Numerical Investigation of the Flow Field inside a Manhole-Pipe Drainage System

Md Nazmul Azim Beg  
*University of Coimbra*, mnambeg@uc.pt

Rita Carvalho  
*University of Coimbra*, ritalmfc@dec.uc.pt

Pedro Lopes  
*University of Coimbra*

Jorge Leandro  
*University of Coimbra*

Nuno Melo  
*University of Coimbra*

Follow this and additional works at: [https://digitalcommons.usu.edu/ishs](https://digitalcommons.usu.edu/ishs)  
Part of the [Hydraulic Engineering Commons](https://digitalcommons.usu.edu/ishs)

**Recommended Citation**

Numerical Investigation of the Flow Field inside a Manhole-Pipe Drainage System

Md Nazmul Azim Beg¹,², Rita F. Carvalho¹,², Pedro Lopes¹,², Jorge Leandro¹,² and Nuno Melo¹

¹ Dep. of Civil Engineering, Faculty of Science and Technology,
University of Coimbra
Portugal

² MARE - Marine and Environmental Sciences Centre & IMAR - Institute of Marine Research,
University of Coimbra
Portugal
E-mail: mnabeg@uc.pt

ABSTRACT

Urban drainage networks contain a large number of structures. The manhole is the most common and the most important as it connects pieces of sewer pipes to form sewer networks. Understanding the hydraulics of a manhole-pipe drainage system is important as this may sometimes become the bottlenecks of the sewer systems. In this study, the flow structure and flow hydraulics of a specific manhole-pipe sewer system was analysed numerically in view of further research to investigate sediment and suspended solid transport in the system. The numerical model was compared with discharge and water pressure/depth data from an experimental model. Different discharges and water levels at the inlet pipe were applied, and corresponding change of flow hydraulics were analysed. Two scenarios were tested: (1) free surface flow and (2) pressurized flow condition in the pipe. In the numerical analysis, the k-ε turbulent model was used within open source CFD tool OpenFOAM®. The numerical results showed similar flows and water pressure levels to that of the experimental work. Different flow patterns were observed in the manhole at different discharges. This flow pattern will give further insight in assessing pollutant flow inside the system.

Keywords: Urban drainage systems, Manhole-pipe system, OpenFOAM®, velocity profiles, sewer junction

1. INTRODUCTION

Urban drainage is composed of several linking elements. The serviceability of drainage systems depends on their efficiency. One of the most common element of a drainage system is a manhole. It collects storm drainage from gullies (or gutters) and intake pipes and conveys its discharge to its outlet pipe. After the rainfall, the surface runoff passes through the paved and unpaved urban surfaces and collects sediment as one of the pollutants (Deletic 1998). During flood events, manholes sometimes may become pressurised. The turbulence may affect the head loss, hydraulic efficiency, and discharge coefficient of the flow (Granata et al. 2011). Understanding of the flow pattern is important for conducting storm water quality investigations.

To interpret the flow behaviour in practical scenarios, use of experimental and numerical models is a well-recognized procedure. Stovin et al. (2008) have shown a number of possible methods to validate CFD model, while Rubinato (2015) has shown uses of scaled models to quantify hydraulic losses in a manhole. This study has been conducted to investigate the flow pattern of a manhole-pipe drainage system using three dimensional CFD tools OpenFOAM®. The numerical work has been validated with discharge and water pressure/depth data from a previously performed experimental work. Both free surface flow and pressurised flow were simulated. The scope of the present paper is to evaluate the performance of the CFD model to capture the overall hydraulic features of the flow. These results may be used to aid the simulation of a pollutant transport model in the drainage system.

The paper starts by presenting the experimental installation and numerical model used in the section titled “Methodology,” followed by the validation of the numerical model in the section “Results and Discussion.” Later in the same section, different findings from the numerical model are presented.
2. METHODOLOGY

2.1. Experimental Model

Experimental work was performed in the model setup established at the hydraulic lab of University of Coimbra (Figure 1). The experimental setup has been described in detail by Santos et al. (2008), Leandro et al. (2009), Carvalho et al. (2013), and Leandro et al. (2014). This is a Multi-Link-Element (MLE) setup with a flume and two gullies connected with two manholes. Only the two manholes and the connecting pipe were used for this experiment. The first manhole (made of acrylic) does not have a guided flow channel, while the second manhole has a more complex interior geometric shape with a guided flow channel, following a pre-built commercial design. Both types can be found in Portuguese drainage systems. The connecting pipe has a length of 8.5 m, with a diameter of 0.3 m. The system is equipped with a closed circuit connecting two reservoirs of 36 m$^3$ (main reservoir) and 32 m$^3$ (secondary reservoir). This is supplied by a system of four circulation pumps and controlled by several valves, and electrical control systems. A combination of valves can control the water depth/pressure in the manhole-pipe system. The complete system is equipped with an SCADA system (Supervision, Control, and Data Acquisition) that allows for the operation and monitoring of flow rates and pressures. The system also controls the pumps as necessary.

Figure 1: Multi-Link-Element experimental model setup at the hydraulic lab of the University of Coimbra

A schematic diagram of the experimental setup in the work can be viewed in Figure 2. For the data collection of the experimental model, different pressure sensors were installed at the bottom of both manholes (P11, P23) and through the length of the pipe (P12, P18 and P22) (Figure 2). The sensors measured piezometric pressures for both free surface and pressure flow conditions, which were converted to piezometric head considering the bottom of Manhole 1 as the zero datum. Pressure head differences along the pipe were used to determine conduit head losses.

A number of experimental runs were performed keeping the outlet valve opened at 30% (low flow), 40% (medium flow), and 60% (high flow) to observe different flow velocities inside the pipe and the manhole for a particular surcharge water depth. All discharges and corresponding water pressures throughout the system were recorded after reaching a steady flow condition. 18 of these experimental run results were used for numerical investigation in this study.
Figure 2: Schematic diagram of the experimental and numerical setup with different locations of the pressure sensors and boundary locations of the numerical model (in orange font) (upper panel) and Computational meshes at some selected locations of the numerical model (lower panel)

2.2. Numerical Model

The numerical model intends to reproduce scenarios similar to the experimental model. The main purpose of numerical modelling is the analysis of the flow characteristics and change of flow patterns when the water enters to the manhole from pipe and vice versa. Another purpose is to check the flow pattern at different pressure and flow velocity combinations. In this study, only manhole 1 with inlet and outlet pipes was evaluated numerically; this manhole type is more common in Portuguese drainage systems. The numerical modelling domain is chosen from $x=0$ m to $x=9.25$ m, where the inlet and pressure transducer P18 was located, respectively (Figure 2). The selected manhole for the numerical study did not have a guided flow channel. Numerical study of the other manhole will be done at a later stage.

OpenFOAM® toolbox version 2.3.0 is used in this study. The model simulation has to deal with both liquid and gas phases in order to capture the water surface properly. For this case, the solver interFoam is chosen as it can be a powerful tool to predict the free surface flow for sharp interfaces and velocity patterns (Lopes et al. 2015), assuming that the air effect on the flow can be neglected. This solver uses a single set of Navier-Stokes equations, where the velocity is shared by both phases and a Volume of Fluid (VOF) method (Hirt and Nichols 1981) capture the free-surface position. PISO algorithm was chosen for pressure-velocity coupling as it is known for less computational effort in comparison to SIMPLER and SIMPLEC algorithm (Versteeg and Malalasekera 1995). One of the most used and reliable turbulence modelling approaches in OpenFOAM® is Reynolds Average Navier-Stokes (RANS), which was used to model the turbulence of the flow because it has less computational cost than Large Eddy Simulation (LES) (Versteeg and Malalasekera 1995). This turbulence calculation approach uses two closure equations for $k$ (Turbulent Kinetic Energy) and $\varepsilon$ (Energy dissipation). The unsteadiness in flow is averaged out in this model and regarded as part of the turbulence (Furbo et al. 2009).

All the numerical results have been acquired at steady state conditions. 18 different scenarios have been simulated, covering both high and low flow conditions as well as high-pressure and low-pressure conditions. Some of the simulations have free surface flow, and some have pressurized flow in the pipe. The work flow for setting up the numerical model is described in the following paragraphs.
2.2.1. Mesh Generation

The computational mesh was constructed using the OpenFOAM® built in mesh generating tool `blockMesh` and `snappyHexMesh` utilities. The `blockMesh` utility divides the model domain into several three-dimensional, hexahedral blocks, while `snappyHexMesh` creates hexahedra and split hexahedra triangulated surface geometries. This utility carves out the smooth outline from a previously created structured mesh with a surface given in Stereolithography (STL) or Wavefront Object (OBJ) format. The STL surface was produced using an open source integration platform for numerical simulations and mesh generator SALOME v.7.5.1. The `blockMesh` plus `snappyHexMesh` create a quasi-regular mesh but variable grid spacing in singularities. For the manhole, all the grid elements featured cubes that measured 0.025m on all sides. For the inlet and outlet pipes, the longitudinal mesh size varied from 0.025m to 0.1m, keeping minimum grid spacing near the manhole-pipe junction and maximum spacing near the distal ends of the inlet and outlet pipes. The transverse mesh sizes were 0.025 m. Boundary meshes were further refined by a factor of two in all three directions (x, y and z) using `castellatedMesh` and a boundary layer is added using `addLayers`. Furthermore, each boundary mesh was implicitly wrapped and smoothed by merging the faces using `snap`.

Thus, the whole computational domain is composed of around 210,000 computational meshes. Some quality parameters of the prepared mesh are given in Table 1. Figure 2 shows some parts of the computational mesh.

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Maximum Aspect ratio</th>
<th>Max skewness</th>
<th>Max non-orthogonality</th>
<th>Min face area (m²)</th>
<th>Max face area (m²)</th>
<th>Min volume (m³)</th>
<th>Max volume (m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>21.87</td>
<td>1.959</td>
<td>65</td>
<td>3.18x10⁶</td>
<td>3.37x10³</td>
<td>8.92x10⁹</td>
<td>9.22x10⁵</td>
</tr>
</tbody>
</table>

2.2.2. Boundary Conditions

Three open boundaries were used for the computational domain. They were Inlet, Atmosphere, and Outlet (Figure 2). The inlet boundary only allows incoming flow, while the outlet allows only outward flow. The data for the boundaries have been measured from the experimental model. The upstream boundary values were chosen from the measured discharge at the inlet, while the downstream pressure values were obtained from collected data at P18 pressure transducer.

For a RANS model setup, each of the open boundaries require six different Boundary Conditions (BC). They are `alpha.water` (water fraction in each cell volume), `U` (velocity vector in Cartesian domain), `p_rgh` (relative bottom pressure corresponding to datum), `k` (turbulent kinetic energy), `ε` (energy dissipation), and `nut` (turbulent viscosity). OpenFOAM® allows many ways to apply each boundary type. It has been challenging to find the best combination of the boundary properties for this study as it has two phases at the boundary. Fixed velocity / discharge boundary conditions `U` (Neumann BC) was prescribed at the inlet, while the outlet BC was prescribed as fixed `p_rgh` pressure (Dirichlet BC). The atmosphere BCs were prescribed as `zeroGradient` velocity so that air can be exchanged. The relative pressure condition at the atmosphere was kept as zero. The wall BCs at the manhole and pipe walls were kept as no-slip condition (i.e., `velocity = 0`). The turbulent modelling parameters (`k`, `ε` and `nut`) were calculated for each case and prescribed at the inlet boundary using the equations as described in FLUENT manual (ANSYS Ins 2009). The Turbulence Intensity (I) and the empirical constant (Cµ) were chosen as 0.05 and 0.09, respectively. Pre-analysis of the model showed that to capture the boundary layer flow velocity, the mesh size would have to be in the range of 10⁻⁴ m at the boundary wall. For efficiency with computational time, the mesh size could not be kept that small. Hence, all the internal wall boundary conditions for turbulence closure (k, ε and nut values) were chosen as `wallFunction`. Use of `wallFunction` eliminates the necessity of possible mesh grading as flow in the near boundary wall cell is not resolved but modelled (Greenshields 2015).

2.2.3. Simulation of the Models

After preparing all the mesh and the boundary setups, the model was ready to run. The simulation was done using adjustable run time so that the model could adjust the time step itself. The maximum courant number was set to 0.5.
Each simulation was run for 40 s to reach a steady state condition. For better computational time efficiency, the model was decomposed into 4 parts to run in MPI mode with 4 processors. A multi core Core i7 computer was used, which required approximately 24 hours to complete each simulation.

3. RESULTS AND DISCUSSION

3.1. Validation of the Numerical Model

During the experimental model run, pressure levels were recorded at different locations in the pipe as well as the bottom centre of the manhole. The numerical model was validated with experimental data of flow depth / pressure head and discharge inside the structure so that flow characteristics observed from the numerical results represent similar scenarios created in the experimental model. The comparison was checked with the pressure data at transducer P23, P22, and P18 (see Figure 2). Figure 3 shows the comparison between experimental and numerical simulations in a pressure head vs discharge plot. The dot markers of different colours show data from the experimental model while the triangles show data from the numerical model.

When the depth at P23 / Manhole 1 stays below 0.3 m, the flow remains as free surface flow. As the depth of flow starts increasing, the upstream end will pressurize before the downstream end, which continues to exhibit free surface flow; this situation is termed as transition flow. When the flow depth at P23 exceeds 0.4 m, the flow throughout the length of the structure becomes pressurised.

![Pressure Head vs Discharge](image)

Figure 3: Pressure level vs Discharge in the two manholes from experimental and numerical simulation

It can be seen from Figure 3 that the model has loss of pressure head from P23 to P18. The head loss increases with the increase of upstream discharge. It can be seen that the head loss also increases with increase of pressure in the flow. The head loss is higher in the pressurised flow than that of the free surface flow.

A percentage of error from the pressure head data comparison has been generated (Figure 4). The positive error means the predicted pressure in the numerical model is higher than that of experimental model and vice versa.
It can be seen from Figure 4 that the percentage of absolute error was always less than 7%, which indicates a good agreement with the experimental data.

3.2. Velocity Profile

An analysis of the three dimensional flow profile inside the manhole was examined. In Figure 5, three dimensional velocity profiles can be seen from Sim 3_9. The figure at the left shows the velocities in xz and yz planes, whereas the figures at the right show horizontal velocity fields at different vertical plane.

Figure 5 right panel shows that the manhole has different horizontal velocity structures at different water depths. The horizontal flow velocity is the highest between Z=0.1 m and 0.4 m, due to strong flow influence from the inlet pipe. At this depth range, a strong flow flowing toward the right can be observed at the central zone (near y=0 line). The flow turns to the opposite direction near the perimeter, creating a horizontal vortex between the perimeter and the central zone. The location of the centres of the vortex varies with the depth of flow. It can also be seen that at the surface (at Z=1.14 m), the dominant flow is no longer in the positive x-direction (downstream), but in the upstream direction. Some vertical vortex can be observed at the left panel at depth Z=0.8 m to Z= 0.9 m.
3.3. Flow Comparison for Different Inflow

The various simulations were used to check and compare the flow behaviour inside the manhole due to different inflow. Sim 1_3, Sim 2_6, and Sim 3_9 were chosen for these comparisons as these three scenarios have a similar pressure level but different inflow. The inflows in these simulations are 44.73 l/s, 58.53 l/s, and 130.72 l/s, respectively; while the corresponding flow depths at the manhole are 1.24 m, 1.24 m, and 1.14 m.

To compare the vortex eye location between the above mentioned three simulation results, the velocity contours at the long section of the structure are shown in Figure 6. The blue-to-red contour pallet represents the magnitude of the velocity, light green arrows show the velocity direction, and the red dot shows the location of the vortex eye.

![Figure 6: Velocity contour and eye of the vortex from Sim 1_3 (left panel), Sim 2_6 (middle panel) and Sim 3_9 (right panel)](image)

Figure 6 shows that the location of the vortex eye is different in each case. The vortex eye was located at \( z=0.78 \) m, 0.9 m, and 0.87 m, which are 37%, 27.4%, and 24% of the flow depth for the water surfaces, respectively. It can also be seen that the vortex eyes move marginally to the right direction (away from the inlet) with the increase of flow.

The stream line path can give good insight into the possible flow path of particulate suspended sediment transport inside the manhole-pipe system. The stream lines presented in Figure 7 show the possible path lines from Sim 1_3, Sim 2_6, and Sim 3_9. The figure at the left shows streamlines seeding from the bottom of the manhole, and the right panel shows streamlines seeding from the centre of the inlet pipe.

![Figure 7: Flow path inside the manhole in Sim 1_3, Sim 2_6 and Sim 3_9 showing seeding from the bottom of the manhole (left panel) and from the pipe (right panel)](image)
Figure 7 indicates that two types of streamlines have different pathways. By analysing the streamlines starting at the bottom centre of the manhole (left panel), it can be said that with the Sim 3_9 scenario with higher flow, it is likely that an at-rest particle at the manhole bottom has a higher probability of approaching the free surface in the manhole than go out through the outlet pipe directly. On the other hand, in the right panel, it can be seen that most of the particles coming through the inlet pipe do not stay inside the manhole. Most of them go out directly through the outlet. However, the Sim 3_9 scenario shows slightly more chance of mixing inside the manhole.

### 3.4. Flow Analysis under Different Surcharge Pressure

Three numerical simulation results are compared to analyse the change of flow pattern for different surcharge pressure inside the manhole: Sim 1_3, Sim 2_4, and Sim 3_3. These three simulations have similar range of inflow: 44.73 l/s, 43.69 l/s, and 41.72 l/s, respectively. The pressure levels in these three scenarios were 1.24 m, 0.59 m, and 0.32 m, respectively. Sim 1_3 and Sim 2_4 have pressure pipe flow, while Sim 3_3 has free surface flow. Figure 8 shows the velocity contour from the three numerical simulations.

![Velocity contour from Sim 1_3 (left), Sim 2_4 (middle), Sim 3_3 (right)](image)

Figure 8: Velocity contour and eye of the vortex from Sim 1_3 (left panel), Sim 2_4 (middle panel) and Sim 3_3 (right panel)

Figure 8 shows that with lower depth of water in Sim 2_4 and Sim 3_3, the simulation does not produce any vortex. The flow vectors in Sim 2_4 and Sim 3_3 are mostly parallel to the dominant flow towards x direction. It is likely that the vortex forms if the water depth increases above a certain level.

The streamline path under the above mentioned three scenarios was analysed in Figure 9.

![Flow path from Sim 1_3, Sim 2_4, and Sim 3_3](image)

Figure 9: Flow path inside the manhole in Sim 1_3, Sim 2_4, and Sim 3_3 showing seeding from the bottom of the manhole (left panel) and from the pipe (right panel)
It can be seen from Figure 9 that like Figure 7, the streamline path starting from the bottom of the manhole is different from the path starting at the inlet pipe. Analysis of the streamline starting from the bottom of the manhole (left panel) shows that the three scenarios have different path lines. Most of the streamlines in Sim 3_3 circulate inside the pipe rather than going out through the outlet. While in the other two simulations, comparably more streamlines pass through the outlet. Streamlines coming through the inlet pipe (right panel) show that in Sim 2_4 and Sim 3_3, with lower depth of flow; most of the streamlines pass directly through the outlet. While in Sim 1_3, some streamlines coming from the pipe have comparably higher chance to reach at the free surface of the manhole as well as mixing inside the manhole.

3.5. Comparison of Pressure Level at the Manhole Bottom

As the flow enters the manhole and passes through the outlet, the velocity creates a pressure gradient on the manhole floor, which may create an uplift force on any pollutant lying on the floor. A comparison of the pressure variation on the manhole floor was analysed. Five case scenarios are in Figure 10, namely Sim 1_3, Sim 2_6, Sim 3_9, Sim 2_4, and Sim 3_3. As mentioned earlier, Sim 1_3, Sim 2_6, and Sim 3_9 have a similar pressure range with different inflow, while Sim 1_3, Sim 2_4, and Sim 3_3 have a similar range of discharge with different pressure levels at the manhole.

![Figure 10: Pressure variation at the manhole bottom from left: Sim 1_3, Sim 2_6, and Sim 3_7, Sim 2_4 and Sim 3_3](image)

Figure 10 demonstrates that the first three simulations, having a similar range of pressure, showed a similar pressure gradient map at the manhole bottom, keeping marginally higher pressure near the inlet and highest pressure near the outlet compared to the central zone. However, each of them has a different range of pressure variation at the bottom. In Sim 1_3, the difference between the highest and the lowest pressure is found in the range of 30 Pa; while this difference rises to the range of 60 Pa and 500 Pa in Sim 2_6 and Sim 3_9, respectively. It is likely that the difference between the highest and the lowest pressure increases with the increase in flow velocity in the inlet.

On the other hand, comparing Sim 1_3, Sim 2_4, and Sim 3_3, where the discharge has similar range, pressure maps at the bottom are not similar. Although these three simulations have a similar range of discharges, having different pressures/flow depths creates different average flow velocities inside the manhole. The average flow velocity is the highest in Sim 3_3 and the lowest in Sim 1_3. The range of pressure variation in Sim 2_4 and Sim 3_3 are in the range of 70 and 90 Pa, respectively, which shows that the pressure variation at the manhole floor increase with higher flow velocity.

4. CONCLUSION

This study provides the first step in the numerical assessment of flow behaviour of a manhole in an urban drainage system. The study presents replication of experimental data using a three dimensional CFD tool, OpenFOAM®. A standard k-ε turbulence modelling approach was used to capture the turbulence phenomena. The flow behaviour was analysed from the numerical model results.
The numerical results showed good agreement with experimental pressure data (less than 7% error). Both free surface flow and pressure flow were analysed. It has been observed that the pressure flow inside the manhole has different horizontal velocity profiles at different water depths. Surcharge flow inside the manhole creates different vortexes at different water depths, which was simulated in the numerical model. The location of the vortex centre changes with the inflow. With higher inflow, the centre rises closer to the surface and moves further from the inlet. It has also been seen that different pressure levels and flow velocities create different pressure gradients in the manhole floor, which might be responsible for different uplift forces to any particular pollutant resting at the manhole floor. Moreover, different flow at the inlet pipe also created different streamline paths inside the manhole. It has also been observed that the stream flow starting from the manhole bottom and coming through the inlet pipe follows different path lines depending on their inflow intensities.

The work will be further continued to develop a numerical approach to better understand the phenomena of particulate transport inside the manhole-pipe drainage system using a transport equation. Datasets obtained in this study will be used to calibrate/validate a numerical model created with the open-source toolbox OpenFOAM®.

5. ACKNOWLEDGEMENTS

The work presented is part of the QUICS (Quantifying Uncertainty in Integrated Catchment Studies) project. This project has received funding from the European Union’s Seventh Framework Programme for research, technological development and demonstration under grant agreement No. 607000 and of FCT (Portuguese Foundation for Science and Technology) through the Project UID/MAR/04292/2013 financed by MEC (Portuguese Ministry of Education and Science) and FSE (European Social Fund), under the program POCH (Human Capital Operational Programme).

6. REFERENCES
