The effect of pontoons on flow fields in small harbours

Bachelor thesis Civil Engineering
Leonie Straatsma
Internship at the University of Aberdeen
25th of April - 5th of July 2013

Enschede, 21 August 2013

UNIVERSITY OF TWENTE.

1495
UNIVERSITY OF ABERDEEN

The effect of pontoons on flow fields in small harbours

Thesis report

Datum	21 August 2013	
Version	Final	
Auteur	L. (Leonie) Straatsma	
Contact	l.straatsma@live.nl	
University	University of Twente	
Faculty	Engineering Technology (CTW)	
Program	Civil Engineering	
Organization	University of Aberdeen	
Faculty	School of Engineering	
Supervisors	dr. K.M. (Kathelijne) Wijnberg	University of Twente
	Professor T. (Tomas) O'Donoghue	University of Aberdeen

UNIVERSITY OF TWENTE.

1 4 9 5 UNIVERSITY OF ABERDEEN

Acknowledgements

This thesis is the final project to finish my Bachelors study in Civil Engineering at the University of Twente. During a 10 week internship at the University of Aberdeen, I conducted my research and applied my knowledge in a practical situation outside the walls from the University of Twente.

I wanted to do this final project in another country than the Netherlands and while I was looking for an interesting assignment this project about water flows in harbours at the University of Aberdeen came upon my path. The subject directly appealed to me and this assignment gave a great opportunity to go abroad, improve my English and learn about an entirely different country and culture.

I really enjoyed my time at the University of Aberdeen even though some parts of the project really frustrated me. During the project I have learned a lot and discovered the value of good advice and support. Hereby I would like to thank Professor Tomas O'Donoghue for his valuable advice and remarks through the learning process of my bachelor thesis. I appreciate all the time he took to sit down with me and help me with all the unsolved questions. Furthermore I would like to thank Kathelijne Wijnberg for her support in the Netherlands before I went to Aberdeen as well as her feedback and engagement during my stay in Aberdeen. Last but not least I would like to thank Simon Buhler for all his effort to introduce me to the software and Yuri Mastenbroek for the times he has read my report before the final version was ready.

Leonie Straatsma
21 August 2013, Enschede

Abstract

Siltation is a problem in every harbour, suspended sediments come into the harbour with the flood tide or via flows from surrounding land. Because of the lower velocities in the sheltered basin the suspended sediments settle down. Many harbours at the northeast coast of Scotland are transformed from a fishing harbour into a recreational harbour which involves the placement of pontoons. In these new marinas an increase of siltation is detected which might be caused by the effect of the pontoons on the flow fields within the harbour. The aim of this research is therefore, to investigate these effects in small tidal harbours.

A 3D hydro dynamical model for a schematic harbour was made in FLUENT. This Computational Fluid Dynamical software from ANSYS uses the Navier-Stokes equations to calculate the flow quantities, such as velocities and direction, inside the harbour domain. The schematic harbour domain was 120m long and 80m wide, with a 10 wide inlet located in the corner. A structured grid is used to divide this domain in multiple control volumes for which the equations can be solved. The tidal range used in this research resembles the spring tide in the northeast of Scotland and is 4.5m. The simulated period was 1 hour instead of the real 12hrs, which leads to higher velocities in the results. A velocity inlet was used to prescribe this time varying boundary conditions which are necessary for this tidal model. The Volume of Fluid method is used to solve the equations for two fluids (water and air) and to simulate the free surface.

After the model was set up three different scenarios with pontoons were designed and simulated in the model. Each of the scenarios had different pontoon depths or number of pontoons but all simulations started with the same boundary conditions. Therefore the resulting differences in horizontal flow fields could be related to the differences in pontoon depth or the number of pontoons.

The main conclusion of this research is that the placement of pontoons has an enormous effect on the flow velocities and circulation inside the harbour. The velocities in all scenarios with pontoons were more than twice as low as the scenario without pontoons. Furthermore in the scenario without pontoons one circulation cell was seen inside the harbour during the whole tidal cycle in contrast to the multiple circulation cells which occurred in the scenarios with pontoons during flood tide and the lack of circulation during ebb tide.

The depth of the pontoons influences the circulations inside the harbour as well. In the case of deeper pontoons stronger circulations are formed between the pontoons. Secondly in the cases with the less deep pontoons the flow directions are more scattered and the flow underneath the pontoons is more similar to the scenario without pontoons. The number of pontoons changes the number of circulation cells in accordance to the number of pontoons and does not include other big changes in the flow fields. However the direction of these circulations can be different when the distance between the pontoons increases.

To conclude, the pontoons have multiple effects on the horizontal flow patterns inside a harbour, depending on the depth and number of pontoons. The main effects are the decrease of velocities and an increase in the number of circulation cells inside the harbour. These effects may contribute to the sedimentation in the new marinas such as the Abroath harbour.

Contents

1. Introduction

In this research the water movements in small tidal harbours are investigated. In these small harbours the tides create an in- and outflow of water which results water circulations inside the harbour basin. The shape of these circulation patterns depend on the dimensions of the harbour such as length, width and width of the harbour entrance. Pontoons are nowadays placed in many of the small harbours to facilitate mooring places for small yachts; do these pontoons influence the flow patterns as well? For this research the flow fields in the harbour were simulated using a numerical model known as ANSYS FLUENT.

1.1 Research motivation

In the northeast of Scotland a lot of fishing harbours exist. These harbours are situated along the shoreline of the sea and are influenced by the tide, therefore these are known as tidal harbours. Most of the tidal harbours have an inner basin where the vessels are protected from big waves coming from the North Sea. Nowadays many of these old harbours have lost their fishing origin and are transformed into a marina for smaller yachts and sailing boats. The bigger fishing vessels do not come into the inner basin anymore and stay in the outer basin. To create enough places for the smaller boats to moor, several pontoons are placed in the inner basin (see Figure 1 and 2). In these new marinas sedimentation is a bigger problem than it was when the harbour still had a fishing purpose.

Siltation is a problem in every harbour and the sediments have to be removed to secure a save passage for yachts. Suspended sediments come into the harbour with the flood tide or via flows from the surrounding lands and because of the lower velocities in the sheltered basin the suspended sediments settle down. In the new marinas sediments seem to build up faster than before, when the pontoons were not present. Sedimentation is a bigger problem in these new marinas because the pontoons make it impossible for a dredger to come into the harbour and remove the sediments.

A possible reason for the increased sedimentation in new marinas could be the effect of the pontoons on the flow fields within the harbour. These pontoons may possibly decrease velocities inside de marina, which enhances sedimentation.

Figure 1 - Arbroath harbour before marina (Arbroath boating and sailing club, 2013)

Figure 2 - Arbroath harbour with marina and pontoons (maps.bing.com, 2013)

1.2 Research aim and research questions

As introduced above the effect of pontoons on flow field could affect the sedimentation in a harbour. Therefore the aim of this research to gain insight in the change of flow fields in a tidal harbour due to the transition of a fishing harbour to a recreational harbour and the related placement of pontoons.

The main research question and the sub questions are the following:
How does the placement of pontoons influence the flow field inside the harbour?

1. What type of water movements exist in small tidal harbours and what do they look like according to literature?
2. What flow patterns and velocities are seen in a hydrodynamic model of a simplified harbour?
3. How do the different configurations of the pontoons (number and depth) influence the flow patterns and velocities in a small harbour?

1.3 Outline of study

This section gives the outline of the study, which is a model study that started with a literature study. Meanwhile a few concepts are presented to define the scope of this study. To give a quick overview Figure 4 visualizes the outline of the study.

To start the research, a small literature study is conducted. The study was set up to gain insights in water flows in small tidal harbours around the world and to answer research question 1. For the following parts of this research the focus is tightened to small harbours at northeast coast Scotland (Figure 3). These small marinas are nowadays meant for recreational purposes and therefore many of them have or will get pontoons to arrange mooring places for yachts.

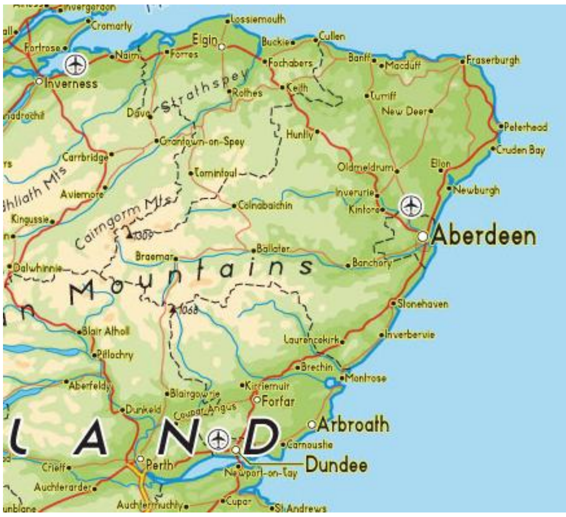

Figure 3 - Northeast of Scotland (Map of UK.co.uk, 2013)

After the literature study, a hydro dynamical model is made to simulate the water flows inside a harbour. FLUENT is used to make this model and can calculate the 3D flow fields within the harbour. Other commercial software could be used for similar purpose, but FLUENT was already available at the University of Aberdeen and provided the possibility of obtaining a very detailed flow model for this application.

A schematic geometry of a small harbour is used to make the harbour model. The dimensions are roughly based on the inner basin of Arbroath harbour (Figure 1 and 2), which is situated on the east coast of Scotland. This harbour was chosen because in this marina a big problem with siltation exists. This basin has a rectangular shape with a length and width of 120m by 80m.

The primary characteristic of harbours on the northeast coast of Scotland is the influence of tides. The tides are semi-diurnal and have an average range of 4.5ms during spring tide and 2m during neap tide. The average flood time is 6hrs and 20mins and the ebb time is 6hrs (Centre for Coastal & Marine Sciences, 2013).

Finally, when the FLUENT model was set up, it was used to simulate the flow fields for several scenarios of pontoons. The scenario without pontoons will give the basic flow fields in a small harbour and are compared to the found water movements in the literature. The effects of the pontoons on the flow fields are visible when the other scenarios are compared with this basic scenario. Analysis of these results will give answer to the second and third research question and conclusions for the entire study could be made.

1.4 Outline of report

This report has the same structure as the conducted research and is structured as follows. The gained insights with the literature study are presented in Chapter 2. After that, basic knowledge about FLUENT and belonging fundamental equations are introduced in section 3.1, and the implementation of FLUENT is described in the following section 3.2. In Chapter 4, the analysis of the results is presented and these are discussed in Chapter 5. This report ends with the conclusions and recommendations.

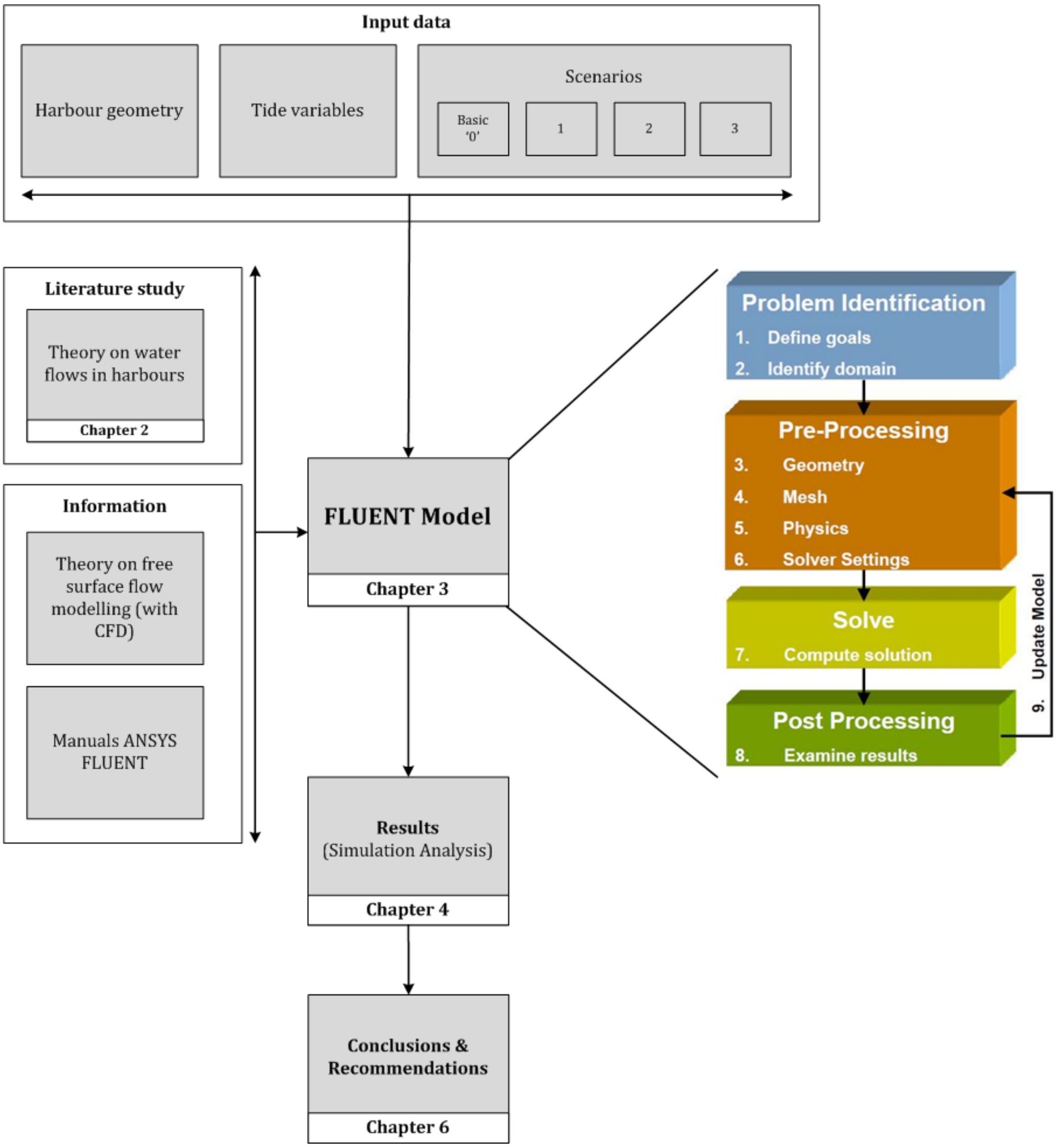

Figure 4 - Research methodology (including CFD modeling overview (ANSYS, Inc. 2009))

2. Background: Flow in harbours

This chapter provides information about water movements in harbours and will simultaneously answer the first research question. In the past, several studies have been executed concerning water movements in harbours. Most of these were aimed at indicating the flushing capacity, the amount of water flushed out and replaced during one tidal cycle. Therefore some of them also show results about water movements inside the harbour basin. The next paragraph will introduce the different water movements in harbours before explaining the influences of several geometrical features on the water movements in the harbour in the rest of this chapter.

2.1 Water movements

In a small tidal harbour several water exchange processes contribute to the water movements inside the basin. Stoschek and Zimmerman (2005) summarized the processes, as illustrated in Figure 5. The flow effect, caused by the river flowing by, initiates circulations inside the harbour. The velocities of this circulation are depending on flow velocities of the river as well as the harbour geometry. The tide or unsteady river flows change the directions of these circulations, which is the tide effect. When a river is discharging in the harbour this freshwater will cause stratification due to the density differences with the salt water. This results in vertical density currents, the density effect. In this research no river is discharging in the harbour, therefore there will be no large scale vertical water movements. Additionally the considered harbour is situated inside another basin and is therefore not affected by the flow effect. The tide effect is the biggest contributor to the water movements in the scope of this study.

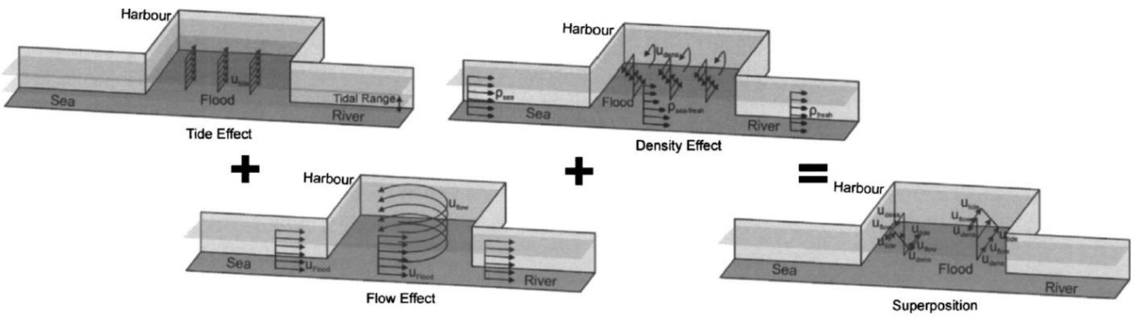

Figure 5 - Water exchange processes in a tidal harbour (Stocheck and Zimmerman, 2005)

In the studies that have been conducted about the tide induced water movements in a harbour, is found that the tide also forces circular water movements in the harbour (e.g. Jiang & Falconer, 1983 or Saalbach, Zorndt, Krämer & Schlurmann, 2012). The velocities and directions of these movements are highly dependent on harbour geometry, such as the harbour entrance and the shape of the basin.

2.2 Influence of entrance geometry

Harbours can have different shapes of entrances: an open entrance (Figure 6a) or a narrow entrance (Figure 6b). Yin, Falconer, Chen & Probert (2000) investigated the influence of an entrance on water movement in a harbour. This experimental research was conducted with scale models (0.7 by 0.7m) and a tidal current with a period of 360s, mean water depth of 173.5 mm and amplitude of 52.5mm. They found that the effect of the tide on the velocities in the harbour is lower with a narrow entrance compared to an open harbour. Figure 6 shows both models at 90s: at this time the patterns show the highest velocities. Both harbours show the same circular pattern, but the velocities in harbour A are about 1.5 times higher than in harbour B.

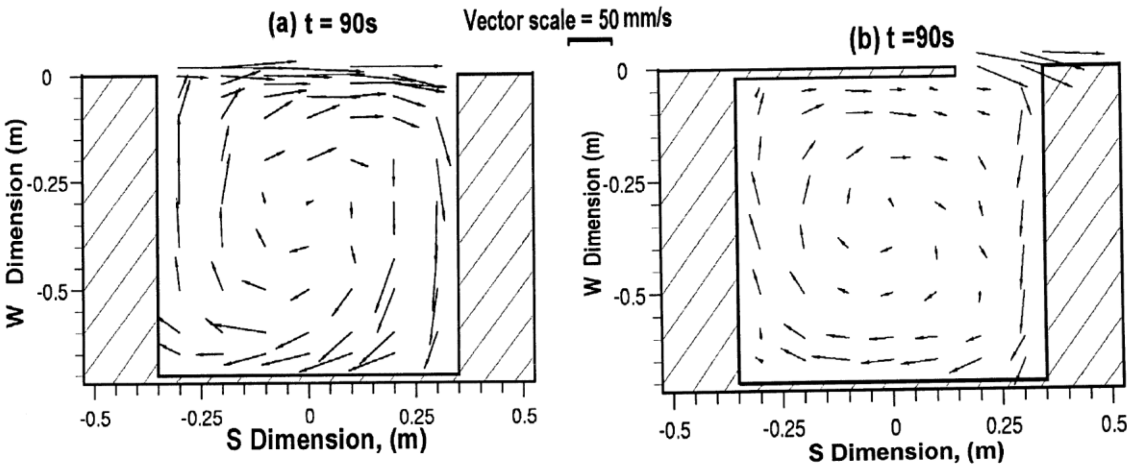

Figure 6 - Tidal flow velocity vectors in a harbour with different entrance geometries (Yin et al. (2000))

2.3 Influence of harbour shape

The geometry of the harbour also influences the water movements in the harbour. Within harbours with a single asymmetric entrance and a simple rectangular shape, the plan-form geometry has a marked influence on the basins' flushing response (Jiang & Falconer, 1983). With experimental studies, they determined the influence and observed pathlines for several harbour configurations, all with the same plan-form area. The formed circular eddies inside the harbour are strongly dependent of the geometry of the harbour, which is also confirmed by Yin et al. (2000). For a square harbour or a harbour with geometry L/B between ½ and 2 the circular eddies will generate the biggest flushing capacity and the highest velocities. When the geometry L/B exceeds these limits a second eddy is formed in the harbour as shown in Figure 7. In the cases with two circulation cells in the harbour, the cell nearest to the entrance will always be in counter clockwise direction, while the second cell rotates in the clockwise direction. Where a clockwise circulation means the water is flowing from north to east and then south and concurrently a west direction.

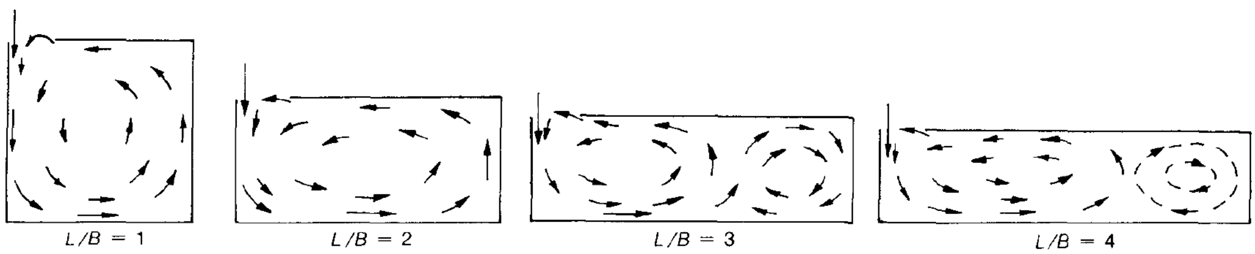

Figure 7 - Observed pathlines at mean water level for four harbour configurations (Jiang & Falconer, 1983)

2.4 Influence of internal structures

Pontoons or other internal structures in the harbour influence the water movements in the harbour as well. Although the flow around floating rectangular structures has been studied in example by Jung, K.H., Chang, K., Huang, E.T. (2003), which shows a vertical flow around the object (Figure 8), the effect of pontoons on the horizontal water movements in a limited space such as a harbour has not been studied yet.

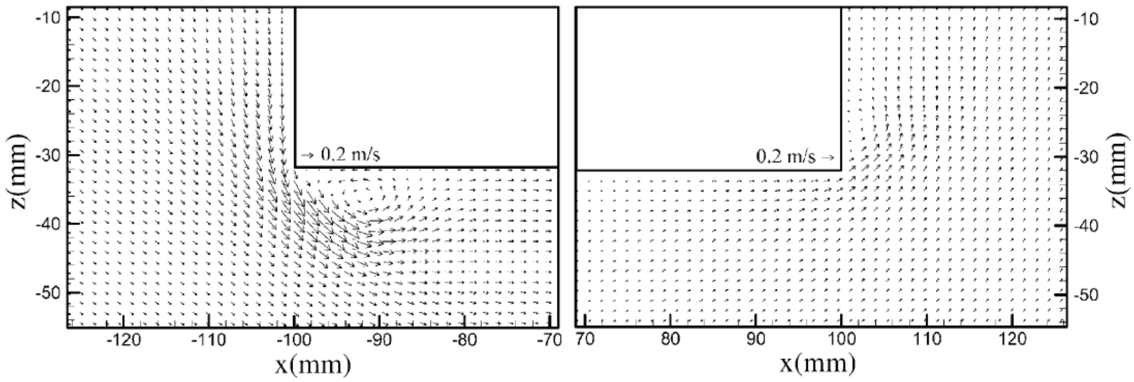

Figure 8 - Mean velocity flow field around a floating rectangular structure (Jung et al, 2003)

Falconer (1980) made a numerical model of a square harbour with an impermeable barrier inserted in one of the models to investigate whether or not the flushing characteristics would be improved. The placement of the barrier did not improve the flushing characteristics, but delivered the flow patterns illustrated in Figure 9. This shows that the single circular cell as expected in a square harbour without a barrier is split in two different cells at each side of the barrier. Whether pontoons have the same effect on the flow patterns is investigated in this research.

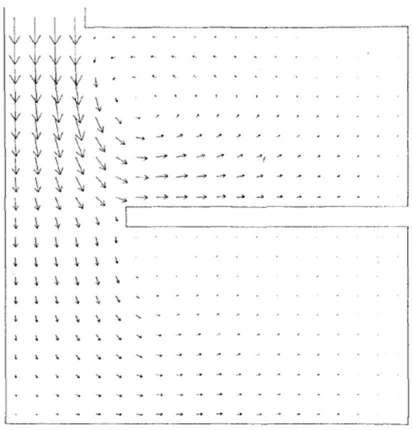

Figure 9 - Tidal circulation in harbour with barrier (Falconer, 1980)

2.5 Chapter closure

All in all, out of the three major effects described by Stochek & Zimerman (2005), the tide effect is the main contributor to the water movements in this study. The tide effect mainly results in horizontal water flows and causes a circular flow field inside the harbour. This flow field is affected by the entrance geometry as well as the harbour geometry. The effect of pontoons on these tidal flow fields is not investigated yet and that is the aim of this research.

3. The harbour model

ANSYS FLEUNT is the program which was used to simulate the flow field in the harbour. The following section will explain the most important equations behind FLUENT. The second section will explain how FLUENT was used in this project.

3.1 Introduction to FLUENT

In this chapter a short introduction to FLUENT is given with the most important equations and physical models which are used in this research. FLUENT is the software package of ANSYS which provides modelling capabilities for a wide range of flow problems. The program is based on Computational Fluid Dynamics (CFD) methods. This branch of fluid dynamics can simulate flows by the numerical solution of governing equations. The fundamental equations of fluid motion that form the basis of these methods have been known for 150 years. Calculating solutions for complex geometries however cannot be done analytically. Computational Fluid Dynamics can provide approximate computer-based solutions by replacing the fundamental equations with systems of algebraic equations (Sayma, 2009).
The role of these computer-based solutions has become so important that today CFD can be viewed as a new third dimension in research about fluid dynamics, the other two dimensions being pure experiment and pure theory (Figure 10) (Wendt, 2009).

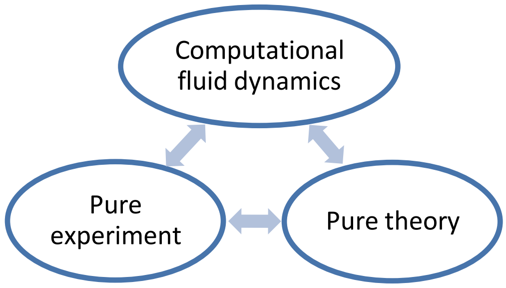

Figure 10 - Relationship between pure experiments, CFD and pure theory (Wendt, 2009)

The governing equations which FLUENT solves are the conservation equations for mass and momentum. These are the continuity equation (3.1) and the Navier-Stokes equations for momentum (3.2-4). The equations used in this case are the following equations, which are simplified for constant density and temperature:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{3.1}$$

$$\underbrace{\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)}_{Inertial\ terms}=-\underbrace{\frac{\partial p}{\partial x}}_{Pressure\ gradient}+\underbrace{\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)}_{Viscous\ terms} \tag{3.2}$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \tag{3.3}$$

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{3.4}$$

where u, v, w, are velocity components in 3 orthogonal directions x, y, z; µ is viscosity; ρ is fluid density and p is pressure.

FLUENT uses a discrete domain to calculate a solution for these equations and find the values of flow quantities at a large number of discrete points spread around the geometry. These points are connected together in a grid which fills the whole domain.

This grid (also called mesh) can exist of cells with different sizes and shapes. The main two categories of grid are a structured and the unstructured grid which are defined by the shape of the cells. A structured grid consists only of hexameter shaped cells (like cubes) in the whole domain (Figure 11a). An unstructured grid can consist of a lot of different shapes of cells such as pyramid, tetrahedron or prisms (Figure 11b). The unstructured grid is especially helpful when a domain consists of a lot of different parts or contains small corners or round edges. For each of these cells FLUENT calculates the flow quantities, therefore more cells inside the domain means a longer calculation time but more detailed results of the flow.

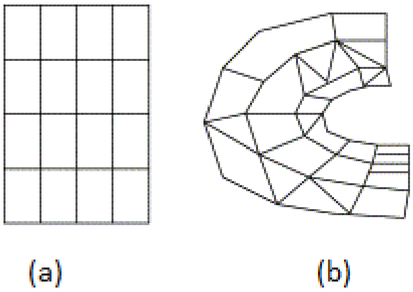

Figure 11 – (a) Structured grid (b) Unstructured grid

After that different boundary conditions are assigned to bound the flow inside the domain and give the initial values for the calculations. These boundary conditions can be the friction coefficient of the walls or the flow conditions (such as velocity and direction) at the entrance and outlet of the domain.

The Volume of Fluid (VOF) solver is used to enable FLUENT to calculate the flow quantities for two fluids at the same time. This is necessary to simulate the free surface level which in this project rises and falls according to the tidal cycle. The VOF method makes it possible to locate the fluid-fluid or fluid-gas interface, in this case the free surface between water (liquid) and air (gas). The Volume Fraction (VF) is introduced as a new variable and resembles the volume fraction of a phase (in this case air or water) inside a control volume. Each grid cell is one control volume and the sum of all volume fractions of all phases in each cell is one. In this case the following three conditions are possible, with VF_a as the volume fraction of air:

1. $VF_a = 0$, the cell is filled with air
2. $VF_a = 1$, the cell contains no air (so is full of water)
3. $0 < VF_a < 1$, the cell contains both water and air.

The third condition is called a mixture in FLUENT and these cells contain an interface between air and water. According to the Volume Fraction, appropriate values are assigned to each control volume. These values are represented by the volume average values for each cell. In this way the governing equations can be solved for each cell in the domain.

To visualize the free surface after the calculation, the interface in the cells containing a mixture has to be reconstructed with the volume fractions in each cell. FLUENT uses the geometric reconstruction scheme (Figure 12) (Theory Guide), where information about inflow from the neighbouring cell and the volume fraction of the cell itself is used to generate a linear slope inside each cell. This piecewise-linear approach is shown in Figure 12d.

In summary, FLUENT is able to simulate the flow patterns and calculates quantities such as velocity

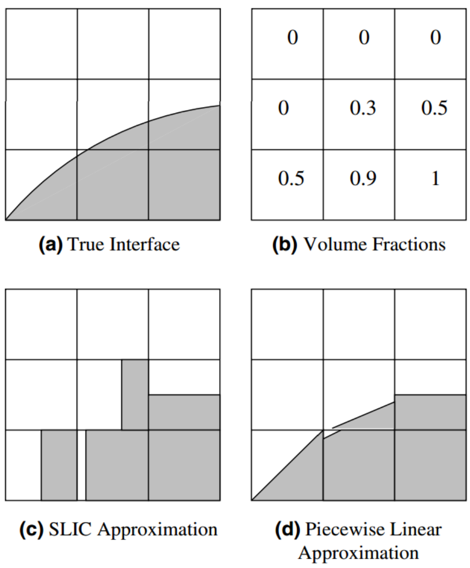

Figure 12 - Interface (a) and associated volume fraction (b) Reconstruction of the interface done with the SLIC method (c) or the piecewise linear approximation (d) (Theory Guide)

and directions for this project. To do so a domain is made which was the basis for the geometry and the mesh. Boundary conditions were then set up to bound the flow inside the domain and give initial values for the calculations of the governing equations. The VOF method enables FLUENT to calculate the flow quantities for both water and air in this project, which is necessary in this project to simulate the free surface inside the harbour.

3.2 Application of FLUENT to the present problem

To explore the effect of pontoons in a harbour basin, the model in FLUENT was first set up for a basin without pontoons (Scenario 0). When this model worked, the other scenarios with pontoons were simulated. In each of these scenarios the number or the depth of the pontoons were altered. In this way the effects of the pontoons on the water flows could be examined. In this section the input data, boundary conditions, settings and assumptions for the FLUENT model are described according to the nine steps to set up a FLUENT model (Figure 4). The time it took to run the calculations in FLEUNT produced the biggest challenge for the short period of this project. To make sure it was still possible to simulate several scenarios within the given time, some choices are made to prevent the calculation time increasing too much. Unfortunately, some of the choices therefore do not resemble the reality as much as possible.

3.2.1 Goal

The aim of this model is to simulate the flow fields in a harbour. The model was set up for a schematic harbour which resembles the marina of Arbroath. In the model the water level has to rise and fall according to the present tidal cycle in the northeast of Scotland and the results of the simulations have to show velocities and direction vectors inside the harbour.

3.2.2 Domain

The domain of this project is illustrated in Figure 13. It is 120m long by 80m wide, with a 10m wide inlet located in the corner. The domain could have included the several outer basins of the Arbroath harbour, then the inflow conditions at the inlet of the marina would be more developed. On the other hand, this bigger domain should contain much more grid cells which would increase the calculation time of the simulation, which is not possible in the given time for this project. To simulate a more developed flow entering the harbour, the inlet is extended with 40m to give the inflowing water space to develop a non-uniform flow profile.

The domain is bound by the vertical harbour walls and the bottom (grey and blue in Figure 13). The upper boundary simulates the connection of the domain with the open air. To ensure that only air and no water can leave the domain through this boundary layer, this boundary is situated at a level which is above the water surface at all times. The initial water depth in the harbour is 1m (low tide) and the simulated tidal range is 4.5m (similar to northeast Scotland), therefore upper boundary layer is at a height of 6.5m with a safety margin of 1m.

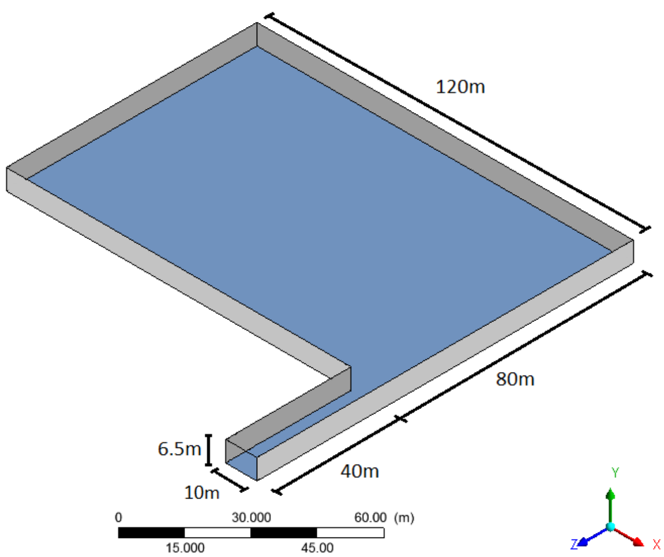

Figure 13 - Domain of the simulated harbour

3.2.3 Geometry and mesh

The geometry is based on the harbour domain and the mesh divides this domain in smaller control volumes for the calculations of FLUENT. The geometry and mesh of the harbour domain could not be made in FLUENT, therefore these were made in a geometry builder and a meshing application which are also provided by ANSYS. This way the geometry and the mesh could be imported to FLUENT.

A structured mesh is used to divide this geometry in many control volumes as shown in Figure 14. This structured mesh provides advantages in this case to an unstructured mesh because the number of cells needed, is lower and the calculation time is reduced. In most cases it is not possible to use a structured mesh since a complex geometry is used. In this case it is possible because the geometry is not complicated since no round edges, sharp corners or small objects are present in the domain. Furthermore a structured mesh is necessary in a shallow domain, where the length and width exceed the height multiple times (Stamou, Katsiris, Moutzouris & Tsoukala, 2004).

The grid is refined in parts of the domain with bigger changes in velocities and directions. The refinement is done near the boundaries and especially around the entrance and pontoons so FLUENT can calculate a more detailed flow. This refinement is best visible at the inlet boundary in Figure 15. The initial grid was set at 25000 cells and then progressively refined until further grid refinement resulted in little difference in results. The final grid had approximately 300.000 grid cells. The simulated results are not completely independent of the mesh size, but an even finer grid resulted in a much longer calculation time. An optimum between the needed detail and the calculation time was reached at this point.

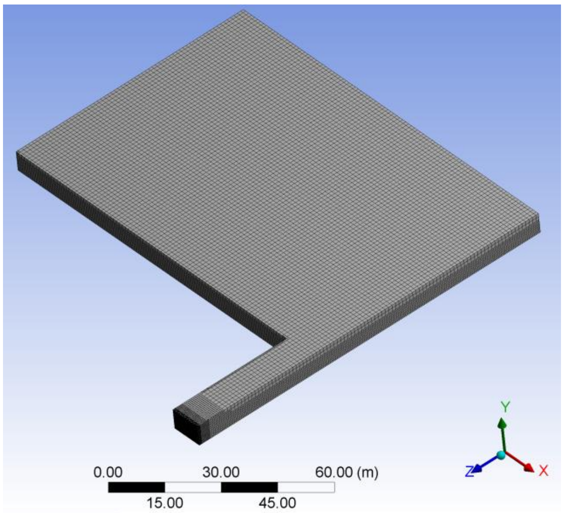

Figure 14 - Structured mesh of the harbour

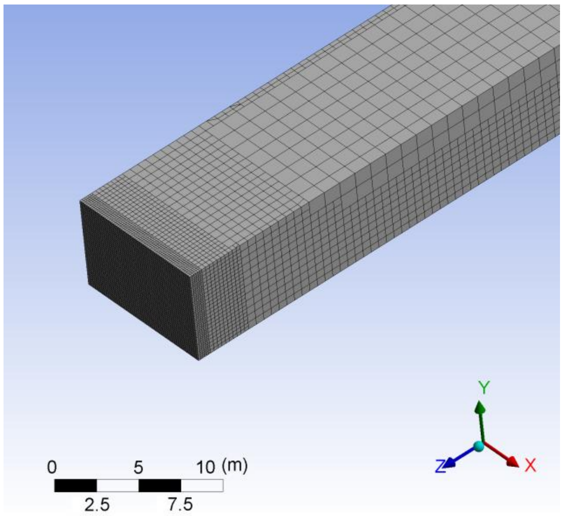

Figure 15 - Refinement of mesh near the inlet boundary

3.2.4 Physics

After setting up the grid, the general physics and the physical models had to be determined. To start the general physics were set in FLUENT, gravity as -9.81 m/s^2 and operating pressure as 1 atm. The operating pressure is initialised at a point (0; 6.5; 0) which always contains the density of the lightest fluid of the domain. If this had not been done an incorrect distribution of the hydrostatic pressure would occur. Simultaneously the transient solver is activated instead of the steady solver because this case is time dependent due to the time varying velocities and the water level at the inlet of the harbour.

Of the physical models available in FLUENT, the laminar flow was selected rather than a turbulence model. A turbulent model would be more realistic, but the calculation time expanded rapidly when a turbulent model was chosen. Additionally a turbulent model has to be selected carefully; not all turbulence models are appropriate in each case (Theory Guide). In this project time was too limited to sort out which turbulence model represented the most realistic flow in the harbour.

The Volume of Fluid multiphase model was used to simulate the interface between two different fluids in this project. This model is designed for two or more fluids where the interface between the fluids is of interest (Theory guide). For this project, the VOF model is best of the multiphase models if compared to the Eulerian model and the Mixture model which are focussed on simulating mixtures of multiple fluids or fluid and particulates. Air and water are selected as the two phases, where air is the primary and water is the secondary phase (as prescribed in ANSYS FLUENT User's Guide, 2013a).

3.2.5 Boundary conditions

The boundary conditions are determined for three different kinds of boundaries; the walls, the outlet and the inlet (Figure 16). The boundary conditions are designed to resemble the reality as good as possible.

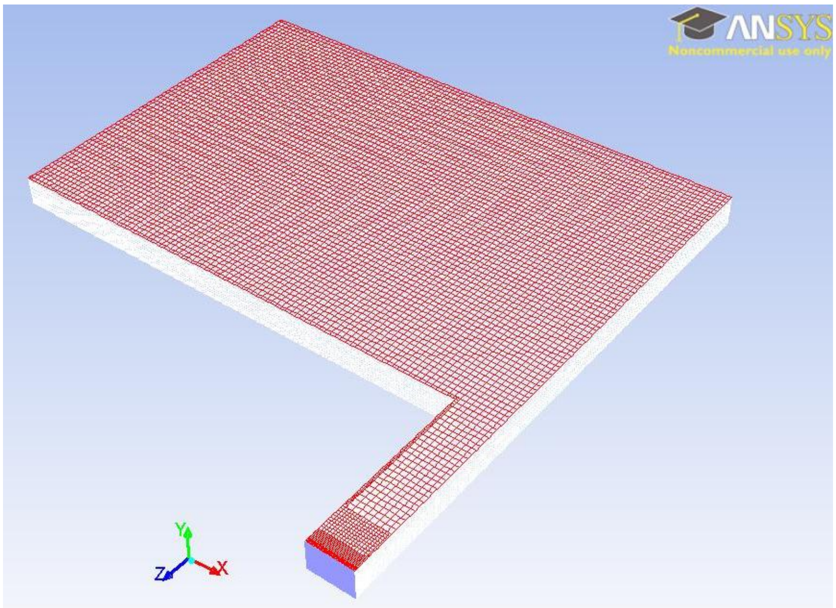

Figure 16 - Boundary conditions: walls (white), outlet (red) and inlet (blue).

Walls

The 'wall function' is used for the wall boundaries (white). This function resembles the friction of the walls. It bounds fluids and solid regions, enforces a zero velocity of all velocity components in the fluid at the boundary and relates the shear stress at the wall to the cell velocity component parallel to the wall.

Pressure Outlet

The red boundary is the connection of the domain with the open air. At this boundary air should be able to move in and out the domain without any obstructions. During flood tide the water level rises and air should leave the domain, while during ebb tide, the air should (re-)enter the domain due to the falling water level. The Pressure Outlet is used to resemble this boundary, because it allows the air to move in and out of the domain even though it is called an outlet. To ensure that the boundary simulates the same pressure profile as the normal atmospheric pressure (1 atm), the Gauge pressure is set as zero.

Velocity Inlet

The inlet (blue boundary) is determined as a Velocity Inlet so that the water in- and outflow caused by the tides can prescribed during the simulation. The Velocity Inlet makes it possible to change the velocity and phase properties at each cell of this boundary during each time step of the simulated tidal cycle, this is necessary because of the rising and falling water level and the time varying velocities at the inlet.

Input formulas

To determine the velocities at the inlet during the whole tidal cycle, a formula for the water level during spring tide was set up according to gained tide tables from the Arbroath harbour. The simulated time was initially intended to be a full tidal cycle of an average of 12 hrs and 24 mins, the average time of one tidal cycle in the northeast of Scotland. Concerning the long time it took to calculate such a period, the simulated period is brought back to a 60 minute cycle with the same tidal range of 4.5 meters. Consequently the velocities at the inlet are much higher than realistic because the same amount of water needs to enter the domain in a 12 times shorter period. The difference between the original and the used velocities is shown in Appendix B. However if the tidal range was adapted to simulate more realistic velocities the range had to be 18cm. The initial water depth of 1m would then be much higher compared to the tidal range which might implicate that only the upper water layers will show any signs of circulation during the tidal cycle. This would not give a good representation of the flow patterns in the whole harbour. Therefore the original tidal range is used as input for the inlet boundary even though the velocities are higher.

The data for the water level during spring tide was collected at the harbour master of Arbroath marina. This data is schematized in the formula 3.1 for the water level and represented by the green graph in Figure 17. The variation of the water level is formulated by the following formula:

$$h(\mathrm{t}) = -2.25m * \cos(\omega t) + 3.25m \qquad (3.1)$$

where t is in seconds and $\omega = \frac{2\pi}{3600}$

The mass flow and velocity at the inlet to reach the water levels have been derived from the formula 3.1 as follows:

$$Q(t) = \frac{\partial h}{\partial t} * A_{harbour} \qquad (3.2)$$

$$V(t) = \frac{Q(t)}{A_{entrance}(t)} \qquad (3.3)$$

where $A_{harbour}$ is the area of the harbour and $A_{entrance}(t) = \frac{width\ of\ inlet}{h(t)}$

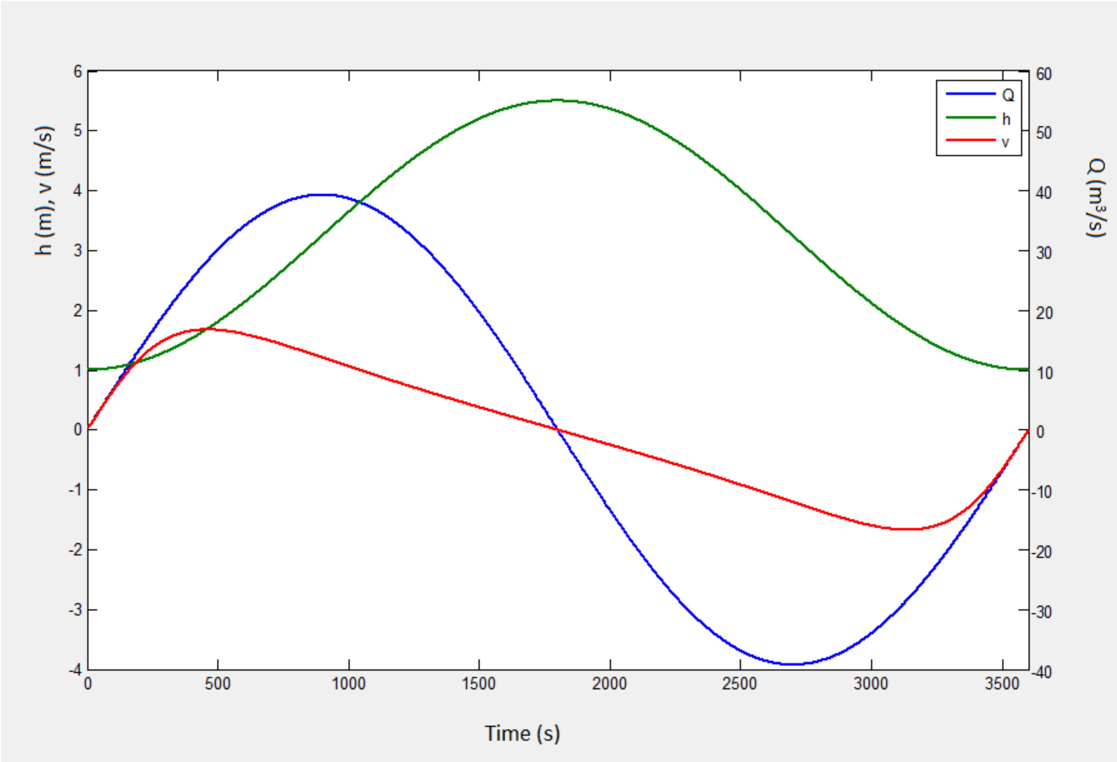

Figure 17 - Variations of the water level and corresponding velocity and mass flow under tidal conditions with a period of 3600s, mean water depth of 3.25m and amplitude of the water level's variation 2.25m.

User Defined Function

The inlet boundary has these time varying boundary conditions (velocity and water level), which have to be determined for each time step according to formulas 3.1 and 3.3. To prescribe the time varying boundary conditions according to formulas 3.1 and 3.3 a User Defined Function (UDF) is needed. This UDF is written in C++ (Appendix A) which enables FLUENT to determine the properties for each cell at the inlet boundary; whether it contains water or air and which velocity it has at that specific time step. Just like the whole domain this inlet boundary is divided in many different cells. The properties of each cell have to be changed at each time step of the simulation to simulate the tidal cycle.

At the first time step, the initial water level at the inlet boundary is 1m and all the cells above this level should contain air and all cells below should contain water to simulate this free surface. In the following time steps the water level rises and more cells of the inlet boundary contain water. Simultaneously velocity has to be assigned to these cells filled with water to simulate water flowing into the domain. Meanwhile fewer cells are filled with air and no velocity should be assigned to these cells because no air is simulated to flow into the domain. This process is done by the UDF during the calculations of FLUENT. Figure 18 shows a flow diagram which resembles the effect of the UDF for each cell at the inlet boundary for a certain time step.

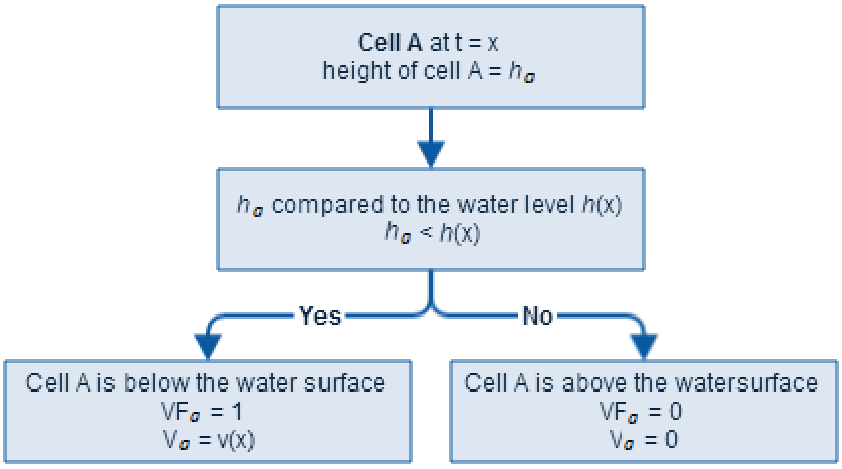

Figure 18 - Calculating the properties of Cell A, a cell inlet boundary at t = x and height h_a

This UDF implies that the velocity at the inlet is initialized as a uniform flow, because all cells at which contain water at a certain t have the same velocity. Therefore the entrance is stretched (as mentioned before) to allow the flow to create a more non-uniform flow profile before entering the basin.

Even though the UDF sets the correct boundary conditions and these conditions are in correspondence with the inflow during the simulation, the water level inside the harbour basin does not rise according to the inlet conditions. The maximum water level in the harbour is 3.9m instead of 5.5m. To correct this limitation several adoptions are made to the model but it took too much calculation time the correct solution for this problem. This limitation is therefore still present in the final results (Chapter 4).

3.2.6 Solver settings

The solvers settings were chosen to prescribe the numerical solution method that FLUENT uses to solve the governing equations. The following settings were used because these were recommended by the User's Guide for incompressible, two phase flows, with a structured mesh and circulations of the fluids within the domain. Also these options did not increase the calculation time too much.

Implicit pressure-based solver containing:

- Pressure-Velocity Coupling scheme: PISO
- Interpolation Scheme for Pressure: PRESTO!
- Interpolation Scheme for Momentum: First order Upwind

After the solver settings the solution variables were initialized. The initialization in this case consisted of allocating the initial water level of 1m in the domain at t=0s. First all the cells in the whole domain were assigned a volume fraction of 0, so that all the cells contain air. Secondly a volume fraction of 1 was patched in all the cells situated below 1m, so that all these cells contain water.

3.2.7 Solve

At this point all the settings for the model were made and the solution could be calculated. To calculate the solutions for this model the time step size was 0.4s and the number of time steps was 9000, which sums up to a period of 3600s. To monitor if the solution convergences the convergence behaviour was monitored in a residual plot during the simulation. The time step of 0.4s was selected so that the residuals reduced around three orders of magnitude within one time step. The time step was kept as large as possible to prevent the calculation time from increasing. The number of iterations per time step was kept at the default 20, because it was better for the calculation time and convergence to reduce the time step size than to do to many iterations per time step (ANSYS, 2009). The total calculation time of each simulation was around 24 hours.

3.2.8 Post processing

The analysis of the results was done using the Post-Processing application of ANSYS Workbench. This made it possible to visualize many kinds of results of the simulation, such as velocity, water surface and flow patterns.

During the assembly of the model the Post-Processing application was used to validate the model. The results of the simulations have been compared to the flow patterns as described in the literature. The model was updated several times to make sure that the results were mesh-independent and the simulations provided good results which were in accordance with the literature. After the validation, the application was used to compare the different scenarios. The Post Processing application makes it possible to compare different scenarios in one image.

3.2.9 Scenarios

Three different scenarios were designed and simulated to discover the impact of the pontoons on the flow patterns. These scenarios have been simulated after the model was validated for the basic scenario without pontoons. To be able to compare each scenario with the others scenarios, all the scenarios have the same boundary conditions as the basic scenario which is described above.

The scenarios are designed to see the effect of the number of pontoons and depth of the pontoons on the flow fields in the harbour. The measurements of the pontoons have been derived from the pontoons which are already situated in the marina of Arbroath. These are 40m long and 3m width. Though in the Arborath marina the pontoons float on the water surface and have a depth of approximately 50cm, this was not possible to simulate in this project. Therefore the pontoons have a fixed position and the depth of the pontoons is changed in the scenarios. The number of pontoons is the second variable, the last scenario has only 2 pontoons instead of the 3 pontoons in the first two scenarios. The scenarios' plan form area and the pontoon dimensions are shown visualized in Figure19.

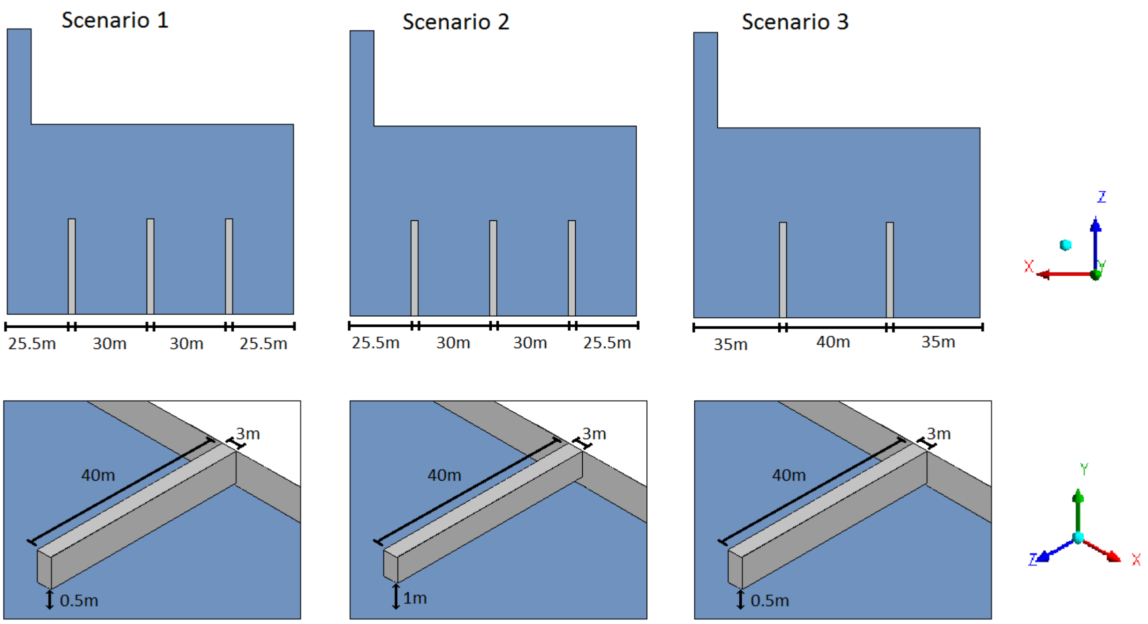

Figure 19 - Above: plan form area for each scenario. Below: pontoon dimension for each scenario

To simulate the pontoons in the geometry the shape of the pontoons is cut out of the domain, resulting in gaps in the domain as shown in Figure 20. In this figure is also visible that the mesh around the pontoon is refined to make sure the velocities and changes around the pontoons can be simulated in detail and the solution is mesh independent.

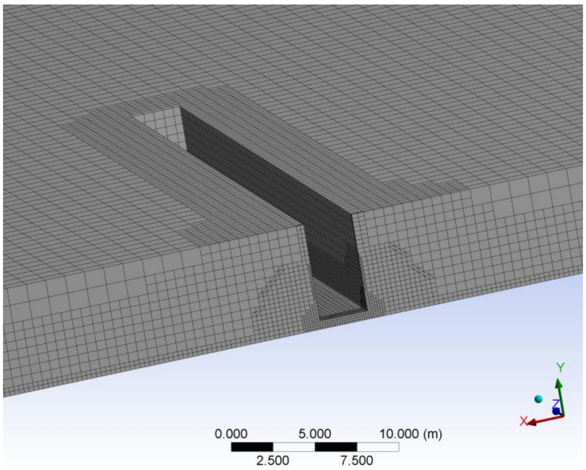

Figure 20 - Refined mesh near the pontoons

3.3 Chapter closure

A lot of settings and boundary conditions are necessary to simulate the harbour in this project. The domain of the model is made to resemble the geometry of a schematic harbour. The structured mesh enables FLUENT to divide the domain in many different cells for which the governing equations can be solved. The physics and boundary conditions are set to resemble the reality as good as possible in the limited time of this project. Three different scenarios are designed to see the effect of the number of pontoons and depth of pontoons in this harbour. To make sure that the simulations for each scenario can be compared to the other ones all boundary conditions and settings are exactly the same. The input for the inlet boundary is shown in Figure 21. The time steps within Figure 21 resemble the time steps for which the horizontal velocity flow fields are presented in Chapter 4. The time steps 900s, 1800s, 2700s and 3600s resemble respectively mid tide, high tide, mid tide and low tide. Time steps 460s and 3140s are the moments with the highest and lowest velocities.

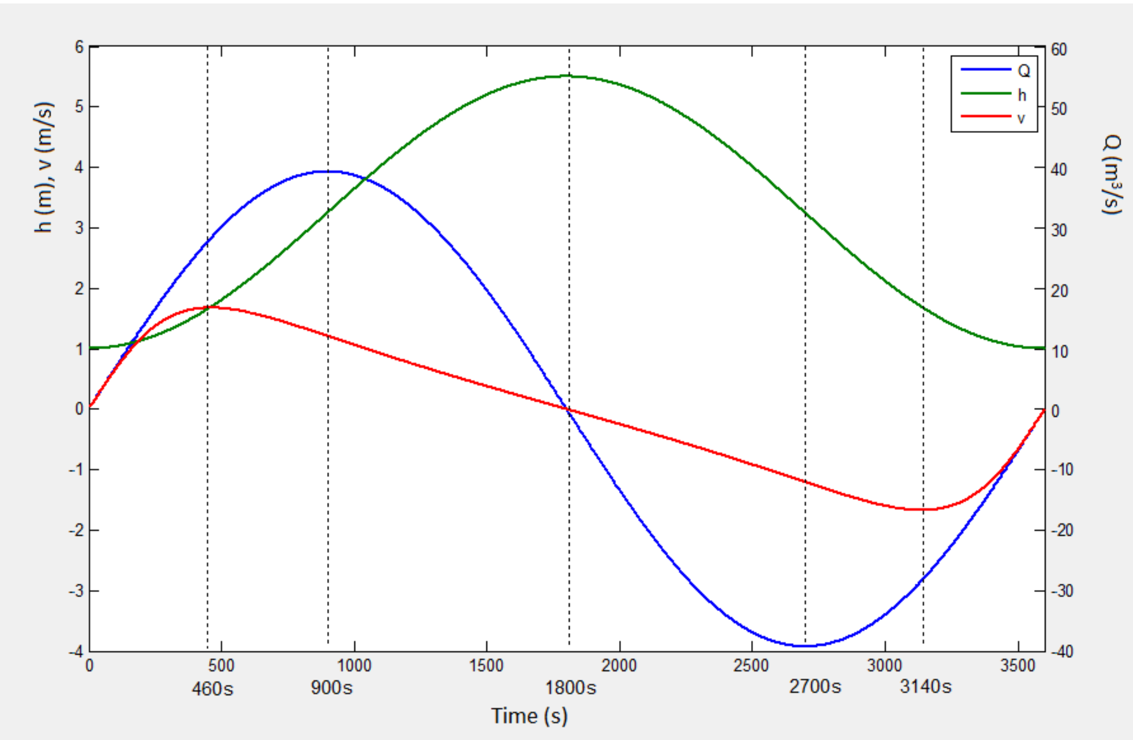

Figure 21 - Variations of the water level and corresponding velocity and mass flow under tidal conditions with a period of 3600s, mean water depth of 3.25m and amplitude of the water level's variation 2.25m. Including indications of time for which the resulting horizontal velocity flow fields are presented in chapter 4.

4. Results

The simulations of the four different scenarios produce an enormous amount of data for the complete 3D flow field inside the harbour. In such a short report it is a challenge to show the most important results without excluding significant data. As indicated in Chapter 2, the horizontal water movements are the biggest contributors to the flow fields in a small tidal harbour without a river discharging in the harbour. Concerning the time for this research the vertical water moments and flow patterns are therefore not further investigated or presented in this research. This does not mean there are no vertical water movements and that these movements do not affect the flow patterns or the sedimentation in the harbour. An indication of the presence of vertical water movements is given in section 2.4 (Figure 8) and in Appendix C.

The results presented in this chapter are the horizontal velocity flow fields at mid depth for 6 time steps for each of the four scenarios (Figures 23 till 26). These flow fields are selected to analyse the effect of the pontoons in the harbour. The time steps and the corresponding inlet parameters; velocity, mass flow and water level are shown in Figure 21. The vectors shown in the flow fields are composed by the horizontal velocity components (u and w). A small indicator (Figure 22) is included to indicate the height of mid water depth for each horizontal flow field. The indicator shows the corresponding water level, depth of the pontoons and the height of the flow field plane for each time step.

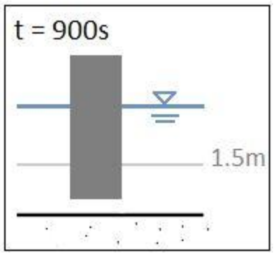

Figure 22 - Depth indicator of horizontal flow field.

The flow fields are presented for a mid depth because these flow fields show the most important features of the horizontal flow in the harbour. Using a fixed height to study the flow fields would cause a distorted picture of the flow because of the big tidal range. The fixed height should be 0.9m or lower to secure it is always below the water surface. In the beginning and end of the tidal cycle the flow field would be situates in the top layer of the flow and at mid tide the flow field at this fixed level would not even be situated near the mid depth of the flow, which shows a distorted picture of the velocities (as shown in Appendix C). Therefore the flow fields at a fixed height for one tidal cycle could not be compared with each other. Secondly these mid depth flow fields are also used in other studies to show the horizontal water movements and can therefore be used to compare the results of different studies.

First the results of Scenario 0 are described since this is the basic scenario. After that the other scenarios are described and compared to scenario 0 in the following sections. Scenario 2 and 3 are also compared to Scenario 1 to indicate differences caused by the depth of the pontoons (Scenario 1 and 2) and the number of pontoons (Scenario 1 and 3).

4.1 Scenario 0

Figure 23 presents the results of scenario 0 at mid depth for the 6 different time steps. These are horizontal velocity flow fields for Scenario 0, which is the scenario without pontoons. The following main observations are made for Scenario 0, first the overall observations and then the main observations per time step:

- A circulation of the water is visible during the whole tidal cycle. The water flows in a counter clockwise directions starting at the inlet and flowing along the west wall to the south, east and north wall.
- The velocities during flood tide are the highest of the tidal cycle, especially until mid tide which is in accordance with the inlet velocities.
- The velocities during ebb tide are low even though the velocities at the inlet reach the same level as during the flood tide. Secondly the velocities are more similar across the circulations in the harbour basin in comparison to the flood tide.
- During the first two time steps the circulation is still developing. The circulation is not fully developed around a point in the middle of the basin and the velocities are not universal in this circulation.
- In the flow field of t=900s a small second circulation is formed in the southeast corner.
- At 1800s the circular pattern is fully developed but the velocities in the harbour are still high (around 1m/s) while the inlet velocities are zero at that moment. The high inlet velocities in the beginning of the tidal cycle caused a big circulating momentum in the water body inside the basin.

The results of this scenario form a base to compare the other three scenarios with. The strong circular movement within this harbour is consistent with the observed pathlines which are found by Jiang & Falconer (1983).

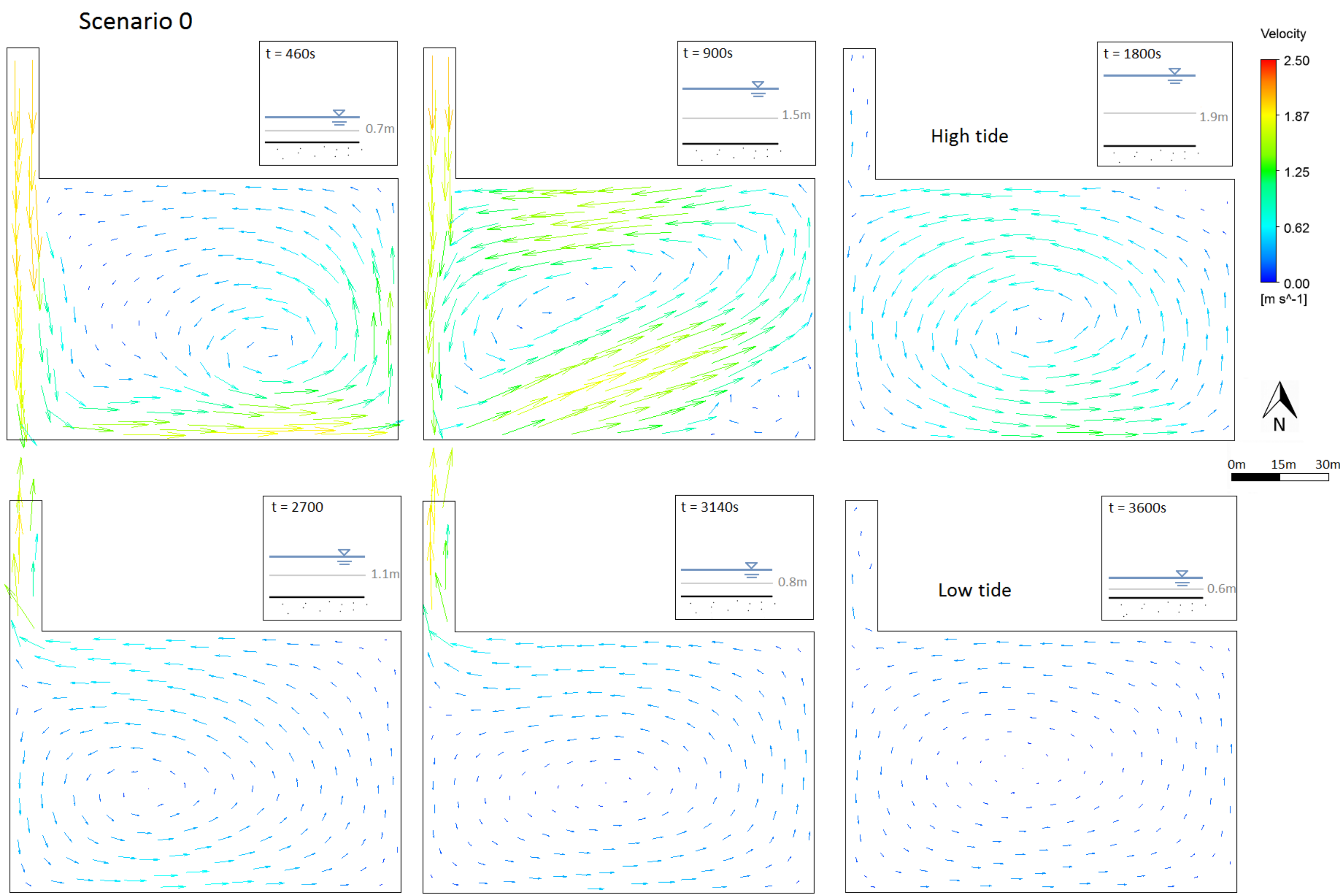

Figure 23 - Horizontal velocity flow fields of Scenario 0 at mid depth on 6 different time steps during a tidal cycle of 1 hour.

4.2 Scenario 1

Figure 24 shows the results of the first scenario with pontoons in the basin. The three pontoons are situated at the south wall of the harbour and are 6m deep. As a result the water depth underneath the pontoons is 0.5m. Scenario 1 has exact the same input as scenario 0, the six flow fields are made at the same time steps and at the same mid depth water level as scenario 0. Therefore Figure 24 can be compared with the results of scenario 0. The main observations for Scenario 1 are presented first.

4.2.1 Observations

- The main observation in this scenario is development of four separate circulation cells between the pontoons and between pontoons A, B and the walls. Each of this circulation cells has another direction, alternately clockwise and counter clockwise, starting counter clockwise at the most west circulation. These alternately circulations arises because the incoming flow (southwards), bounded between the wall and pontoon A, starts a circulation (counter clockwise). This influences the flow direction underneath pontoon A and creates an incoming flow between pontoon A and B in northeast direction. Again the flow is bounded by pontoons and the flow starts circulating, this time clockwise because of the reversed inflow direction. This process influences also the flow direction of the other circulations between pontoons B and C and pontoon C and the wall.
- The velocities during flood tide are the highest in the most west circulation cell, the further east the velocities decrease to almost zero.
- During ebb tide the flow fields do not show any sign of circulation any more. The water flows in almost straight lines to the exit of the harbour. At t = 3600 the velocities inside the basin are almost zero and all smaller than 0.1m/s.
- At the first time step (460s) a scattered flow field is developing in the basin. Even though the flow field is at 0.7m which is higher than the flow underneath the pontoons, the vectors in the flow field seem to go through the pontoons. This suggests that the water flow under de pontoons (below 0.5m) affects the water at the mid depth of 0.7m to flow in the same direction as the flow underneath the pontoons.
- At t = 900s however the mid depth is situated at 1.5m, the distance between the flow underneath the pontoon and mid depth is larger and the pontoons form a barrier for the flow through the basin. The four circulations between the pontoons are developed at this point.

4.2.2 Comparison with Scenario 0

Scenario 1 shows a few significant changes, when compared to Scenario 0 (without pontoons).

- Scenario 1 does not show a single circulation cell inside the harbour, between the pontoons and the harbour walls four smaller circulations are developed.
- Nevertheless the inlet velocities are exactly the same as Scenario 0, the velocity at the various time steps of Scenario 1 are lower. Where the velocity of the circulation in scenario 0 reaches velocities of 1.8 m/s, the velocities of the circulations in Scenario 1 does not exceed the 1 m/s. Finally at 3600s in Scenario 1, the velocities in the water are almost zero compared to the 0.3m/s in Scenario 0.
- The water in Scenario 0 continued to make a circular movement during the ebb tide. In contrast to the flow patterns in Scenario 1, these show the water flowing out of the basin without any circulation.
- A lot of directions in the flow field of Scenario 1 are different compared to Scenario 0 because of the increase of circulations. The biggest difference is the flow directions at the north wall. At t=900 in Scenario 0 the flow near the north wall is directed west, at the same time in Scenario 1 the flow is directed in the opposite direction, flowing east.

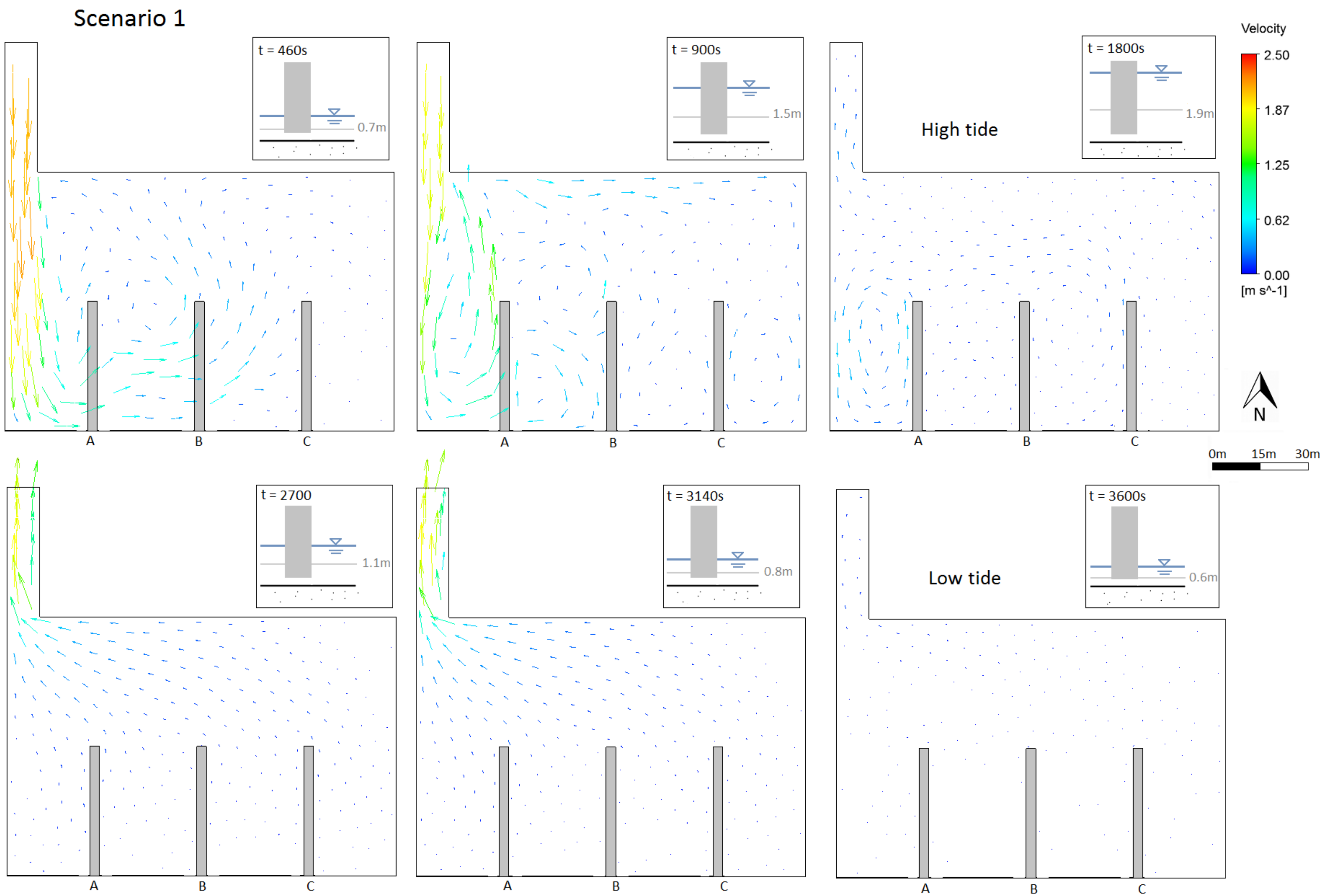

Figure 24 -Horizontal velocity flow fields of Scenario 1 at mid depth on 6 different time steps during a tidal cycle of 1 hour.

4.3 Scenario 2

The resulting horizontal velocity flow fields of Scenario 2 are presented in Figure 25. This scenario has exactly the same plan form layout as Scenario 1; the only difference is the depth of the pontoons. Due to the less deep pontoons the water depth beneath the pontoons is 1m, twice as much as in Scenario 1. All the other conditions are the same as in the previous scenarios and the resulting flow fields are made at the same time steps and water levels. Consequently the results of this scenario can be compared to the previous ones. First the main observations of Scenario 2 are described and these are compared to Scenario 0 and 1.

4.3.1 Observations

- During the tidal cycle no obvious development of circulation cells is visible. Only at 460s a circulation is visible and at 900s a smaller circulation cell seems to develop between pontoon A and the west wall, but neither of this circulation continues during the rest of the tidal cycle.
- The flood tide produces high velocities inside the domain, but at different places. At 460s these velocities are the highest in the southwest corner and at 900s these velocities are the highest near the north wall.

4.3.2 Comparison with Scenario 0

The most important changes and similarities of Scenario 0 and 2 are the following:

- The flow pattern at t=460 in Scenario 1 looks similar to the circulation at t=460 of Scenario 0, both of the circulations are centred on a point in the southeast corner. The velocities in Scenario 2, however, are lower than the ones of Scenario 0. Even though the flow field in Scenario 2 is situated below the pontoons, the velocities do not reach the same level as the velocities in the scenario without pontoons. As introduced in Section 2.4 vertical flows around pontoons affect the flow underneath the pontoons. The effect is visible here because the flow velocity between the pontoons is lower and vertical flows affect the flow velocity underneath the pontoons to decrease.
- In contrast to flow patterns at 460s, all the other flow patterns do not show any similarities at all. Just as Scenario 1 during the ebb tide this scenario does not show any circulation during ebb tide.
- The velocities during the whole tidal cycle for Scenario 2 are much lower than the velocities in Scenario 0.

4.3.3 Comparison with Scenario 1

The geometry of Scenario 1 and Scenario 2 look alike, only the depth of the pontoons is different. These are most important differences between these two scenarios:

- The flow fields during flood tide do not look the same. Especially at t=460s the contrast is big. In contrast to the scattered flow field in Scenario 1, were the velocities in the southeast corner stay almost zero, a circulation exists in Scenario 2 which covers the whole basin and the in the southeast corner reaches velocities of 1m/s.
- The four circulations between the pontoons which develop in Scenario 1, do not arise in Scenario 2. The flow field between the west wall and the Pontoon A, shows a similar circulation as Scenario 1, only the velocities are lower. But on the other hand, there are no circular movements between the other pontoons in Scenario 2.
- The different pontoon height results in a flow with an opposite direction at the north wall, especially at t =900.
- Ebb tide looks the same in both scenarios; the water leaves the basin without any circulation and straight to the exit. The velocities at 3600s are just as in Scenario 1 almost zero.

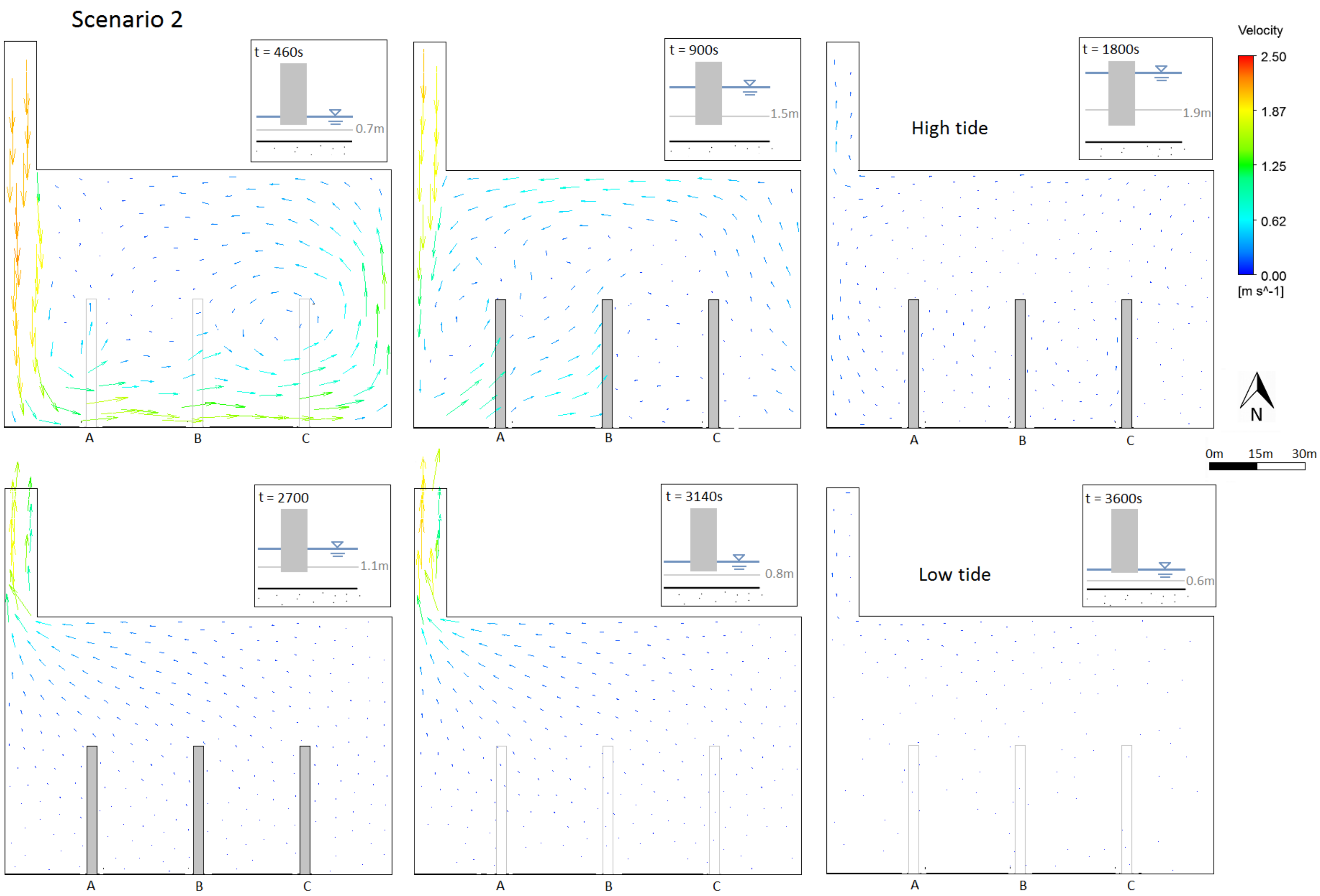

Figure 25 - Horizontal velocity flow fields of Scenario 2 at mid depth on 6 different time steps during a tidal cycle of 1 hour.

4.4 Scenario 3

The results of Scenario 3 are presented in Figure 26. In this last scenario the pontoons have the same depth as in Scenario 1, only the number of pontoons is different. There are only two pontoons placed on the south wall of the basin. The boundary conditions in this scenario are exactly the same as the previous scenarios so one can compare the results of Scenario 3 with the other scenarios.

4.4.1 Observations

The main observations for Scenario 3 are described below. But because the results of this scenario look so much like Scenario 1, the observations of Scenario 1 are not repeated; only the differences with Scenario 1 are described.

- The flow patterns in Scenario 3 develop three circulation cells, one less than in Scenario 1. Though this is consistent because the number of pontoons in Scenario 3 is also one less, the directions of these circulations are different. The directions in Scenario 1 are alternately clockwise and counter clockwise, but both the directions in Scenario 3 are not. The two most west circulations are both counter clockwise, while the third circulations is clockwise. The bigger distance between the pontoons enables the flow to develop the same direction in two circulations next to each other.
- Despite of the extra pontoon in Scenario 1, the flow fields at t = 460 look almost the same in both scenarios, only the velocities near the north wall in Scenario 1 slightly are higher. This continues in the following time step where in Scenario 1 the flow near the north wall is in an east direction while the flow pattern in Scenario 3 is more scattered.

4.4.2 Comparison with Scenario 0 and 1

When Scenario 3 is compared to the scenario without pontoons a few big differences are visible. Most of them are much the same as the comparison between Scenario 1 and Scenario 0.

- Just like Scenario 1, Scenario 3 does not show a development of a single circulation either during the tidal cycle. The flow field is more scattered and finally ends up in a circulation between each pontoon and the walls and the pontoons.
- Velocities in this scenario are much lower than the ones in Scenario 0. They have the same magnitude as the velocities in Scenario 1.
- Finally the ebb tide in Scenario 3 shows no sign of circulation at the flow fields, just like the flow fields during ebb tide of Scenario 1.

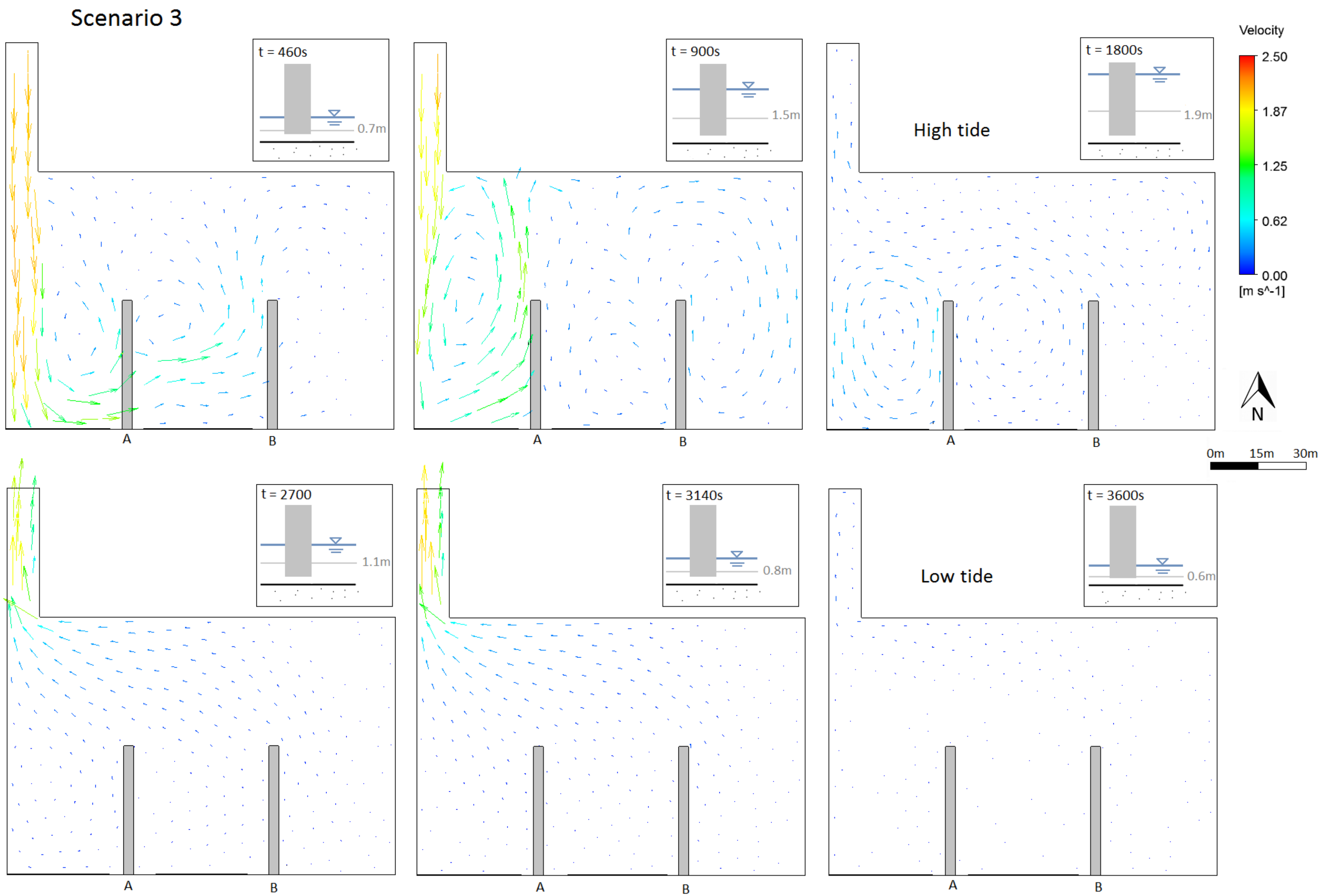

Figure 26 -Horizontal velocity flow fields of Scenario 0 at mid depth on 6 different time steps during a tidal cycle of 1 hour.

5. Discussion

In this project a 3D hydrodynamic model of a schematic harbour has been developed to predict horizontal flow fields in the harbour basin. The resulting flow fields are consistent with found flow patterns in the literature. However some assumptions and simplification have been made, mainly to reduce the time necessary for the calculations. This chapter discusses these limitations and the influence of choices that were made on the results. More time should be addressed to improve this model to gain better and more reliable results.

A first limitation of the model is the simulated period of the tidal cycle. The simulated period of 1 hour, using the same tidal range as in a 12 hour cycle, does not represent a real harbour. This results in much higher velocities at the inlet (as shown in Appendix B) and these high inlet velocities cause higher velocities through the whole domain. Due to the high inlet velocities a momentum is created inside the water body and the shown circulations in the basins probably last longer than circulations with lower initial velocities.

Secondly the results should be completely independent of the mesh size. As mentioned in Section 3.2.3 this point is not reached yet. Therefore the results could be slightly different when a finer mesh is used. However the maximum differences in the predicted velocities should not be higher than 10%, which is the difference between the predicted velocities of the used mesh and the second finest mesh.

Another issue is the water level during the simulations. The water level inside the domain does not rise as expected with these inlet boundary conditions during the simulation. The maximum water level at high tide in the harbour is 4m instead of 5.5m. The cause of this shortcoming is not figured out yet, but it might result in higher velocities through the domain because the boundary conditions are set for the same discharge with higher water levels.

By using FLUENT for the simulations, the air flow had to be included in the model. In all the simulations the air seems to have the same and sometimes even higher velocities than the water. This was not expected because the boundary conditions were set that no velocity was initialized for the air and the air should not interact with the water. The reason for these velocities in the air has not been found yet, but improvement must be made to increase the reliability of the model.

Finally, the laminar flow assumption is not realistic for a real harbour situation. When a turbulence model would be included in this project's simulations, the predicted velocities probably would be lower because of the turbulent flow pattern.

The use of a CFD model to simulate the water flow inside the harbour has lots of advantages compared to physical models. The simulations are more flexible, cost a lot less time and money to set up and the scenarios can be adapted very fast. However calculating mathematical models still has computer limitations and their accuracy depends on empirical values of constants used for the calculations. A sort of spurious precision is therefore present.

In brief the model has delivered flow fields that are in accordance with the used literature, but these results could be more reliable if some improvements to the model are made. However to implement these improvements a lot more time has to be spent on this project.

6. Conclusions & recommendations

This chapter answers the main research question of this research: ‘How does the placement of pontoons influence the flow field inside the harbour?’. The following section provides the conclusion of this research and the last section gives recommendations about the use of the results and for further research about this subject.

6.1 Conclusion

In this research a hydrodynamic model for a harbour has been established and is capable of simulating water flows inside a harbour. This model has been applied to a number of different scenarios for the placement of pontoons in a small rectangular harbour. Because of some limitations of the model the results are interpreted qualitatively. The essential effect of the pontoons is clear. The model has to be improved further for more reliable quantitatively results. The main effects of the pontoons on horizontal flow field inside a small tidal harbour are as follows.

The calculated effects of pontoons on horizontal circulations in a harbour are big and are visible in the direction and magnitude of the horizontal flow velocities. A single circulation cell, which is seen during the whole tidal cycle in a harbour without pontoons and is consistent with the literature, is not present in any case of a harbour with pontoons. Even when horizontal flow field at mid depth is situated below the pontoons this single circulation cell is not visible. Instead of this single circulation cell, several circulation cells are formed in harbours with pontoons, most of them between the pontoons or the pontoons and the walls. And whereas the circulation in a harbour without pontoons continues to circulate during ebb tide, the water in the harbours without pontoons shows no signs of circulation at all. The horizontal flow velocities inside the harbour with pontoons are more than twice as low compared to a harbour without pontoons. The pontoons form a barrier for the horizontal flow through a basin, which lower the velocities and change the flow patterns drastically.

The pontoon’s depth affects the circulation of horizontal flows in the harbour as well. Deeper pontoons enforce several circulations between the pontoons, while a more scattered flow field, lacking clear circulations, is developed when the pontoons are less deep. During ebb tide however, the depth of the pontoons does not affect horizontal flow field.

The number of pontoons in a harbour does not change the horizontal flow very much, taking into account that the number of circulations between pontoons and walls increase in accordance to the number of pontoons. However, the direction of these circulations can be different depending on the distance between the pontoons or the pontoon and the wall. Just like the pontoon depth, the number of pontoons does not affect the flow fields during ebb tide.

On the whole the pontoons have multiple effects on the horizontal flow fields inside a harbour. These effects can be influenced by the depth and number of pontoons. Finally, the effects of pontoons on the flow fields may contribute to the sedimentation in the new marinas such as the Arbroath harbour.

6.2 Recommendations

In this study a lot of information has been gathered about horizontal flow fields in a harbour. This information can be used to discover the effect of the pontoons on sedimentation if studied further. Meanwhile the used model can be further developed to investigate much more other features about flow fields in harbours.

This research shows harbour masters and other people associated with small harbours that pontoons affect the flow patterns inside the harbour. It shows that these people should think about the consequences of pontoons before placing them inside the harbour; however more research needed to indicate the exact consequences of the placement of pontoons. On top of that, the changed flow patterns could affect sedimentation in the harbour, which is a main issue and biggest expense for all harbour masters. It could lead to lower maintenance costs for small harbours when the influence of pontoons on sedimentation is further investigated and harbour designs are based on the outcomes.

Simultaneously the hydrodynamic model used in this project has much more potential than what is shown in this study. Using FLUENT creates a lot of opportunities and delivers much more output which can be used to investigate much more than only the horizontal flow fields.

For example more analysis of the obtained results in this project can describe the influence of pontoons on vertical velocities or describe how water flows around the pontoons. Adapting the simulated scenarios can lead to much more insights about the influences of pontoons in a harbour. To make the scenarios more realistic the pontoons can be modelled as floating objects and the pontoons can be modelled as a porous media which is more realistic when floating pontoons are used. Of course at the same time more plan form orientations of the pontoons can be analyzed using the same model.

Secondly, when the model would be further adapted it is possible to use particle tracking and include particulate materials. This enables the possibilities to follow the sediments through the harbour basin and analyze where the highest siltation is happening and where artificial measures have the most impact on reducing siltation. This would be a major contribution to the main reason for starting this project.

When one would like to investigate flow patterns in tidal open water systems maybe less complex models can be used such as Princeton Ocean Model (POM). These could probably be set up and calculated in less time than this project and will give insights in the water movement, but these models do not use the same fluid mechanics and do not deliver the same amount and detailed information as FLUENT.

All in all, this study provides information to think about for harbour masters or other people concerned with small harbours. Secondly much more useful information can be gained from this model in the future which can lead to results about how siltation is influenced by pontoons in the harbour.

References

ANSYS, Inc. (2009). *Introduction to CFD.* Several lectures at Introductory FLUENT training form ANSYS, Inc.

ANSYS, Inc. (2013a). ANSYS FLEUNT User's guide. Retrieved 30-04-2013 from ANSYS.com/en_uk

ANSYS, Inc. (2013a). ANSYS FLEUNT Theory guide. Retrieved 30-04-2013 from ANSYS.com/en_uk

Arbroath sailing and boating club. (2013). [Online picture]. Downloaded on 19-05-2013, http://www.Arbroathsailingandboatingclub.org.uk/

Centre for Coastal & Marine Sciences (2013), Arbroath tide tables 2013. Denton Island: Douglas Press

Jiang, J.X., Falconer, R.A. (1983). On the tidal exchange characteristics of model rectangular harbours. *Proceedings - Institution of Civil Engineers,* 75, Sept., 475-489

Jung, K.H., Chang, K., Huang, E.T. (2003) Two-dimensional flow characteristics of wave interaction with a fixed rectangular structure. Ocean Engineering 31, 975-988

Falconer, R.A. (1980) Modelling of planform Influence on circulation in harbours. Coastal Engineering Proceedings, 1(17), 2726-2744

Nece, R.E., Falconer, R.A. (1989). Hydraulic modeling of tidal circulations and flushing in coastal basins. *Proceedings - Institution of Civil Engineers,* 86, Oct., 913-935

Saalbach, J., Zorndt, A., Krämer, K., Schlurmann, T. (2012). Reducing siltation in the Juist marina. Retreived 05-05-2013 from, Leibniz Universität Hannover, Franzius-Institute for Hydraulic, http://www.fi.uni-hannover.de

Sayma, A. (2009). Computational Fluid Dynamics. Abdulnaser Sayma & Ventus Publishing ApS

Stamou, I., Katsiris, I.K., Moutzouris, C.I., Tsoukala, V.K. (2004). Improvement of marina design technology using hydrodynamic models. *Global Nest: The international journal.* No 1, 63-72

Stochek, O., Zimmermann, C. (2005) Water exchange and sedimentation in an estuarine tidal harbour using three-dimensional simulation. *Journal of Waterway, Port, Coastal and Ocean Engineering*, 132, 410-414

Wendt, J.F. (2009). Computational Fluid Dynamics: An Introduction. Springer-Verlag Berling Heidelberg.

Yin, J., Falconer, R.A., Chen, Y., Probert, S.D. (2000). Water and sediment movements in harbours. *Applied energy*, 341-352

Appendix A. User Defined Function

The following C++ code is used as a User Defined Function in the FLUENT model to prescribe the input properties for each cell of the Velocity Inlet at each time step.

```
#include "udf.h"

DEFINE_PROFILE(velocity,thread,index)
{
 face_t f;
real x[ND_ND];
real PI;
real f_time = RP_Get_Real("flow-time");
real Y;
begin_f_loop(f,thread)
PI=3.1415926;

{     F_CENTROID(x,f,thread);
 Y=-2.25*cos((2*PI/3600)*f_time)+3.25;

if (x[1]<=Y)
          F_PROFILE(f,thread,index)=4.5*PI/3600*sin((2*PI/3600)*f_time)*(120*80+10*40)/(10*Y
);
else
          F_PROFILE(f,thread,index)=0.0;
}
end_f_loop(f,thread)
}

DEFINE_PROFILE(volumefraction,thread,index)
{
face_t f;
real x[ND_ND];
real Y;
real PI;
real f_time = RP_Get_Real("flow-time");
begin_f_loop(f,thread)
PI=3.1415926;

{     F_CENTROID(x,f,thread);
Y=-2.25*cos((2*PI/3600)*f_time)+3.25;

if (x[1]<=Y)
          F_PROFILE(f,thread,index)=1;
else
          F_PROFILE(f,thread,index)=0.0;
}
end_f_loop(f,thread)
}
```

Appendix B. Effect of shorter tidal cycle

The following two figures show the water level, mass flow and velocities at the inlet for the tidal harbour, both have the same scale to enhance comparison. Figure 1 shows the quantities when the tidal cycle is 12 hours. The second (Figure 2) shows the same variables for a 1 hour tidal cycle.

Due to this shorter tidal cycle the change of the water level per minute is much higher and therefore the mass flow and the velocity at the inlet are higher. The highest inlet velocities for the 12hrs cycle are 0.14 m/s compared to the 1.7 m/s for the 1 hour cycle.

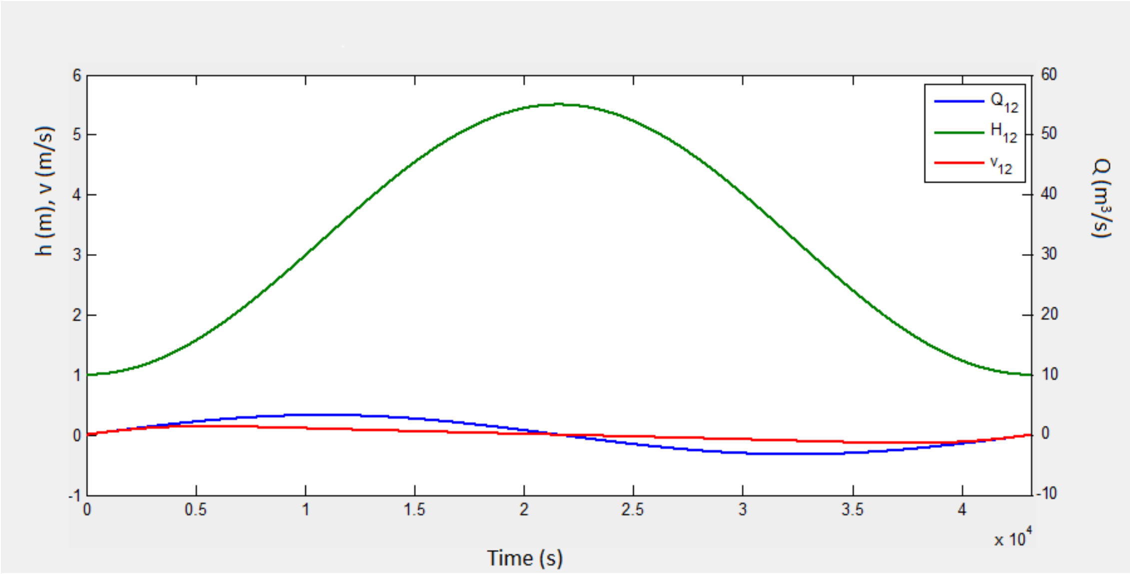

Figure 1 - Variations of the water level and corresponding velocity and mass flow for a 12hrs tidal cycle. Mean water depth of 3.25m and amplitude of the water level's variation 2.25m

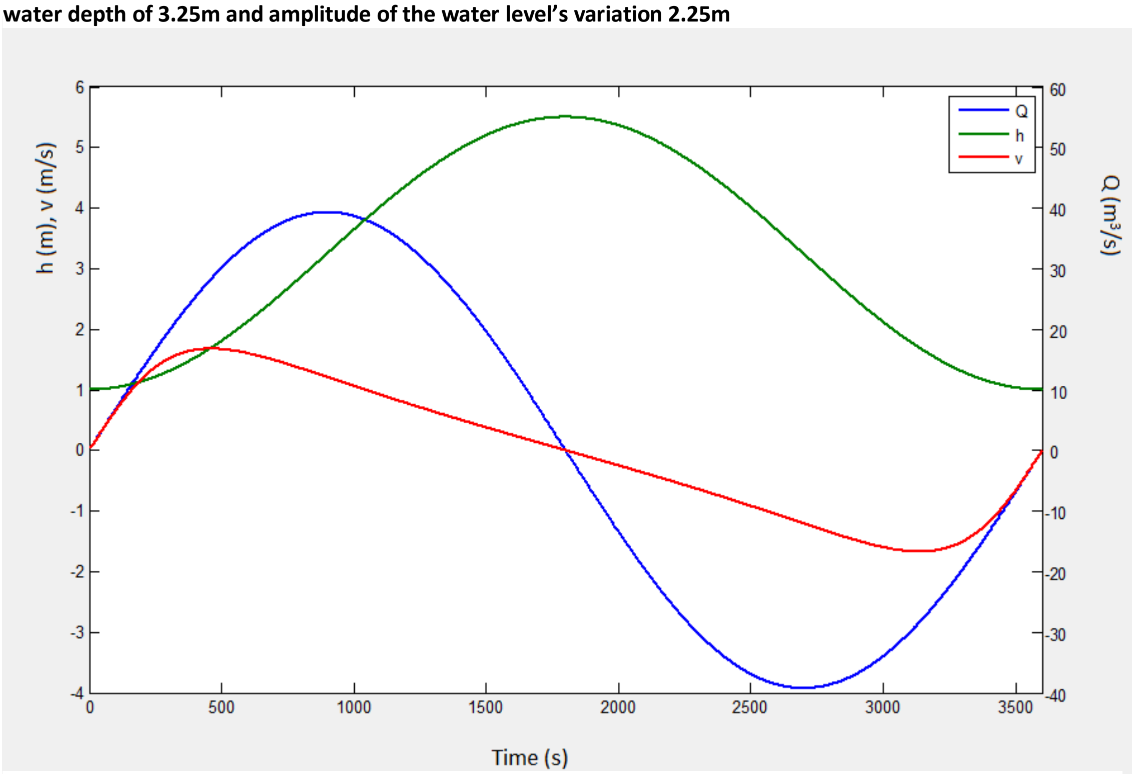

Figure 2 - Variations of the water level and corresponding velocity and mass flow under tidal conditions with a period of 3600s, mean water depth of 3.25m and amplitude of the water level's variation 2.25m.

Appendix C. Vertical water movements

This appendix shows that horizontal flow fields at the same time on different water depths do not show the same flow fields, the flow is affected by the vertical flow around the pontoons. Figure 1 shows the horizontal flow fields for two scenarios, for each scenario the left flow field is at mid depth and the right flow field is at a fixed height of 0.9m. For Scenario 0 both flow fields are almost the same. But in Scenario 2 the flow fields differ from each other. The flow field at 0.9m is situated below the pontoons and the flow has higher velocities and a more clear circulation than the flow field at mid depth.

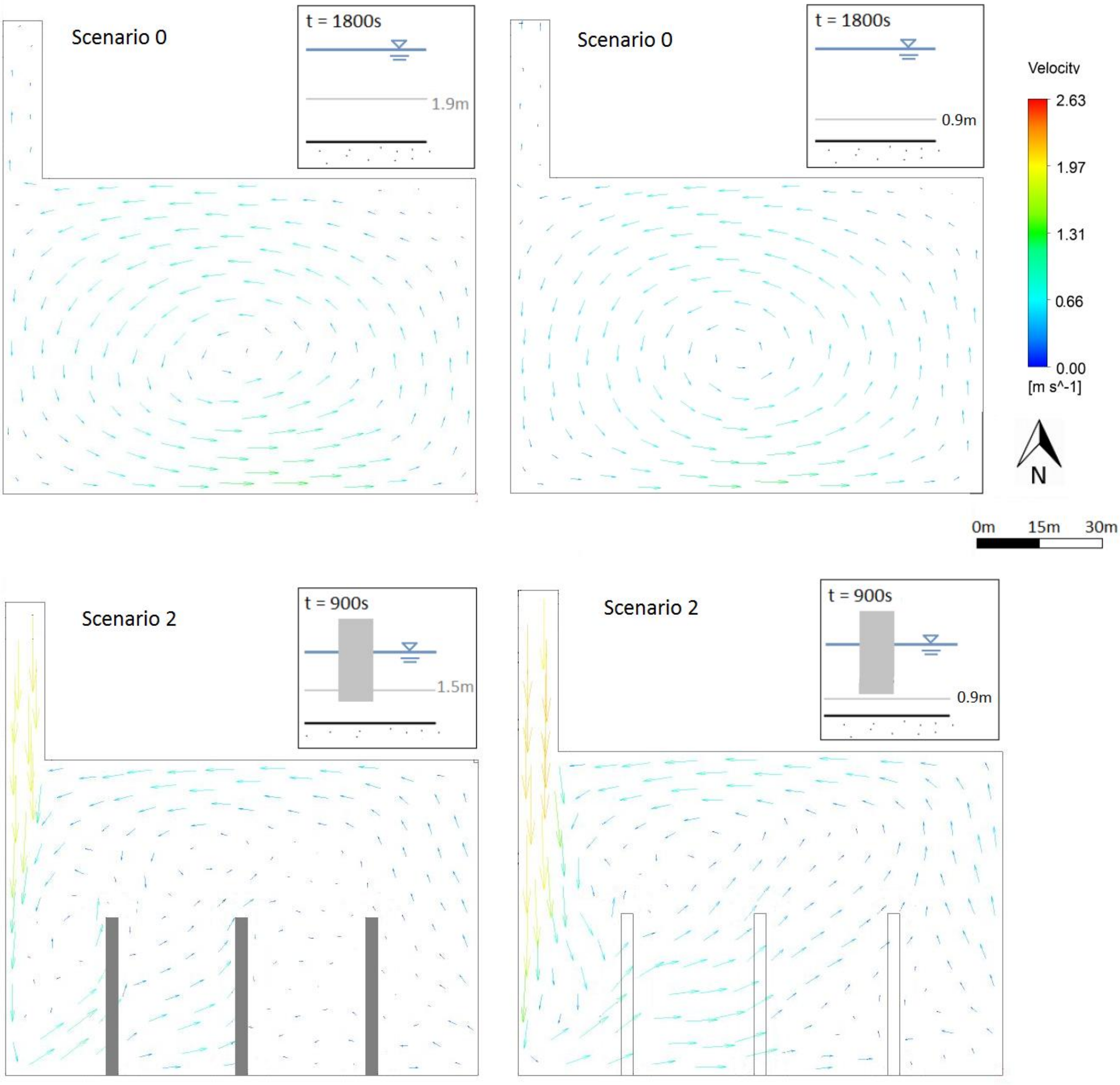

Figure 1 - Horizontal flow fields at different depths for Scenario 0 and 2

Figure 2 shows the absolute water velocity on 4 different vertical lines(Figure 3) in the domain at 900s for Scenario 2. The pontoons in this scenario are situated a meter above the bottom. The graph does not indicate the directions of the velocity vectors, only the magnitude, but it shows that the velocities are highly influenced by the pontoons. The velocities in the water under the pontoons(0-1m) are a lot higher than inbetween two pontoons or the pontoons and the walls(1-3m).

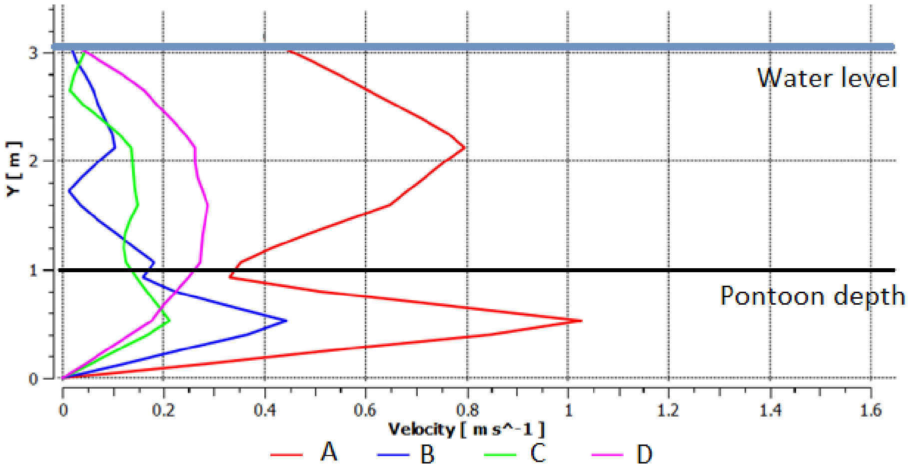

Figure 27 – Velocity magnitude between 0 and 3m for Scenario 2, t=900s

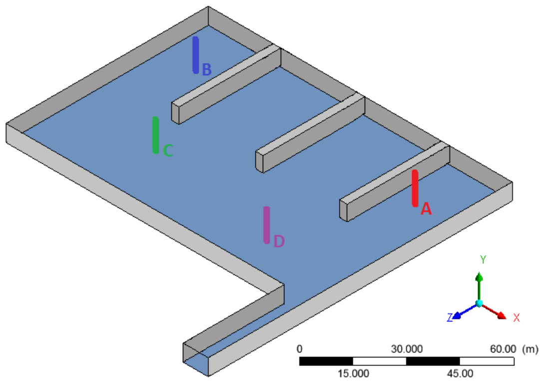

Figure 3 - Measure points A, B, C and D situated in domain