paragraphs it has shown a brief introduction to solids settling theories, as well as available standards to calculate clarifier main dimensions. However, until the present moment, almost nothing has been mentioned about specific details in clarifier design. At the time of a clarifier design several options must be selected, regarding tank shape, input/output configuration, sludge removal mechanism, weir configuration,… In the Table 2.1 it is shown a variety of combinations that can be usually selected:

Apart from the option indicated in Table 2.1, some other design options are available as weir baffles, etc.. With such a big variety of options it is sometimes difficult to select the most appropriate combination of design factors that drives us to the best design condition. In this
sense apart from experience and testing, CFD tools can provide an essential insight in order to select the best design options. In Figure 2.5 it it shown some different flow patterns.

<table>
<thead>
<tr>
<th>Tank shape</th>
<th>Inlet position</th>
<th>Water position</th>
<th>Water out type</th>
<th>Sludge out position</th>
<th>Scraper type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular (or polygonal prism shape)</td>
<td>Center</td>
<td>Peripheral</td>
<td>Weir Submerged tube Or</td>
<td>Center bottom</td>
<td>Mechanical</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>All around bottom</td>
<td>Hydraulic suction</td>
</tr>
<tr>
<td>Radial</td>
<td>Center</td>
<td>Weir Submerged tube Or</td>
<td>Center bottom</td>
<td>Mechanical</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>All around bottom</td>
<td>Hydraulic suction</td>
</tr>
<tr>
<td>Peripheral</td>
<td>Center</td>
<td>Weir Submerged tube Or</td>
<td>Center bottom</td>
<td>Mechanical</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>All around bottom</td>
<td>Hydraulic suction</td>
</tr>
<tr>
<td>Rectangular shape</td>
<td>Short wall 1</td>
<td>Short wall 2 (in front of wall 1)</td>
<td>Weir Submerged tube Or</td>
<td>Short wall 2 end</td>
<td>Mechanical</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Short wall 2 end</td>
<td>Mechanical</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Middle length</td>
<td>Mechanical</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>All around bottom</td>
<td>Hydraulic suction</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1/3 end perpendicular to Short wall 2</td>
<td>Weir Submerged tube Or</td>
<td>Short wall 2 end</td>
<td>Mechanical</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Short wall 1 end</td>
<td>Mechanical</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Middle length</td>
<td>Mechanical</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>All around bottom</td>
<td>Hydraulic suction</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 2.1

Figure 2.5
Some circular clarifier flow profiles, (from [12])
3. **Sludge rheology**

Sludge rheology has an important effect on density current decay, what in turn will influence the velocity field inside the clarifier [7].

Ac. to Ratkovich *et al.* [25], activated sludge is usually modelled using the pseudoplastic power-law or Bingham plastic rheological models. The apparent viscosity which depends among others of TSS content and temperature, is the most critical parameter. In openFoam 6, for driftFluxFoam solver, plastic, slurry and BinghamPlastic models are available. In Figure 3.1 the shear stress / shear strain rate relationships for Newtonian and some non-Newtonian fluids are summarised. In the present work Bingham plastic model has been used with the standard parameters supplied in [7].

![Figure 3.1](image)

*Figure 3.1 Shear stresses-shear strain rate diagram. Adopted from [11]*

### 3.1. Bingham model

In the following paragraphs the Bingham model is described. This derivation has been taken from ref [8] and [11].

Apparent viscosity is calculated as:

\[
\mu_m = \frac{\tau_{ij}}{\dot{\gamma}_{ij}} \tag{3.1}
\]

where

- \( \mu_m \) apparent mixture viscosity
- \( \tau_{ij} \) Stress tensor
- \( \dot{\gamma}_{ij} \) Strain rate tensor ;

Shear rate is the component of the stain tensor describing velocity gradients perpendicular to velocity direction.
**Bingham model**

\[
\tau_{ij} = \left( \frac{\tau_0}{\dot{\gamma}} + k \right) \dot{\gamma}_{ij}; \quad \text{for } \tau \geq \tau_0
\]
\[
\dot{\gamma}_{ij} = 0; \quad \text{for } \tau < \tau_0
\]

(3.2)

Where: \( \dot{\gamma} = \sqrt{\frac{1}{2} \dot{\gamma}_{ij}^2} \) Magnitude of the strain rate:

- \( k \) is a fluid consistency index;
- \( \tau_0 \) plastic yield stress

Models with a yield stress (minimum shear stress required to initiate fluid strain), show an infinite apparent viscosity when the stress applies is below yield stress. To avoid this inconsistency in the flow field, several solutions have been proposed (see [8]).

In the Figure 3.2 it can be seen a real rheogram that shows the non existence of a plastic yield stress. This behaviour can be modelled through the perturbation model, among others.

![Typical rheogram for sludge. From [8]](image)

Implementation of Bingham Model in openFoam 6 can be found in archive “BinghamPlastic.C”.

---

Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations. 18
4. Mathematical model of two phase flow

Multiphase flows are present in many engineering applications, in particular we can find many examples in water treatment; for instance: primary and secondary clarification (solid-liquid phases), dissolved air flotation (solid-liquid-gas phases), activated sludge aeration (solid-liquid-gas phases), fluidized bed reactors (UASB) (solid-liquid-gas phases), etc...

The main difficulties in modelling multiphase or multicomponent flows relies on the presence of a deformable moving inter-phase, with fluctuating variables and discontinuities at the inter-phase [2].

In order to describe the performance of this kind of flows, different approaches can be used [7]:

- Euler-Lagrange approach: each particle trajectory of the dispersed phase is tracked individually through the flow domain. The momentum equation is formulated in coordinates that follows the particle trajectory.

- Euler-euler approach [26]: in this approach the phases are treated as a inter-penetrating continua with a connecting inter-phase. The concept of volume of fraction of each phase ($\alpha_k$) is introduced. The sum of $\alpha_k$ of all phases present is equal to 1.

  * Multi phase model: the mathematical description of this model, considers a set of equations (mass, momentum and energy conservation) for each phase separately. An interacting force between phases is introduced in the momentum equation. This interfacial interaction between phases incorporate mathematical complications and uncertainty [27].

  * Mixture model (diffusion model): the mathematical description of this model, considers only 1 set of equations (mass, momentum and energy conservation) for the mixture as a whole plus one additional transport equation (Diffusion equation or Dispersed phase continuity equation), which takes account for the concentration changes. Additionally constitutive equations for relative velocities between phases are included. The mixture properties are defined according to the properties of each phase the corresponding phase fraction ($\alpha_k$) present in the mixture. The drift-flux model is an example of a mixture model that includes diffusion model, slip flow model and homogeneous flow model [28]. The use of the drift-flux model is appropriate when the motions of two phases are strongly coupled [27].

In the case of settling tanks an experimental settling velocity is used to model the relative velocity between solid and liquid phase velocity (see section 2.1).
The Volume Of Fluid (VOF) model can model two or more immiscible fluids, it considers only 1 set of equations (mass, momentum and energy conservation) plus one additional equation for volumetric fraction tracking, modelled as a scalar transport equation.

In the present work the drift-flux model implemented in the software openFoam [15] will be used. In the following section, only mathematical formulation of this model will be described. Acc. to Ishii et al.[28], for most practical applications, the drift-flux model is the best mixture model that is highly developed for normal gravity as well as microgravity conditions.

4.1. Drift-flux model

In the following section the drift flux model equations are presented. The formulation that follow is based on the center of mass and Drift-flux Velocities. These equations are extracted from [1, 15, 28].

4.1.1. Equations of State and Mixture Properties

Phase fractions: \( \alpha_1 + \alpha_2 = 1 \) (\( \alpha \) is volume fraction) \( (4.1) \)

Mixture density: \( \rho_m = \alpha_1 \rho_1 + \alpha_2 \rho_2 \) \( (4.2) \)

Mixture pressure: \( P_m = \alpha_1 P_1 + \alpha_2 P_2 \) \( (4.3) \)

Velocity for mixture centre of mass: \( U_m = \frac{\alpha_2 \rho_2 U_2 + \alpha_1 \rho_1 U_1}{\rho_m} \) \( (4.4) \)

4.1.2. Kinematic Constitutive Equations.

Relative velocity of phase 1 to phase 2: \( U_r = U_1 - U_2 \) \( (4.5) \)

Diffusion velocity or velocity of phase k relative to velocity for mixture centre of mass: \( U_{km} = U_k - U_m \); \( k = 1,2 \) phases \( (4.6) \)

Introducing (4.2) and (4.6) into (4.4), the following relationship for the diffusion velocity is obtained:

\( \alpha_1 \rho_1 U_{1m} + \alpha_2 \rho_2 U_{2m} = 0 \) \( (4.7) \)
Only one diffusion velocity is required $U_{km}$, by a kinematic constitutive equation. The diffusion velocity is defined as:

$$U_{2m} = -\frac{\alpha_1 \rho_1}{\alpha_2 \rho_2} U_{1m} = -\frac{\alpha_1 \rho_1}{\rho_m} (U_1 - U_2) \quad (4.8)$$

**Drift velocity** of the phases:

$$U_{1d} = -\alpha_2 U_r \quad (4.9)$$

$$U_{2d} = \alpha_1 U_r \quad (4.10)$$

Relationship between the **diffusion velocity** $U_{km}$ and the **drift velocities** $U_{kd}$

$$U_{2m} = \frac{\rho_1}{\rho_m} U_{2d} = -\frac{\alpha_1 \rho_1}{\alpha_2 \rho_m} U_{1d} \quad (4.11)$$

**Total volumetric flux**: $J = \alpha_k U_k$ ; with $k:1,2 \quad (4.12)$

4.1.3. **Transport Equations**

*Mixture Continuity equation:*

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m U_{m,i})}{\partial x_j} = 0 \quad (4.13)$$

**Diffusion equation or Dispersed phase continuity equation:**

$$\frac{\partial \alpha_2 \rho_2}{\partial t} + \frac{\partial (\alpha_2 \rho_2 U_{m,i})}{\partial x_j} = -\frac{\partial (\alpha_2 \rho_2 U_{2m,i})}{\partial x_j} \quad (4.14)$$

*Mixture momentum equation:*

$$\frac{\partial (\rho_m U_{m,i})}{\partial t} + \frac{\partial (\rho_m U_{m,i} U_{m,j})}{\partial x_j} = -\frac{\partial P_m}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \alpha_k \tau_{k,ij} + \alpha_k \rho_k u'_{k,i} u'_{k,j} - \alpha_k \rho_k U_{km,j} U_{km,i} \right] + \rho_m g_{m,i} + M_{m,i} \quad (4.15)$$

Where $k=1,2$ (phases 1 and 2)

$l,j=1,2,3$

$M_{m,i}$: capillary force, that takes into account the surface tension effects as a momentum sink or source

$u'_{i,k}$: velocity fluctuation

Diffusion stress: this represents the momentum diffusion due to relative motion between the two phases.
4.1.4. Relative velocity model

Relative velocity model used to describe the slip velocity between solid phase (activated sludge) and water is defined as settling velocity. Settling velocity of the solid phase depends on the suspended solid fraction, and also on sludge characteristics. The details of this settling velocity are presented in section 2.2.1.

4.1.5. Turbulence modelling

In two phase flow the presence of a second phase normally with higher density, makes buoyancy a mayor force that affects turbulent kinetic energy not only its dissipation but also its generation in case of inverted density stratification [7]. In driftFluxFoam turbulence is modelled through a “buoyant modified k-ε model”. The reader can refer to Brennan [7] for further information.
5. Physical model and simulation set up

5.1. Secondary clarifier design

Due to the limited time for the development of the present study only circular clarifier with center well feed, and center sludge purge will be studied. Two positions for effluent outlet weir will be studied: perimeter and 1/3 of diameter. Although we did not find data for Europe, Bender and Crosby declares in [29] that “most activated sludge facilities in the United States uses center-feed, peripheral overflow, circular clarifier”.

As a base point for simulations, the standard ATV-a131 [24] was chosen. This standard is a recognized international protocol that includes design equations and recommendation for activated sludge process design including biological reactor and secondary clarifier.

The geometry must comply the following conditions:

- It must be representative of a real design including size and shape, so obtained results can be used in real designs.

- Time required for each simulation must be short enough, so it is possible to perform a reasonable number of simulations during the time available for this master work.

According to ATV-a131 [24], the surface overflow rate must be selected depending on the suspended solids concentration (TSS) in the influent, sludge sedimentation characteristics (SVI), and clarifier flow type (horizontal/vertical). This standard considered the clarifier flux predominantly horizontal if:

$$\frac{\text{vertical distance from inlet to water surface}}{\text{horizontal distance from inlet to outlet}} < \frac{1}{3}$$

In any case, if diameter is less than 20m, clarifier must be designed as vertical flux secondary clarifier. Depending on the mentioned ratio, a maximum sludge loading rate ($q_{SV}$) must be selected.

In horizontal settling tank flux type, an influent disturbance length is considered in the design. This length is considered equal to side wall depth of the tank. In the design adopted in the present work, the diameter used is 16.4 m (<20m), so vertical flow must be adopted. Nevertheless as Vertical/Horizontal coefficient is equal to 1/3, it has decided to include disturbance length in the adopted diameter.

The following parameters and dimensions have been selected (see Table 5.1):
### Secondary clarifier design parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design Influent flow (m³/h)</td>
<td>200</td>
</tr>
<tr>
<td>Recirculation flow (m³/h)</td>
<td>150</td>
</tr>
<tr>
<td>Total Influent flow to clarifier (m³/h)</td>
<td>350</td>
</tr>
<tr>
<td>Influent Suspended Solids (g/m³)</td>
<td>3.28</td>
</tr>
<tr>
<td>SVI (ml/g)</td>
<td>105</td>
</tr>
<tr>
<td>Sludge volume loading rate q(_{SV}) (l/m² h)</td>
<td>400</td>
</tr>
<tr>
<td>Surface overflow rate q(_A) (m/h)</td>
<td>1.16</td>
</tr>
<tr>
<td>Required total diameter (m)</td>
<td>16.30</td>
</tr>
<tr>
<td>Adopted total diameter (m)</td>
<td>16.40</td>
</tr>
<tr>
<td>Required total vertical height at side (m)</td>
<td>3.42</td>
</tr>
<tr>
<td>Adopted total vertical height at side (m)</td>
<td>3.50</td>
</tr>
</tbody>
</table>

*Table 5.1*

The operation conditions of the designed clarifier have been verified by the solid flux theory. In Figure 5.1 we show two solid flux curves:

- Black line represents solid flux line for an activated sludge with SVI=105. Settling parameters have been calculated according to Härtel and Pöpel correlation [18] (see chapter 2.2). The calculated settling values are: \(v_0=9.24\text{ m/h}, r_0=0.51, r=10*r_0\) (see eq (2.1))

- Gray line represents solid flux line for an activated sludge with same SVI, but the settling parameters have been adapted to eq (2.2), were the solid content is expressed in volume fraction. So in this case settling parameters are: \(V_0=9.24\text{ m/h}, a=524, r=10*r_0\).

Blue line represents the underflow rate line. If this line does not cross solid flux curve for a concentration greater to influent concentration, then the system is not overloaded. As it is shown in Figure 5.1 the system is operating very close to the critical condition, so the safety margin is almost zero. Under these conditions, depending on the hydraulic structures design the system can suffer a failure condition, with the lack of capacity for enough sludge removal.

In the Figure 5.2 the general dimensions of model rev1.1_refined_9 are shown. Apart from the modifications to influent center well, baffles and outlet weir, all other general dimensions are the same for all the studied models.
5.2. Experiment matrix for simulations

In the Table 5.2 it is shown the experimental matrix with the different simulations to be performed. The objective is to cover different number of combinations of inlet-outlet structures in order to find the design with the best performance, this matrix has been adapted during the study according to the results obtained in the simulations.
<table>
<thead>
<tr>
<th>Weir</th>
<th>Shape: Standard-inside Position; Perimeter Additional baffle: NO</th>
<th>Shape: Standard-outside Position; Perimeter Additional baffle: Double angle Position middle</th>
<th>Shape: Standard-outside Position; Perimeter Additional baffle: Double angle Position up</th>
<th>Shape: Standard-inside Position; Perimeter Additional baffle: NO</th>
</tr>
</thead>
<tbody>
<tr>
<td>Center well: Straight-Narrow</td>
<td>rev1.1_refined9</td>
<td>rev6_1_refined10</td>
<td>rev6_2_refined10</td>
<td>rev6_3_refined10</td>
</tr>
<tr>
<td>Center lower baffle: Horizontal-Normal</td>
<td>rev2_1_refined9</td>
<td>rev12_1_refined10</td>
<td>rev12_1_refined10</td>
<td></td>
</tr>
<tr>
<td>Center well: Straight-Narrow</td>
<td>rev6_1_refined10</td>
<td>rev6_2_refined10</td>
<td>rev6_3_refined10</td>
<td></td>
</tr>
<tr>
<td>Center lower baffle: Horizontal-Normal + Inclined Baffle</td>
<td>rev12_1_refined10</td>
<td>rev12_1_refined10</td>
<td>rev12_1_refined10</td>
<td></td>
</tr>
<tr>
<td>Center well: Straight-Narrow</td>
<td>rev4_1_refined9</td>
<td>rev7_1_refined10</td>
<td>rev7_1_refined10</td>
<td></td>
</tr>
<tr>
<td>Center lower baffle: NO</td>
<td>rev10_refined10</td>
<td>rev10_refined10</td>
<td>rev10_refined10</td>
<td></td>
</tr>
<tr>
<td>Center well: Straight-wide</td>
<td>rev3_1_refined9</td>
<td>rev9_1_refined10</td>
<td>rev9_1_refined10</td>
<td></td>
</tr>
<tr>
<td>Center lower baffle: NO</td>
<td>rev9_1_refined10</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
</tr>
<tr>
<td>Center well: Straight-Narrow + Inclined end 1</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_4_refined10</td>
</tr>
<tr>
<td>Center lower baffle: Horizontal-Short + Inclined Baffle 1</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_4_refined10</td>
</tr>
<tr>
<td>Center well: Straight-Narrow + Inclined end 2</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_5_refined10</td>
</tr>
<tr>
<td>Center lower baffle: Horizontal-Short + Inclined Baffle 2</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_5_refined10</td>
</tr>
<tr>
<td>Center well: Straight-Narrow + Inclined end 1</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_5_refined10</td>
</tr>
<tr>
<td>Center lower baffle: Horizontal-Short + Inclined Baffle 3</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_5_refined10</td>
</tr>
<tr>
<td>Center well: Straight-Narrow + Inclined end 2</td>
<td>rev11_1_refined10</td>
<td>rev11_2_refined10</td>
<td>rev11_3_refined10</td>
<td>Rev11_5_refined10</td>
</tr>
<tr>
<td>Center lower baffle: NO</td>
<td>rev5_1_refined9</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 5.2
Experiment matrix

Secondary clarifier inlet/outlet internal structures optimization by the use of CFD simulations.
6. **Modelling**

6.1. **Software**

For the development of the present study the following software has been used:

* **gmsh**: version gmsh-4.2.3-Linux64, for geometry and mesh development

* **openFoam**: version openFoam 6 (this work was started with version 3.0; however, due to issues in the relative velocity model, it was decided to upgrade to the latest version). The solver used is driftFluxFoam.

* **ParaView**: version 5.4.0 64 bit, for post-processing.

* LibreOffice Version: 6.2.3.2

All the software packages used in the present study are openSource and free licence software.

6.2. **Initial and Boundary conditions (B.C.) implementation**

Due to limited computational resources, it was decided to simulate the system in 2D, with axil-symmetric boundary condition in z plane. The geometry is specified as a wedge of angle 2°, and 1 cell thick.

Figure 6.1 shows names for model 1.1_rev9. These names are the same for all the models apart from those cases where additional baffles have been included. In Table 6.1 boundary conditions are specified for all the patches. In the following paragraphs some of the more remarkable features are mentioned:

For 2D axil-symmetry simulation, wedge boundary condition is used in both symmetry planes, for all variables.

Free surface of secondary clarifier can be modelled with slip boundary condition in all variables except p_rgh.

Velocity (flow) is fixed in inlet and Outlet_Sludge, while in Outlet_Water velocity is calculated through the use of “pressureInletOutletVelocity” boundary condition.

All physical walls, apart form Pend_Dec, are treated with no slip condition, so velocity is fixed as 0.
A fixed translating velocity was imposed to the bottom of the clarifier (Pend_Dec) in order to simulate the bottom scraper, that move the sludge from all the surface up to the hopper.

Pressure: driftFluxFoam solver uses \( p_{rgh} \), defined as: \( p_{rgh} = p - \rho \cdot g \cdot h \);

Pressure B.C. is fixed as fixedFluxPressure is all the patches except, symmetry planes, and Outlet_Water. According to openFoam Users Manual [30] “This boundary condition is used for pressure in situations where zeroGradient is generally used, but where body forces such as gravity and surface tension are present in the solution equations”. See also [31].

Alpha.sludge defines the suspended solids contents, as a volumetric fractions. For the simulations performed the following inlet value has been used:

\[
[\text{MLSS}] = 3.28 \frac{g_{\text{sludge}}}{l_{\text{total}}} ; \quad \alpha_{\text{sludge}} = \frac{3.28 \left( \frac{g_{\text{sludge}}}{l_{\text{sludge}}} \right)}{1042 \left( \frac{g_{\text{sludge}}}{l_{\text{sludge}}} \right)} = 0.00315 \frac{l_{\text{sludge}}}{l_{\text{total}}} \text{ (volume fraction)}
\]

Regarding turbulence model, wall functions were used, apart from inlet/outlet boundary conditions (see Table 6.1).
Figure 6.2
Clarifier model section view
### Table 6.1
Model boundary conditions

<table>
<thead>
<tr>
<th>Patch name</th>
<th>U</th>
<th>p_rgh</th>
<th>alpha.sludge</th>
<th>k</th>
<th>ε</th>
<th>μt</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simetry_Plane1</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>Simetry_Plane2</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
<td>wedge</td>
</tr>
<tr>
<td>Inlet</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>type fixedValue;</td>
<td>type fixedValue;</td>
<td>type fixedValue;</td>
<td>type calculated;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0.7737 0);</td>
<td>value uniform 0;</td>
<td>value uniform 0.00315</td>
<td>value uniform 0.000259</td>
<td>value uniform 1.973e-07</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Free_Surface</td>
<td>slip</td>
<td>type fixedFluxPressure;</td>
<td>slip</td>
<td>slip</td>
<td>slip</td>
<td>slip</td>
</tr>
<tr>
<td>Column</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Inlet_pipe</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Campana</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Outlet_Sludge</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>type inletOutlet;</td>
<td>type inletOutlet;</td>
<td>type inletOutlet;</td>
<td>type calculated;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 -0.02763107 0);</td>
<td>value uniform 0;</td>
<td>inletValue uniform 0</td>
<td>inletValue uniform 0.000259</td>
<td>inletValue uniform 1.973e-07</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Tolva</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Pend_Dec</td>
<td>type translatingWallVelocity;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>U (-0.003 0.0 0.003);</td>
<td>value uniform (-0.003 0.0 0.003);</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Out_Wall</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Weir_Wall</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Outlet_Water</td>
<td>type pressureInletOutletVelocity;</td>
<td>type fixedValue;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
<tr>
<td>Deflectora</td>
<td>type fixedValue;</td>
<td>type fixedFluxPressure;</td>
<td>zeroGradient;</td>
<td>type kqRWallFunction;</td>
<td>type epsilonWallFunction;</td>
<td>type nutkWallFunction;</td>
</tr>
<tr>
<td></td>
<td>value uniform (0 0 0);</td>
<td>value uniform 0;</td>
<td></td>
<td>value $internalField;</td>
<td>value $internalField;</td>
<td>value $internalField;</td>
</tr>
</tbody>
</table>

Secondary clarifier inlet/outlet internal structures optimization by the use of CFD simulations.
6.3. Equations discretization schemes

In order to configure the simulation, tutorial tank3D (available in openfoam6/tutorials/multiphase/driftFluxFoam/RAS/tank3D) was used as starting point. Although this case works properly with the default discretization schemes of tutorial case in version 3.0, it didn't work properly in version 6. Simulation started but after some steps, deltaT started decreasing, until system crashed. After some optimization test stable discretization parameters were found. These are specified in third column of Table 6.2.

<table>
<thead>
<tr>
<th>ddtSchemes</th>
<th>tank3D</th>
<th>rev1.1_refined9</th>
</tr>
</thead>
<tbody>
<tr>
<td>gradSchemes</td>
<td>Gauss linear</td>
<td>Gauss linear</td>
</tr>
<tr>
<td>divSchemes</td>
<td>Gauss linearUpwind grad(U)</td>
<td>Gauss linearUpwind grad(U)</td>
</tr>
<tr>
<td>div(rhoPhi,U)</td>
<td>Gauss linear</td>
<td>Gauss linear</td>
</tr>
<tr>
<td>&quot;div(phi,alpha.*)&quot;</td>
<td>Gauss vanLeer</td>
<td>Gauss vanLeer</td>
</tr>
<tr>
<td>&quot;div(phirb,alpha.*)&quot;</td>
<td>Gauss linear</td>
<td>Gauss linear</td>
</tr>
<tr>
<td>div(rhoPhi,k)</td>
<td>Gauss limitedLinear 1;</td>
<td>Gauss upwind;</td>
</tr>
<tr>
<td>div(rhoPhi,epsilon)</td>
<td>Gauss limitedLinear 1</td>
<td>Gauss upwind;</td>
</tr>
<tr>
<td>div((rho*nuEff)*dev2(T(grad(U))))</td>
<td>Gauss linear</td>
<td>Gauss linear</td>
</tr>
</tbody>
</table>

Table 6.2
Comparative discretization schemes between tutorial tank3D and the model used in the present study.

6.4. Mesh implementation and mesh independence analysis

Model geometry as well as mesh implementation was developed with the software gmsh. Acc. to “Gmsh Reference Manual” [32], “Gmsh is a three-dimensional finite element grid generator with a build-in CAD engine and post-processor”. This software was used to perform geometries and mesh. Meshing tool allowed structured and unstructured meshing (or a mixture of both) in 1D, 2D and 3D, with an important control over the mesh generation process.

In the present work a mixture of structured and unstructured mesh was used. The starting point was half of the axil-section view of the clarifier. From the global clarifier domain, it was generated 2 additional layers to the inside of the domain in order to stablish an structured mesh in the walls and patches, and an unstructured mesh inside the domain. Between them a transition unstructured domain (yellow mesh in Figure 6.3) was added in order to control more accurately both: the mesh size at the wall and the mesh size in the bulk.
liquid, inside domain. In Figure 6.3 this features can be appreciated. In annex III, there is an example of an archive created to generate the geometry and mesh in the present job.

![Image](image.png)

Figure 6.3
Model rev1.1_refined9: mesh layers detail, left: general view, right: sludge hopper detail.

6.4.1. Mesh independence analysis

In order to perform a mesh independence analysis case "rev6_refined10" was selected. From this case mesh size was refined (case "rev6_refined11_1"), obtaining the results shown in Table 6.3. Refined mesh has more than double number of cells, while maximum cell volume is almost the same, minimum volume cell is reduced one order of magnitude.

Figure 6.4 illustrates the differences between both meshed. From Figure 6.5 to Figure 6.8 we can see the comparative results of the simulation for coarse and refined mesh, under the same simulation conditions. In general solid contours, velocity contours and turbulence kinetic energy contours shows a similar shape for both cases. We can focus mainly in two differences:

- Low solids profile: in Figure 6.6 we can see a small difference in the TSS cloud located in the top section, of the last half part of the clarifier. While coarse model shows a alpha around 3.1 e-05 in that area, refined model shows an alpha around 5.0 e-05.

- Flow at the entrance from the well to the open area: in Figure 6.7 and Figure 6.8 we can see a difference in the flow field just in the entrance of the sludge form the well to the open area of the clarifier. In refined model the current that moves the sludge to the hopper at the bottom of the tank (at x=2 m) goes up and impact with the currents from center well. This phenomenon happens in both simulations, but in refined model the deviation of the incoming current is more pronounced. In respect to the flow field in the rest of the clarifier, not only the shape but also the magnitude is similar in both cases.
Regarding simulation time, coarse model took 4472 real seconds to simulate 1000 sec, while refined mesh, took 21880 real seconds to simulate the same period, this means 4.9 times more for refined version.

Considering the original objective of simulating the maximum number of combinations inside the experimental frame fixed for this work, it was decided to use the "coarse mesh" version for the development of all the simulations. In view of the obtained results for both meshed presented, the results from a quantitative point of view should be taken with care. In any case in view of the complexity of the mathematical model used to describe activated sludge sedimentation, as well as the uncertainty about numerous model parameters, an
experimental verification of the obtained results is required before consider the same as valid. Specially if reliable quantitative data are required.

<table>
<thead>
<tr>
<th>Mesh stats</th>
<th>rev6_refined10 (coarse)</th>
<th>rev6_refined11_1(refined)</th>
</tr>
</thead>
<tbody>
<tr>
<td>points:</td>
<td>19059</td>
<td>39114</td>
</tr>
<tr>
<td>faces:</td>
<td>57999</td>
<td>123437</td>
</tr>
<tr>
<td>cells:</td>
<td>16165</td>
<td>34636</td>
</tr>
<tr>
<td>Min volume</td>
<td>3.29069e-07</td>
<td>4.55601e-08</td>
</tr>
<tr>
<td>Max volume</td>
<td>0.00178495</td>
<td>0.00179006</td>
</tr>
<tr>
<td>Total volume</td>
<td>4.28393</td>
<td>4.28393</td>
</tr>
<tr>
<td>Mesh non-orthogonality Max:</td>
<td>54.5483</td>
<td>49.8874</td>
</tr>
<tr>
<td>Mesh non-orthogonality average:</td>
<td>15.0132</td>
<td>12.9139</td>
</tr>
<tr>
<td>Max skewness</td>
<td>2.34614</td>
<td>2.14228</td>
</tr>
</tbody>
</table>

Table 6.3
Main Mesh parameters for original and refined model

Figure 6.5
Low range TSS contour, left case rev6_refined10, right case rev6_refined11_1

Figure 6.6
Low range TSS contour, left case rev6_refined10, right case rev6_refined11_1
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
7. **Results and analysis**

In annex I, we have included figures of solid contours, velocity contours (including stream lines) and turbulence kinetic energy contours, for all the simulations specified in Table 5.2.

The first step that we have followed in data analysis is to check stationary conditions for the specific simulation results. To perform that, we have made a mass balance for water and sludge. In this balance total flow and Total Suspended Solids through Inlet, Outlet_Sludge, and Outlet_Water patches were calculated. With these values it was verified that inlet mass flow is equal to outlet mass flow, so no accumulation is present in the system, hence stationary condition were reached (refer to annex II, for calculation details).

![Image](image_url)

**Table 7.1**
Simulations mass balance results

In the Table 7.1, the mass balance results are shown. In red we present several simulation results in which stability was not reached in the simulated time. In black we present simulation results that reached steady state in the simulated time.

In Figure 7.1 we can see TSS profile along a vertical line placed at R=5m, for all the simulated cases. Sludge height and concentration determines sludge inventory in the clarifier, as well as sludge retention time. Sludge retention time is important to avoid denitrification and subsequently floating sludge in the clarifier that can disturb effluent quality.
So, less sludge height implies more sludge storage capacity for peak flow events, and less sludge retention time with less risk of undesired de-nitrification.

![TSS profile along clarifier height, at a distance of R=5 m](image)

**Figure 7.1**
TSS profile along clarifier height, at a distance of R=5 m

### 7.1. Cases were steady state was not reached:

In order to analyse the causes of failure to reach the steady state of some hydraulic configurations we can focus on one of them. In Figure 7.2, we have a case were no influent well lower baffle is installed. As we can see in the velocity contour plot, the inlet flow hits the influent well and falls straight up to the bottom of the clarifier. At that point the current impact the sludge hopper and returns in ascending direction up to the middle of the tank height. After that, a defined narrow current is redirected up to the peripheral wall, where hits the same and splits into two different fractions: one to the top of the surface, where the weir is installed, and the other to the bottom where the sludge is moved again to the hopper.

We can interpret that two phenomena are happening, in one hand there exists a short circuit, so the influent is redirected to outlet pipe, without previous thickening. On the other hand the thickened sludge that is moved from all the bottom wall to the sludge hopper, can not reach the outlet pipe inside the hopper because the turbulence inside the hopper acts as a barrier for this movement. These phenomena can be appreciated also in other inlet configurations without center well lower baffle, see Figure 10.10 or Figure 10.14 in annex I, for reference.

Although we can find references in literature [33] which corroborate that short-circuit happens in full scale real installations, it is not clear that the intensity of the phenomenon found in the simulation is according to reality. This doubt is supported by the fact that there
are many installations operating with center feed, center sludge withdrawal but without center well lower baffle.

In order to explain the mentioned issue some hypothesis are postulated:

- Improper selection of sludge rheological parameters. The influence of rheological properties in sludge recirculation system short-circuit will be analyze later in section 7.3.1.

- System overload: as can be seen in Figure 5.1, according to solid flux theory the operation point of the system although apparently not overloaded, is almost at the limit. The influence of operation point in sludge recirculation system short-circuit will be analyze in section 7.3.2.

- Improper simulation of sludge transport mechanism (scraper movements simulation). As mentioned in section 6.2, sludge scraper stimulation has been implemented as a moving bottom wall. So the scraper blades that in real systems moves the sludge up to the hopper are not implemented in the actual simulations. Although some authors (see [8]), has modelled scraper blades in a 2D model, that option was not studied in present work due to time limitations.

One interesting characteristic that we appreciate in overloaded simulations (see for example case rev5_refined9 (in annex I) is that the density current tends to rise above the sludge layer. Also the turbulence generated in sludge blanket is not strong enough to transport the sludge from sludge blanket up to effluent weir. We can appreciate that the velocity magnitude of the horizontal velocity current that goes to the outer wall is lower in these cases that in cases as rev3_refined9 or rev6_refined10. This is apparently caused by
the energy dissipation in the incoming section of the clarifier. Dissipation is higher in simulations were sludge transport failure has happened.

### 7.2. Steady state simulation results:

In this section we are going to focus on simulation results that have reached steady state, in order to analyse the influence of design parameter for inlet and outlet configuration on sludge blanket height and effluent suspended solid profile in clarified water. The different options studied are summarized in Figure 7.3:

In Figure 7.4 we can see the velocity profile for two different inlet well configurations: straight well with lower baffle and inclined end well with lower baffle, both with peripheral weir. In this figure we can appreciate how the mixture sludge/water, enters the clarifier and generates an strong density current. After that, the current is redirected to the clarifier perimeter wall, where it hits the wall and rises to the surface. Finally the current comes back to the center of the clarifier. Depending on the position of the center well baffles, this flow current falls more or less sharply up to the bottom of the clarifier. Apart from that, the general flow field characteristics are very similar in both cases even with different inlet configurations, especially the density current that moves to the perimeter.

**Figure 7.3**

The position of the baffles has two main targets:

- Avoid interference of inlet density current with the sludge in the hopper.

The sludge in the hopper must be thickened in order to be able to remove the necessary quantity to maintain the solid balance in the clarifier without sludge accumulation. Several bottom baffle has been simulated.
- Avoid the drag of the sludge from the sludge blanket up to the weir. To get this objective is important to dissipate as much energy as possible with the inlet configurations. Several strategies can be followed:

* Stop the density current that goes to the perimeter (see Figure 10.36 and Figure 10.38). This kind of baffles was implemented in [29].

* Redirect the current back to the center of the clarifier, avoiding the current to raise directly from the wall to the effluent weir. To achieve this different perimeter baffles were proposed, see [12]. In the present study apart from the weir wall located inside the side wall, two more different options were simulated: see Figure 10.34, Figure 10.28 in annex I (double angle baffle was simulated at 2 different heights).

* Place the effluent weir inside the clarifier, instead of in the perimeter.

In the following paragraphs only the result of simulations without short-circuit will be analysed (black lines in Table 7.1).

In model rev2_refined9 the inclined baffle contributes to dissipate the incoming turbulent kinetic energy in hopper area, and on the other hand density current generated by Influent Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

### Figure 7.4
Velocity contour for simulation case “rev6_refined10” (left) and case rev3_refined9 (right)

#### 7.2.1. Inlet structures results analysis

In general, from simulated cases that reached steady state, there are no important differences in sludge blanket height. There are two exceptions case rev2_refined9 and rev12_1_refined10, where the sludge blanket is about 0.5 below the other cases. Case rev12_1_refined10 will be analysed in section 7.2.2.

In model rev2_refined9 the inclined baffle contributes to dissipate the incoming turbulent kinetic energy in hopper area, and on the other hand density current generated by Influent...
water/sludge mixture is redirected far away the sludge hopper thanks to the additional inclined baffles (see Figure 7.5). This current promotes TSS dragging up to the effluent weir. As a consequence of that, effluent TSS are the highest of all tested simulations.

Regarding lower baffle option, the model studied in rev9_refined10, and rev11_refined10 (Center well: straight Narrow + Inclined end), shows a big dependency of size and position of lower baffle, changing from a successful behaviour in model rev11_refined10 to a risky behaviour in model rev9_refined10, where short-circuit is almost imminent due to the turbulence generated in the hopper area by the incoming water/sludge current (see Figure 7.6). This turbulence prevents the sludge transport from the clarifier bottom up to the sludge hopper. However, in model rev11_refined10 this phenomenon is reduced by moving the bottom baffle to the center of the clarifier. Due to that sensitivity to baffle position and size this configuration should be used with care.

On the other hand, models rev1.1_refined9 and rev6_refined10, were simulated with the shape recommended by ATV [24]. Two options are evaluated, in the first one the diameter of the lower baffle is the same of the inlet well. In the second one the lower baffle extend 1.2 times the radius of the center well, and the open space between vertical well and horizontal baffle is 0.5 times. Apparently results do not show any important difference regarding velocity currents (see Figure 7.7). The more remarkable difference is related to center well opening size. In model rev6_refined10 this opening is higher; this causes a greater recirculation of water from the clarifier open area to inside center well. Finally it should be noted that effluent suspended solid concentration are similar in both models, 34 mg/l versus 32 mg/l (Table 7.1).
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

Figure 7.6
\( k \) contour: Left: DecCircular3D_rev9_refined10 / Right: DecCircular3D_rev11_refined10

Figure 7.7
U magnitude contour: Left: DecCircular3D_rev1.1_refined9 / Right: DecCircular3D_rev6_refined10
7.2.2. Perimeter vertical ring result analysis

In model rev12_refined10 and rev12_1_refined10, a vertical ring baffle was introduced at a distance of R/2, from clarifier center. This configuration was used by Crosby (see [34]) for the retrofit of existing installations. The ring baffle is located 0.24 m over tank bottom, to allow sludge scraper to transport the sludge up to the hopper. On the other hand 2 heights have been used, 1 meter in model rev12 and 1.5 m height in model rev12_1.

In figure Figure 7.8 velocity contour is shown for both models. In case with 1 meter high baffle, the heigh of the baffle disturbs density current, but disturbance is not enough to change notably the velocity profile inside clarifier, compared with velocity profile without ring baffle (see Figure 7.7). However if baffle high is increased up to 1.5 m high, velocity profile changes. Baffle almost stops the density current from the influent, so that density current still travels up to the wall but the rebound with the wall generates a current at a closer distance to the bottom. This phenomenon reduces the drag of solids up to the effluent weir compared to other cases (see Figure 10.11, Figure 10.35 and Figure 10.37 in annex I). In model rev12_1 (see Figure 10.37 in annex I) it can be seen an small area close to effluent weir, where alpha.sludge is around 2.5 e-5 (TSS=26 mg TSS/L). In model rev12 however, effluent weir area is over TSS>36 mg TSS/L.

It is worthy to note that in model rev12_1_refined10 the sludge blanket at R=5 m, this is, after ring baffle position (see Figure 7.1) is maintained very low, so baffle is effective in confining sludge near the centrer of the clarifier. Its interesting to note that efficiency of 1 m high baffle (rev12) is limited compared with the model of 1.5 m baffle (rev12_1).
7.2.3. **Weir baffles and weir position**

Regarding weir baffle and weir location impact on clarifier performance, three options were studied for straight narrow center well (cases rev6_) and for Straight narrow + inclined end center well (cases rev11_) (see Table 5.2). Although all figures are available in annex_I, we are going to include here some of them for clarity.

**Inlet structure: straight inlet well+ lower baffle:**

From Figure 7.9 to Figure 7.16 different baffle configurations and weir locations are shown for the straight + lower baffle inlet structure. As we can see in those figures, baffles located in the outside wall are not able to modify sufficiently the flow field, and solids content of the effluent are not significantly influenced.
However the results obtained for model rev6_3 are more interesting: the weir located inside the clarifier has an important impact on effluent suspended solids content. The solids content has been reduced up to 20 mg/l (Table 7.1) this is a 37% of solid content reduction compared with the original model (rev6).

In Figure 7.16, the solids contour clearly shows how the solid content near the weir area is considerable lower than close to the wall. This means that even with the higher velocity, close to the weir, the influence of the density current is minimized due to the position of the weir. The higher weir velocity in this model is justified by the lower weir length available due to radial position of the same.
**Inlet structure: Straight narrow + inclined end, center well:**

From Figure 7.17 to Figure 7.24 different baffle configurations and weir locations are shown for the straight narrow + inclined end center well inlet structure. Unlike what happened in the previous studied case, in this case peripheral baffle noticeably influence solid content in weir area. Specifically from both baffles studied the one in model rev6_11_4, produced better results.

![Velocity profile](image1)

![TSS profile](image2)

![Velocity profile](image3)

![TSS profile](image4)

Although data displayed in Table 7.1 do not show a big improvement in effluent TSS, solid profiles figures shows that in base model (rev6_11) there is an area of alpha>3.5e-5, very close to the effluent weir. However in the other models the TSS concentration in this
area is reduced to alpha values below 2.5e-5 (28% percentage reduction). From both baffle models, the second one (rev6_11_4) shows better results. On the other hand the model with the weir inside the clarifier (rev6_11_3) also shows good results. Differences in effluent TSS between the last two options mentioned are not significant.
7.3. Sludge transport failure: hypotheses verifications

7.3.1. Influence of sludge rheology on simulation results

In this section we are going to study the influence of some rheological parameters on clarifier performance, in order to verify whether these properties could be the cause of the failure in transporting the sludge up to the sludge hopper. As mentioned by Clercq in [8], sludge rheology can greatly influence flow field developed in the clarifier. In Figure 7.25 it is shown the differences in flow field inside sludge hopper for two different rheological properties set.

![Figure 7.25 Scheme of velocity field near sump for rheologies without (left) and with (right) yield stress. Only half a cross-section of the sump is shown (from [8])](image)

From case “rev4_refined9”, as a starting point, we modified different parameters of the rheology model (see Table 7.2). After some “small” parameters changes without remarkable results it was decided to adopt the values used in “dahl” tutorial available in openFoam 6. The implementation of these new parameter (t=164600 s) resulted in a different flow field, where the short circuit disappeared, and the sludge reached the hopper easily. In Figure 7.26 alpha sludge with velocity streamlines is shown for time t=14800 s, and t=177600 s. It is also interesting to note the difference in kinetic energy dissipation between both cases (Figure 7.27). Density current over extension due to low turbulence energy dissipation was mentioned in [7].

Among others, solids density for both models shows a great difference. While default value used in present work is 1042 kg/m$^3$ (acc. to [7]), “dahl” tutorial default value is 1996 kg/m$^3$. We have found a wide variety of results in bibliography, for example: Burt [11] used a value of 1445 kg/m$^3$, for dried sludge density. According to the correlation shown in [6] the calculated value for dried sludge density is 2119 kg/m$^3$. De Clercq [8] reported a dry solids density of 1750 kg/m$^3$ for a mixture of biomass-zeolite.
Table 7.2
Rheology parameters at different times for model rev4_refined9_2

<table>
<thead>
<tr>
<th>t</th>
<th>coeff</th>
<th>exponent</th>
<th>BinghamCoeff</th>
<th>mHat</th>
<th>Density</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-07</td>
<td>95.25</td>
<td>30</td>
<td>1042</td>
</tr>
<tr>
<td>0</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>8.237E-07</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>5000</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>8.020E-07</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>10100</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>9.021E-07</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>14500</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>9.441E-07</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>49400</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>9.441E-07</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>52100</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>1.019E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>66700</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>1.098E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>70700</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>1.179E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>90400</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>99300</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>111100</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>123800</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>145600</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>147600</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>159600</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>7.845E-06</td>
<td>95.25</td>
<td>30</td>
<td>1042 No changes, short circuit present</td>
</tr>
<tr>
<td>164200</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>5.996E-04</td>
<td>105.8</td>
<td>20</td>
<td>1041 Values of Df format, Flow left exchange, short circuit NOT present</td>
</tr>
<tr>
<td>177800</td>
<td>2.314E-04</td>
<td>1.792E-01</td>
<td>5.996E-04</td>
<td>105.8</td>
<td>20</td>
<td>1041 Flow left change, short circuit NOT present</td>
</tr>
</tbody>
</table>

Figure 7.26
Alpha sludge contour for model rev4_refined9_2: Left t=14800 seconds (sludge level going up due to short circuit). Right t=177600 seconds (sludge level going down).
In order to distinguish between rheological and density effects on the simulation results, we performed 3 simulations (see Table 7.3):

<table>
<thead>
<tr>
<th>Simulation 1</th>
<th>Simulation 2</th>
<th>Simulation 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rheological parameters acc. to Dhal (see parameter of t=177800 s in Table 7.2)</td>
<td>Rheological parameters acc. to Dhal (see parameter of t=164600 s in Table 7.2)</td>
<td>Rheological parameters acc. to Brennan (see parameter of t=original in Table 7.2)</td>
</tr>
<tr>
<td>Low density</td>
<td>High density</td>
<td>High density</td>
</tr>
<tr>
<td>Model: rev4_refined9_2_0; t=16599 s</td>
<td>Model: rev4_refined9_2_0; t=21300 s</td>
<td>Model: rev4_refined9_2_2; t=21600 s</td>
</tr>
</tbody>
</table>

Table 7.3

Figure 7.27
Turbulence kinetic energy contour k for model rev4_refined9_2: Left t=14800 seconds. Right t=177600 seconds.
Figure 7.28
SS_profile: Simulation 1 / Simulation 2 / Simulation 3 (see Table 7.3)

Figure 7.29
U_profile: Simulation 1 / Simulation 2 / Simulation 3 (see Table 7.3)
Figure 7.30
$k_{profile}$: Simulation 1 / Simulation 2 / Simulation 3 (see Table 7.3)
The results of the three simulations are shown in Figure 7.28 to Figure 7.30:

First of all it is interesting to note that in simulations with density=1996 kg/m$^3$, short-circuit is not present. As can be seen in eq. (2.2), density directly affects relative velocity between phases, an increase in dispersed phase density increases relative velocity at high TSS ranges.

Regarding flow field, in both simulations with higher dispersed phase density, the velocity current (density current) generated from the inlet well, has more intensity and consequently it reaches the outer wall with higher velocity. Velocity is higher in the simulation 3, showing less turbulent kinetic energy dissipation (this is confirmed in Figure 7.30). Also it is interesting to note in the simulation 3, the higher velocity in the bottom of the clarifier, which moves the sludge up to the hopper. Remember that moving wall velocity is the same in all simulations. Finally we want to highlight the current that returns to inlet well in sim 3. Although this current is also present in other cases (see for example Figure 10.2) its magnitude is higher in sim 3.

Regarding turbulence kinetic energy we can see in Figure 7.30, a great difference between simulation 1 and 2. Also it is interesting ot see in sim 3, how in the sludge hopper the turbulence kinetic energy is lower than sim 1 in some areas. This issue is related to the capacity of this model to transport the sludge up to the hopper.

### 7.3.2 Influence of operation point on simulation results

In order to check the hypothesis of system overload, the following simulations were performed:

- Recirculation flow increase from 150 m$^3$/h up to 240 m$^3$/h. With this action the slope of the underflow rate is increased, so the system operation point is further from overload point. (Case rev4_refined9_3)

As is is shown in Figure 7.31, the short-circuit still is present, and sludge fail to reach the sludge hopper.

- Inlet flow reduced from 200 m$^3$/h to 100 m$^3$/h, and recirculation flow equal to 120 m$^3$/h. (Case rev4_refined9_5). As shown in Figure 7.32, under these conditions the sludge reaches the hopper, but still the incoming flow falls directly on the hopper, which apparently still hinders the entry of sludge into the hopper. According to the mass balance the TSS recovery is 99.38%, so the system is under stationary conditions.
If we compare velocity contour of original simulation (see Figure 7.33) with reduced velocity simulation (see Figure 7.34), we see that the velocity magnitude is smaller just in the area were the sludge reach the hopper for the second case. This reduced velocity allows the sludge to reach the hopper.

Also in Figure 7.34 we see the density current that moves horizontally over the sludge blanket, from inlet up to the wall. Once the current hits the wall, it is redirected up to the weir area located in the top side of the wall. So still in reduced flow simulation, the flow field in the bulk area of the clarifier is similar to other cases.
8. **Summary and conclusions**

In view of the results obtained in the present work we can get the following conclusions.

- **Inlet structure has an important role in avoiding sludge short circuit, and energy dissipation:**

  Simulations have shown that density current generated just in the entrance from the center well to the clarifier main volume, can promote problems in sludge transport up to the hopper that can end in a clarifier total failure.

  In this respect it is very important, in clarifier with center feed and center sludge purge configurations, to provide any kind of inlet bottom baffle that avoid an hydraulic short circuit and excess of turbulence in the sludge hopper.

  From the several studied systems the influent well bottom baffle of cases rev6_refined10 and rev12_refined10, are the most reliable. Although other configurations have shown better results in some cases (see case rev3_refined9), they suffer from more sensitivity to size and position, what also suggests that they will be more sensitive to flow variations or system overloads.

- **Effluent weir baffle can significantly reduce effluent TSS concentration**

  Simulations results show how the inlet density current moves directly to the outer wall in a narrow section flow current. For configurations with effluent weir in the peripheral wall, this current can transport an excess of sludge directly to the treated water reducing the clarifier performance.

  To avoid this phenomenon several configurations have shown their efficiency. In particular the one used in case rev11_4 has evidenced best results. The peripheral baffle redirects the density current from outer wall, again to the center of the clarifier. As the water leaves the tank in the perimeter top side, part of the redirected current is separated from the main stream and goes to the effluent weir. This current combinations reduces the effluent TSS content due to the presence of a reduced velocity area close to the effluent weir.

- **Effluent weir position can significantly reduce effluent TSS concentration**

  Simulations have exhibited good results for all cases with the effluent weir located at 1/3 R from the clarifier inlet. Even with the higher weir load, TSS content in treated water has been reduced significantly. As it happens in the cases with weir located in the perimeter, in this case also the influent current goes directly from the center well up to the peripheral wall.
At that point the current is redirected back to the center but velocity magnitude is reduced so the TSS transported by the water current to the effluent weir are also reduced significantly.

- **Rheological properties and sludge density have a great impact on simulation results**

  Sludge rheological properties and density are key points in kinematic energy dissipation and flow field on clarifier. From simulations performed, the following can be deducted:

  Rheological properties have a great impact on turbulence kinetic energy dissipation, however density is the parameter that most influences the sludge transport to sludge hopper.

  Given the previous results, it follows that it is important to study the sludge properties experimentally, in order to obtain reliable simulation results. In any case we can expect that the inlet structure designed to dissipate turbulent kinetic energy of an activated sludge with low viscosity and low density, will have also a positive effect over a system operated with a higher viscosity or density sludge, as general pattern (not intensity) of flow field is similar even with different density and rheological parameters, apart form the differences already highlighted.

**8.1. Future work**

Considering the complexity of the currents developed in the clarifier and the 3D nature of the turbulence phenomenon, it would be very interesting to extend the simulations study to 3D models. A 3D model would also allow the implementation of a more realistic model for the simulation of sludge scraper. This issue would be important in order to analyse the effect on the scraper on the sludge transport an its relationship with sludge short-circuit.

  On the other hand, although from a qualitative point of view, results shown are in consonance with simulation results of other authors [8, 11] in view of the complexity of the mathematical model used to describe activated sludge sedimentation, as well as the uncertainty about numerous model parameters, an experimental verification of the obtained results should be made in order to verify the quantitative validity of the same.

  Regarding the development of design guidelines, more cases should be studied in order to find the best design solution. Also it is very important to make simulations at different scales and operating conditions in order to find adimensional parameters that allows the implementation of the design guidelines, that fits different model sizes and operating conditions.
9. Bibliography:


10. ANNEX I: Results

**Figure 10.1**
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

**Figure 10.2**
Left: U magnitude contour  Right: k contour
Figure 10.3
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

Figure 10.4
Left: U magnitude contour  Right: k contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Figure 10.9
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

Figure 10.10
Left: U magnitude contour  Right: k contour

Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

Figure 10.11
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

Figure 10.12
Left: U magnitude contour  Right: k contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

---

**Figure 10.15**
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

**Figure 10.16**
Left: U magnitude contour  Right: k contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

**Figure 10.21**
Alpha sludge contour (Left: High TSS scale / Right: Low TSS scale)

**Figure 10.22**
Left: U magnitude contour  Right: k contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
**Figure 10.25**
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

**Figure 10.26**
Left: U magnitude contour  Right: k contour
Figure 10.27
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

Figure 10.28
Left: U magnitude contour  Right: k contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

Figure 10.29
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

Figure 10.30
Left: U magnitude contour  Right: k contour
**Figure 10.31**
Alpha sludge contour (Left High TSS scale / Right Low TSS scale)

**Figure 10.32**
Left: $U$ magnitude contour  Right: $k$ contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.

Figure 10.35
Alpha sludge contour (Left: High TSS scale / Right: Low TSS scale)

Figure 10.36
Left: U magnitude contour  Right: k contour
Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations.
11. **ANNEX II: Mass balance calculation**

The fluxes through the model patches has been calculated with the openFoam tool: 
`postProcessing/FlowratePatch`:

Sum of flow through cell faces (m3/s): 
\[ \text{flowRatePatch} = \int \vec{u} \, d\vec{A} \]

*Total Suspended Solids (TSS)* mass balance has been calculated as follows:

\[ \text{Total TSS flux (kg/h)} = \rho_{\text{sludge}} \frac{1}{A} \int \alpha d\vec{A} \]

In case of “Inlet” and “Outlet_Sludge” patches, as velocity is imposed as a boundary condition as a uniform value \( \Rightarrow \) \( \text{Q}=\text{constant} \), so:

\[ \text{Total TSS flux (kg/h)} = \rho_{\text{sludge}} \frac{1}{A} \int Q \alpha d\vec{A} = \rho_{\text{sludge}} \frac{1}{A} Q \int \alpha d\vec{A} \]

In “Outlet_Water” patch, the boundary conditions imposed are not specified as a fixed velocity, so, this implies that \( \rho_{\text{sludge}} \frac{1}{A} \int Q \alpha d\vec{A} \neq \rho_{\text{sludge}} \frac{1}{A} Q \int \alpha d\vec{A} \)

In any case according to the calculations with data extracted from the analysis with paraview, the difference between considering variable velocity or constant velocity across “Outlet_Water” patch, is less than 0.4% for effluent TSS, which implies an error <0.002% in the sludge recirculation TSS. So considering the small error, this calculation is followed to verify the solid mass balance.
12. ANNEX III: Geometry and meshing example
SetFactory("OpenCASCADE");

// Clarifier general dimensions:
m = 1;

Pendiente_Tolva=1.7;
Pendiente_Dec=1/15;

R_Total=8.2*m;
R_Campana=1.45*m;
H_Campana=2.68*m;
Ap_Campana=0.21*m;
Esp_Campana=0.01*m;
R_T_in=0.2*m;
Esp_Col=0.2*m;
Bp_tolva=0.4*m;
D_T_Fango=0.25*m;
D_T_Agua=0.3*m;
H_Ver=0.5*m;
B_Ver=0.5*m;
H_Lam=0.035*m;
D_Def=0.3*m;
Sub_Def=0.2*m;
Esp_Def=0.01*m;
Hi_tolva=(2*D_T_Fango);
Bi_tolva=Hi_tolva/Pendiente_Tolva;
H_Recta=3.5*m;

R_pend_Dec=R_Total-R_T_in-Esp_Col-Bp_tolva-Bi_tolva;
H_Total=(2*D_T_Fango)+(R_pend_Dec*Pendiente_Dec)+H_Recta;

Lc1 = 0.1;
Lc2 = 0.005;
i=1;
Desp_mall1=0.025;
Desp_mall2=0.05; //Inlet Area
Desp_mall3=0.15; //Carification area
Desp_mall4=0.0; //Central symmetry axis

// Then we define some points and some lines using these variables:
// We use i variable to introduce new points, in between in an easy way

//Geometry
Point(i) = {0,0,0,Lc1}; //Point(1)
i=i+1;
Translate {R_T_in,0,0} { Duplicata { Point[i-1]; } } //Point(2)
i=i+1;
Translate {0,H_Total-0.3-Esp_Col-R_T_in,0} { Duplicata { Point[i-1]; } } //Point(3)
i=i+1;
Translate {Esp_Col,0,0} { Duplicata { Point[i-1]; } } //Point(4)
i=i+1;
p1=newp;
Point(p1) = {R_T_in+Esp_Col,H_Total+H_Lam-H_Campana,0,Lc1}; //Point(5)
i=i+1;

//Points 6 and 8 define influent well lower baffle diameter
Translate {1.2*R_Campana-Esp_Col-R_T_in,0,0} { Duplicata { Point[i-1]; } } //Point(6)
i=i+1;
Translate {0,-Esp_Campana,0} { Duplicata { Point[i-1]; } } //Point(7)
i=i+1;
Translate {-(1.2*R_Campana-Esp_Col-R_T_in),0,0} { Duplicata { Point[i-1]; } } //Point(8)
i=i+1;
Translate {Esp_Col,0,0} { Duplicata { Point[i-1]; } } //Point(9)
i=i+1;
Translate {Bp_tolva,0,0} { Duplicata { Point[i-1]; } } //Point(10)
i=i+1;
Translate {Bi_tolva,Hi_tolva,0} { Duplicata { Point[i-1]; } } //Point(11)
i=i+1;
Translate {R_pend_Dec,R_pend_Dec*Pendiente_Dec,0} {Duplicata {Point[i-1]; }} //Point(12)
i=i+1;
Translate {0,H_Recta-(2*H_Campana/3),0} { Duplicata { Point[i-1]; } } //Point(13)
i=i+1;
Translate {-1.5*B_Ver,H_Ver,0} { Duplicata { Point[i-1]; } } //Point(14)
i=i+1;
Translate {1.5*B_Ver,H_Ver,0} { Duplicata { Point[i-1]; } } //Point(15)
i=i+1;
Translate {0,(2*H_Campana/3)-(2*H_Ver),0} { Duplicata { Point[i-1]; } } //Point(16)
i=i+1;
i=i+1;
Translate {{0,-H_Lam,0}} { Duplicata { Point{i-1}; } } //Point(17)

i=i+1;
Translate {{-D_Def,0,0}} { Duplicata { Point{i-1}; } } //Point(18)
i=i+1;
Translate {{0,-Sub_Def,0}} { Duplicata { Point{i-1}; } } //Point(19)
i=i+1;
Translate {{-Esp_Def,0,0}} { Duplicata { Point{i-1}; } } //Point(20)
i=i+1;
Translate {{0,Sub_Def,0}} { Duplicata { Point{i-1}; } } //Point(21)
i=i+1;
Marca=newp;
Point(Marca) = {{R_Campana ,H_Total+H_Lam ,0 }, Lc1}; //Point(22)
i=i+1;
Translate {{0,-(H_Campana-Ap_Campana),0}} { Duplicata { Point{i-1}; } } //Point(23)
i=i+1;
Translate {{-Esp_Campana,0,0}} { Duplicata { Point{i-1}; } } //Point(24)
i=i+1;
Translate {{0,H_Campana-Ap_Campana,0}} { Duplicata { Point{i-1}; } } //Point(25)
i=i+1;
Marca=newp;
Point(Marca) = {{0,H_Total+H_Lam ,0 }, Lc1}; //Point(26)
//Printf(" Marca=%g", Marca); This is used to print in the screen the actual poin number
i=i+1;
Translate {{0,-H_Lam-D_Def,0}} { Duplicata { Point{i-1}; } } //Point(27)
i=i+1;
Translate {{R_T_in+Esp_Col,0,0}} { Duplicata { Point{i-1}; } } //Point(28)
i=i+1;
Translate {{0,-Esp_Col,0}} { Duplicata { Point{i-1}; } } //Point(29)
i=i+1;
Translate {{-(R_T_in+Esp_Col),0,0}} { Duplicata { Point{i-1}; } } //Point(30)
i=i+1;
Translate {{0,-R_T_in,0}} { Duplicata { Point{i-1}; } } //Point(31)
i=i+1;
ult_P=newp-1;
//This is used to print in the screen the actual poin number
Printf(" Numero de puntos contorno exterior ult_P=%g", ult_P);
k=1;
//Internal point to make the structured meshAhora creamos los puntos internos para el mallado estructurado
k=k+1;
Translate {{-Desp_mall1,Desp_mall4,0}} { Duplicata { Point{k}; } } //Point 32
k=k+1;
Translate {-Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point 33
k=k+1;
Translate {Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point 34
k=k+1;
Translate {Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point 35
k=k+1;
Translate {Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point 36
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(37)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(38)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(39)
k=k+1;
Translate {-Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(40)
k=k+1;
Translate {-Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(41)
k=k+1;
Translate {-Desp_mall1,Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(42)
k=k+1;
Translate {-Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(43)
k=k+1;
Translate {-Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(44)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(45)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(46)
k=k+1;
Translate {-Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(47)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(48)
k=k+1;
Maska=newp-1;
Marca=newp-1;
Printf(" Marca=%g", Marca);
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(49)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(50)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(51)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(52)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(53)
k=k+1;
Translate {Desp_mall1,-Desp_mall1, 0} { Duplicata { Point{k}; } }  //Point(54)
185  \text{k=k+1;}  \\
186  \text{Translate \{-\text{Desp\_mall1},-\text{Desp\_mall1}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
187  \text{ //Point(55)}  \\
188  \text{k=k+1;}  \\
189  \text{Translate \{0,-\text{Desp\_mall1}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
190  \text{ //Point(56)}  \\
191  \text{k=k+1;}  \\
192  \text{Translate \{\text{Desp\_mall4},\text{Desp\_mall1}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
193  \text{ //Point(57)}  \\
194  \text{k=k+1;}  \\
195  \text{Translate \{\text{Desp\_mall1},\text{Desp\_mall1}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
196  \text{ //Point(58)}  \\
197  \text{k=k+1;}  \\
198  \text{Translate \{\text{Desp\_mall4},-\text{Desp\_mall1}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
199  \text{ //Point(59)}  \\
200  \text{k=k+1;}  \\
201  \text{ult\_P1=newp-1;}  \\
202  \text{Printf(" Ultimo numero de punto contorno mallado estructurado ult\_P1=%g", ult\_P1);}  \\
203  \text{//We create another inside layer to control size of internal mesh}  \\
204  \text{k=ult\_P+1;}  \\
205  \text{//Ahora creamos otra capa mas para controlar el mallado interior}  \\
206  \text{//Translate \{-\text{Desp\_mall2},0, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
207  \text{ //Point 61}  \\
208  \text{k=k+1;}  \\
209  \text{Translate \{-\text{Desp\_mall2},\text{Desp\_mall2}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
210  \text{ //Point 62}  \\
211  \text{k=k+1;}  \\
212  \text{Translate \{-\text{Desp\_mall2},\text{Desp\_mall2}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
213  \text{ //Point 63}  \\
214  \text{k=k+1;}  \\
215  \text{Translate \{-\text{Desp\_mall2},\text{Desp\_mall2}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
216  \text{ //Point 64}  \\
217  \text{k=k+1;}  \\
218  \text{Translate \{-\text{Desp\_mall2},\text{Desp\_mall2}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
219  \text{ //Point 65}  \\
220  \text{k=k+1;}  \\
221  \text{Translate \{-\text{Desp\_mall2},\text{Desp\_mall2}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
222  \text{ //Point 66}  \\
223  \text{k=k+1;}  \\
224  \text{Translate \{-\text{Desp\_mall3},\text{Desp\_mall3}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
225  \text{ //Point 67}  \\
226  \text{k=k+1;}  \\
227  \text{Translate \{-\text{Desp\_mall3},4^{*}\text{Desp\_mall3}, 0\} \{ Duplicata \{ Point(\text{k}); \} \} }  \\
228  \text{ //Point 68}  \\
229  \text{k=k+1;}  \\
230  \text{k=k+1;}
Translate {-2*Desp_mall3,4*Desp_mall3,0} { Duplicata { Point{k}; } }  //Point 71
k=k+1;

Translate {-2*Desp_mall3,4*Desp_mall3,0} { Duplicata { Point{k}; } }  //Point 72
k=k+1;

Translate {-2*Desp_mall3,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 73
k=k+1;

Translate {-2*Desp_mall3,0,0} { Duplicata { Point{k}; } }  //Point 74
k=k+1;

k=k+1;

Translate {-Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 76
k=k+1;

Translate {-Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 77
k=k+1;

Translate {Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 78
k=k+1;

Translate {Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 79
k=k+1;

k=k+1;

Translate {-Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 80
k=k+1;

Translate {-Desp_mall3,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 81
k=k+1;

Translate {Desp_mall3,-Desp_mall3,0} { Duplicata { Point{k}; } }  //Point 82
k=k+1;

Translate {Desp_mall3,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 83
k=k+1;

Translate {-Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 84
k=k+1;

Translate {Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 85
k=k+1;

k=k+1;

Translate {Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 86
k=k+1;

Translate {Desp_mall2,Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 87
k=k+1;

Translate {Desp_mall2,Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 88
k=k+1;

Translate {Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 89
k=k+1;

Translate {Desp_mall2,-Desp_mall2,0} { Duplicata { Point{k}; } }  //Point 90
k=k+1;

Translate {Desp_mall2,0,0} { Duplicata { Point{1}; } }  //Point(91)
k=k+1;

ult_P2=newp-1;
// Print in screen for reference
Printf(" Ultimo numero de punto mallado interno ult_P2=%g", ult_P2);

// Rotate all the generated points in order to be able to use wedge boundary conditions in openFoam
For i In {1:ult_P2}
  Rotate {{0, 1, 0}, {0, 0, 0}, -Pi/180} {
    Point[i];
  }
EndFor

//Macro create lines see tutorial t5.geo y t8.geo
Macro CrearLineas
  l1 = newl; // "newl" gives number+1, where number is last existing line
  Line(l1) = {t,t+1};
Return

// We can use a `For' loop to generate lines:
fin=ult_P-1;
For t In {1:fin}
  // We call the macro:
  Call CrearLineas ;
EndFor

//This is the last line that closes the bucle
Line(fin+1) = {fin+1,1}; //Create a newline
ult_L=newl-1;

Printf(" Numero de lineas contorno exterior ult_L=%g", ult_L);

//**************************IMP LINES CAN NOT BE DUPLICATED OTHERWISE IT GOES TO ERROR
// We can use a `For' loop to generate lines:
inicio=ult_P1+1;
fin=ult_P1-1;
For t In {inicio:fin}
  // We call the macro:
  Call CrearLineas ;
EndFor
ult_L1=newl-1;
Printf("Ultimo numero de linea contorno mallado estructurado ult_L1=%g", ult_L1);

//Create the lines that connects external lines and the lines for the structured mesh
// in order to be able to generate the structured mesh
fin=ult_P-1;
Macro CrearLineas1
  l1 = newl; Line(l1) = {t,t+fin};
  thelines[t] = newl ;
Return
For t In {2:fin}
  // We call the macro:
  Call CrearLineas1 ;
EndFor
ult_L2=newl-1;
Printf(" Ultimo numero de linea contorno mallado estructurado ult_L2=%g", ult_L2);

// We can use a 'For' loop to generate lines:
inicio=ult_P1+1;
fin=ult_P2-1;
For t In {inicio:fin}
  // We call the macro:
  Call CrearLineas ;
EndFor

//This is the last line that closes the internal bucle
l1=newl;
Line(l1) = {ult_P2, ult_P1+1}; //genero la linea nueva
ult_L3=newl-1;
Printf(" Ultimo numero de linea contorno interno ult_L3 =%g", ult_L3);

//Lines to close the internal loop
l1=newl;
Line(l1) = {ult_P1, 1}; //new line
l1=newl;
Line(l1) = {1, ult_P2}; //new line
l1=newl;
Line(l1) = {ult_P1+1, ult_P+1}; //new line
Las lineas de cierre del bucle intermedio son \%g, \%g y \%g, L_bucle_intermedio_1, L_bucle_intermedio_2, L_bucle_intermedio_3;

Loops and areas for structured mesh

Index=newll;
Printf( "newLine=%g", index);
i=2; //1;
j=i+ult_L1-1;
k=i+ult_L-1;
cont=1;
Printf( "Area inicio mallado estructurado=%g", cont);
For t In {index:index+(2*ult_L)-8}
    //Printf(" i=%g", i);
    //Printf(" j=%g", j);
    //Printf(" k=%g", k);
    //Printf(" t=%g", j+1);
    If (i!=(ult_L-5)) //Me salto el area que caeria en el eje
        Curve Loop(t) = {i,j,k,j+1};
        Plane Surface(cont) = {t};
        cont=cont+1;
    EndIf
    i=i+1;
    j=j+1;
    k=k+1;
    t=t+1;
EndFor
//in each round 2 number are used one for the "curve loop" and the other for the "Palne surface"
cont=cont-1;
Printf(" Area fin mallado estructurado=%g", cont);

//Internal area
//Create a vector with lines number that I'm going to use in cueve loop
//Index start in 0

j=0;
For k In {ult_L2+1:ult_L3}
    //Printf(" k=%g", k);
    CL_array[j]=k;
    j=j+1;
EndFor

//Middle area with hole

j=0;
For k In {ult_L+1:ult_L1}
    CL_array0[j]=k;
    //Printf(" CL_array0[%g]=%g",j, CL_array0[j]);
    j=j+1;
EndFor
For k In {ult_L3+1:ult_L3+2}
    CL_array0[j]=k;
    //Printf(" CL_array0[%g]=%g",j, CL_array0[j]);
    j=j+1;
EndFor

//This loop goes in reverse direction
r=ult_L3-1;
For k In {ult_L2+1:ult_L3-1}
460           CL_array0[j]=r;
461           //Printf(" CL_array0[%g]=%g",j, CL_array0[j]);
462           r=r-1;
463           j=j+1;
464           EndFor
465
466       CL_array0[j]=ult_L3+3;
467       //Printf(" CL_array0[%g]=%g",j, CL_array0[j]);
468
469       //We use previous vector to make the loopp
470       t=newll;
471       cont2=cont1+1;
472       Curve Loop(t) = CL_array0[];
473       Plane Surface(cont2) = {t};
474       Printf(" Curve loop%g", t);
475       Printf(" Area interna intermedia cont2=%g", cont2);
476
477       //**********************************************************Mesh with transfinite
478       // Instead of using the umber of point to divide each line we use a size of division
479       // To do this we need to know the coordenates of each point
480       // This is done: c[] = Point{1}; // c[] contains the coordinates 1,2,3
481       // Coordenate over we fi x the distance, 0=x, 1=y, 2=z
482
483       //Mesh of external and middle lines
484       inicio=ult_L1+1;
485       For i In {inicio:ult_L1}
486       j=i+1;
487       delta=0.05*m;       //Mesh size
488       If (i==40+2)
489           delta=0.02*m;       //Mesh size
490           EndIf
491       If (i==41+2)
492           delta=0.02*m;       //Mesh size
493           EndIf
494       If (i==44+2)
495           delta=0.005*m;       //Mesh size
496       EndFor
EndIf
If (i==43+2)
delta=0.02*m;  //Mesh size
EndIf
If (i==45+2)
delta=0.02*m;  //Mesh size
EndIf
If (i==46+2)
delta=0.02*m;  //Mesh size
EndIf
If (i==47+2)
delta=0.02*m;  //Mesh size
EndIf
If (i==48+2)
delta=0.02*m;  //Mesh size
EndIf
c1[] = Point{i}; c2[] = Point{j};
Ndiv=Round(Sqrt(((c1[0]-c2[0])^2)+((c1[1]-c2[1])^2))/delta);
Transfinite Curve{i} = Ndiv Using Progression 1;  //Bump 0.15;
Transfinite Curve{i-ult_L+1} = Ndiv Using Progression 1;  //Bump 0.15;
EndFor

//Mesh of connexion lines between external and middle lines, fix the thickness of mesh
j=2;
k=j+ult_L-1;
par=1;
For i In {ult_L1+1:ult_L2}
  If (par==2)
    par=par-2;
  EndIf
  delta=0.01*m;  //Mesh size
c1[] = Point{j}; c2[] = Point{k};
Ndiv=3;
Transfinite Curve{i} = Ndiv Using Progression 1;  //Bump 0.15;
par=par+1;
EndFor

//Internal lines meshing
inicio=ult_L2+1;
in=ult_L3;
Printf(" inicio=%g", inicio);
Printf(" fin=%g", fin);
j=ult_P1+1;
k=j+1;

For i In {inicio:fin}

    delta=0.1*m; //Mesh size
    If (i==98+5)
        delta=0.02*m; //Mesh size
    EndIf
    If (i==99+5)
        delta=0.005*m; //Mesh size
    EndIf
    If (i==100+5)
        delta=0.02*m; //Mesh size
    EndIf
    If (i==101+5)
        delta=0.02*m; //Mesh size
    EndIf
    If (i==102+5)
        delta=0.02*m; //Mesh size
    EndIf

c1[] = Point{j}; c2[] = Point{k};
Ndiv=Round(Sqrt(((c1[0]-c2[0])^2)+((c1[1]-c2[1])^2))/delta);
Transfinite Curve{i} = Ndiv Using Progression 1; //Bump 0.15;

j=j+1;
If (j==ult_P2)
    k=ult_P1+1;
EndIf
If (j<ult_P2)
    k=j+1;
EndIf

EndFor

For k In {1:cont}
    Transfinite Surface{k};
    Recombine Surface {k};
EndFor
// We make the strusion to generate the sector
fin=cont2;
For k in {1:fin}
    Extrude {{0, 1, 0}, {0, 0, 0}, (Pi/90) } {
        Surface{k}; Layers{1}; Recombine;
    }
EndFor

// We Define the surfaces and the physical volumen

// +
Physical Surface("Inlet") = {220, 191, 250, 33};
// +
Physical Surface("Inlet_pipe") = {29+1};
// +
Physical Surface("Column") = {34+1, 39+1, 59+1, 152+1, 158, 147+2};
// +
Physical Surface("Outlet_Sludge") = {64+1};
// +
Physical Surface("Outlet_Water") = {100};
// +
Physical Surface("Free_Surface") = {125, 105, 145};
// +
Physical Surface("Out_Wall") = {79+1};
// +
Physical Surface("Weir_Wall") = {84+1, 89+1, 95};
// +
Physical Surface("Deflectora") = {114+1, 109+1, 109+1+10};
// +
Physical Surface("Pend_Dec") = {74+1};
// +
Physical Surface("Tolva") = {69+1};
// +
Physical Surface("Campana") = {134+1, 140, 129+1, 54+1, 44+1, 49+1};
// +
Physical Surface("Simetry_Plane1") = {1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29};
// +
Physical Volume("fluidVolume") = {1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29};

//IMP after export the mesh a ASCII 2 with the name "DecCircular_rev6_1_refined10.msh" execute
//Merge "DecCircular_rev6_1_refined10.msh";
//Coherence Mesh;
Erratum

In page 25, Figure 5.1 must be replaced by the following figure:

In page 24 the following text

“As it is shown in Figure 5.1 the system is operating very close to the critical condition, so the safety margin is almost zero. Under these conditions, depending on the hydraulic structures design the system can suffer a failure condition, with the lack of capacity for enough sludge removal.”

must be replaced by:

“As it is shown in Figure 5.1 the system is operating below the critical condition. Under these conditions, according to solid flux theory the system should not suffer a critical failure due to under-sizing.”

Secondary clarifier inlet/outlet structures optimization by the use of CFD simulations