

On the effect of non-hydrostatic simulation on buoyant jets

Firmijn Zijl

M.Sc. Thesis, September 2002
Delft University of Technology
Faculty of Civil Engineering and Geosciences
Section Fluid Mechanics

Graduation committee:

Prof. dr. ir. G.S. Stelling (chairman)
Prof. dr. ir. J.A. Battjes
Ir. J.A.Th.M. van Kester
Dr. ir. A.C.H. Kruisbrink
Prof. dr. ir. H.J. de Vriend

Abstract

As the title says, this thesis investigates the effect of non-hydrostatic simulation on the stability of buoyant jets. A jet is an injection of water with a certain velocity (better: momentum) in an ambient water body. ‘Buoyant’ refers to the temperature of the jet water being different from that of the surrounding (ambient) water. This so-called buoyancy has an effect on the flow of the jet. Two regions in a jet can be distinguished: the near-field and the far-field. Each region has its own hydrodynamic characteristics. Strictly speaking, in the near-field, the use of the hydrostatic pressure relation is not justified. (‘Hydrostatic’ means that vertical accelerations are neglected in the determination of the pressure.) This might give rise to instabilities in computational simulations.

A jet simulation (Point Beach Power Station case) with Delft3D and CFX showed that the far-field results of a non-hydrostatic model can be similar to those of a non-hydrostatic model. Especially the jet trajectory, the centreline dilution and the layer thickness of the plume using the hydrostatic program Delft3D-FLOW where in good agreement with non-hydrostatic results from CFX. This confirms the idea that a stable (but not physically correct) near-field with the buoyant jet water coming to the surface can be enough to provide accurate results in the far-field.

Two-dimensional (2DV) simulations in Delft3D and CFX show that the near field of a hydrostatic simulated jet is indeed physically incorrect. This is shown by unrealistically high vertical velocities located where the jets enter the ambient water body and an overestimated angle of the density fronts. It is also shown that hydrostatic simulation can lead to results where the (positively) buoyant jet water does not come to the surface.

To further investigate the effect of the hydrostatic pressure assumption, simulations are made using a program that can switch between a hydrostatic and a non-hydrostatic mode by switching the pressure correction off and on. Simulations of two purely buoyancy driven cases showed improvements resulting from non-hydrostatic simulation (compared to hydrostatic simulation): more realistic (lower) vertical velocities in the density fronts, a smoother spatial velocity distribution and a better agreement between the theoretically predicted and simulated density fronts. The main differences and highest dynamic pressures were found at locations with high horizontal pressure gradients (density fronts). Horizontal front speeds were almost unaffected by the pressure correction.

Simulation of various buoyant jets showed that non-hydrostatic simulation is necessary to prevent physically incorrect results. Only non-hydrostatic simulations yielded correct jet trajectories. Furthermore, the same improvements as in the purely buoyancy driven cases were noticeable here.

Attachment of the jet to the bottom in the hydrostatic runs was accompanied by a high densimetric Froude number at the outfall ($F_0 > 5$) or a strong ambient (co-) flow. In the case of the strong ambient current, non-hydrostatic simulation did not improve the situation. In the case of a high densimetric Froude number, non-hydrostatic simulation made the warmer water to come

to the surface and form a stable layer. This is an important improvement compared to non-hydrostatic simulation.

A non-hydrostatic 2DV program capable of uncoupling the number of pressure and velocity layers is used to investigate if by diminishing the number of pressure layers stable non-hydrostatic results can still be obtained. To accommodate a variable density and thus buoyancy induced flow, his program was first extended with a transport equation for heat and a baroclinic pressure term. After verification of the model, simulations showed that, with respect to the simulation of purely density driven flow, it can be concluded that diminishing the number of pressure layers yields accurate results.

In case of buoyant jets, the number of pressure layers can be decreased with virtually no negative influence if the pressure layer thickness remains smaller or equal to the vertical dimension of the outlet.

Further lowering the number of pressure layers causes the near-field to become physically incorrect with the jet water immediately going to the free surface. However, unlike in the hydrostatic computations, this does not result in an irregular velocity vector field. The far-field remains stable. This means that using only a few pressure layers is certainly an important improvement compared to hydrostatic modelling.

Acknowledgements

This report has been written in the framework of my Master of Science (M.Sc.) thesis as part of the study Civil Engineering at Delft University of Technology. The work has been carried out at WL | delft hydraulics.

I would like to express my gratitude to the people at WL | delft hydraulics and especially prof. dr. ir. A.E. Mynett, for giving me this opportunity.

Specifically, I would like to thank my supervisors, prof. dr. ir. G.S. Stelling (Delft University of Technology), prof. dr. ir. J.A. Battjes (Delft University of Technology), ir. J.A.Th.M. van Kester (WL | delft hydraulics), dr. ir. A.C.H. Kruisbrink (WL | delft hydraulics) and prof. dr. ir. H.J. de Vriend (WL | delft hydraulics & Delft University of Technology) for their suggestions and comments during my research.

Furthermore I am grateful to my friends, family and fellow students for their support and interest in my study.

Delft, September 2002,

Firmijn Zijl

Table of contents

Abstract	iii
Acknowledgements	v
Table of contents	vii
1 Introduction	1
1.1 General	1
1.2 Problem definition.....	1
1.3 Objectives of this study.....	2
1.4 Outline of this report	2
2 Physical processes involved in buoyant jets	5
2.1 Introduction	5
2.2 Ambient conditions	5
2.3 Discharge conditions	6
2.4 Near-field processes	6
2.4.1 <i>Submerged buoyant jet mixing</i>	6
2.4.2 <i>Surface buoyant jet mixing</i>	7
2.4.3 <i>Boundary interaction processes</i>	8
2.5 Far-field processes.....	10
2.5.1 <i>Buoyant spreading</i>	10
2.5.2 <i>Passive ambient diffusion</i>	11
3 Governing equations	13
3.1 Introduction	13
3.2 The Equations	13
3.2.1 <i>Conservation of mass</i>	13
3.2.2 <i>Conservation of momentum</i>	14
3.2.3 <i>Boussinesq approximations</i>	15
3.2.4 <i>Conservation of energy</i>	16
3.2.5 <i>Equation of State</i>	17
3.3 Turbulence treatment	17
3.4 Equations in a Cartesian co-ordinate system.....	20
3.5 Equilibrium and stability of stratification	21
3.6 Important (dimensionless) parameters	22
3.7 Subdividing the pressure	23
4 Flow modelling software	25
4.1 Delft3D-FLOW	25
4.1.1 <i>Introduction</i>	25
4.1.2 <i>Equations</i>	25
4.1.3 <i>Hydrostatic pressure assumption</i>	27
4.1.4 <i>Turbulence modelling</i>	28

4.1.5 Boundary conditions.....	29
4.2 CFX.....	30
4.2.1 Introduction	30
4.2.2 Buoyancy model.....	31
4.2.3 Turbulence modelling.....	31
4.2.4 Boundary conditions.....	32
4.2.5 Methods to improve convergence rate.....	33
4.3 CORMIX.....	33
4.3.1 Introduction	33
4.3.2 Length scales.....	34
4.3.3 Notes on the use of CORMIX	35
5 Test case Point Beach Power Station.....	37
5.1 Introduction	37
5.2 Outline of the case.....	37
5.2.1 General outline.....	37
5.2.2 The grid	38
5.2.3 Boundary conditions.....	39
5.2.4 Implementation of the jet.....	41
5.2.5 Computations and available data.....	41
5.3 The results	41
5.3.1 Trajectory of the plume centreline.....	42
5.3.2 Centreline temperature decay.....	43
5.3.3 Vertical temperature profile.....	44
5.3.4 Plume thickness.....	46
5.3.5 The near-field.....	46
5.4 Sensitivity analysis	47
5.4.1 Open boundaries.....	48
5.4.2 Discharge method.....	48
5.4.3 Results of the sensitivity analysis	49
5.5 The temperature distribution around the centreline.....	52
5.6 Conclusions.....	55
6 2DV jets with Delft3D and CFX	57
6.1 Introduction	57
6.2 Delft3D buoyant jet simulation	57
6.2.1 Outline of the case	57
6.2.2 Implementation.....	58
6.2.3 Results	59
6.3 CFX buoyant jet simulation.....	60
6.4 Sensitivity analysis	60
6.5 Modified pressure.....	62
6.6 Conclusions.....	63
7 2DV model with pressure correction	65
7.1 Introduction	65

7.2 Lock exchange	65
7.2.1 <i>Model set-up</i>	65
7.2.2 <i>Expected phenomena</i>	66
7.2.3 <i>Results</i>	67
7.2.4 <i>Comparison with theory</i>	67
7.3 Intrusive gravity current	70
7.3.1 <i>Model set-up</i>	70
7.3.2 <i>Expected phenomena</i>	70
7.3.3 <i>Results</i>	71
7.3.4 <i>Comparison with theory</i>	71
7.4 Buoyant momentum jet	72
7.4.1 <i>Model set-up</i>	72
7.4.2 <i>Results</i>	73
7.5 Conclusions	75
8 2DV hydrodynamic sigma model with pressure interpolation	77
8.1 Introduction	77
8.2 Lock exchange - sigma layer model	77
8.2.1 <i>Model set-up</i>	77
8.2.2 <i>Results</i>	77
8.2.3 <i>Comparison with theory</i>	78
8.3 Intrusive gravity current - sigma layer model	80
8.3.1 <i>Model set-up</i>	80
8.3.2 <i>Results</i>	80
8.3.3 <i>Comparison with theory and fixed layer model</i>	80
8.4 Buoyant momentum jet	81
8.4.1 <i>Model set-up</i>	81
8.4.2 <i>Results</i>	81
8.5 Pressure interpolation	82
8.5.1 <i>Lock Exchange</i>	82
8.5.2 <i>Intrusive gravity current</i>	83
8.5.3 <i>Buoyant momentum jets - run1</i>	84
8.5.4 <i>Buoyant momentum jets - run2</i>	85
8.6 Conclusions	87
9 Conclusions and recommendations	89
9.1 Conclusions	89
9.2 Recommendations	91
References	93
List of main symbols	95
Appendix A: Simplification of Continuity Equation	
Appendix B: Energy equation	
Appendix C: Derivation of kinematic boundary conditions	
Appendix D: Depth integrated Continuity Equation	

Appendix E: Sigma transformation

Appendix F: Derivation of the 2DV hydrodynamic sigma model

Appendix G: Transformation and discretisation of the baroclinic pressure term

Figures

1 Introduction

1.1 General

Many industries use water in their production process. This water is often not used as a primary product and has to be disposed of once the production value has diminished. For example, in nuclear and coal power plants, water is used as cooling water. Once the water is heated to a certain level, the water has to be disposed of. Usually this means discharging in a large water body, often the same as where the water came from before usage.

Interest is taken in the prediction of the discharged water flow. In this example, it should be prevented that the heated discharge water flows to the intake point. This would cause inefficiency in water usage or overheating of the plant. Predictive studies like this are called recirculation studies. Other cases where interest is taken in the prediction of the flow and density distribution are environmental studies, for example when toxic substances are discharged with the jet water.

Nowadays, simulation is done with computational 3D hydrostatic models (e.g. Delft3D, TRIWAQ). The problems for which these flow solver packages are used for generally, have much larger length scales in the horizontal directions than in the vertical direction (e.g. estuaries, seas and lakes). This makes it justifiable to use a hydrostatic assumption in the model, which means that the vertical accelerations are neglected in the determination of the pressure. Still, vertical flow can occur, when more than one layer is used. The hydrostatic assumption makes the model much faster to solve, which is of tremendous importance for practical purposes.

1.2 Problem definition

In some situations, the vertical accelerations are not to be neglected. In case of buoyant jets, this is the case immediately after the water is discharged. ‘Buoyant’ refers to the fact that the discharged water is of different density than the surrounding (‘ambient’) water. It is a well-known phenomenon that warmer water floats over colder water. Due to this principle, the warmer discharged water is forced to the surface, thereby undergoing a considerable vertical upward acceleration. This process occurs in the immediate surroundings of the outfall (the near-field). Further away, in the far-field, vertical accelerations are of less importance.

If a hydrostatic model is used, this can give problems with instabilities and unrealistic results caused by the incorrect simulation of the near-field. Because usually only interest is taken in the far-field, it is an option to omit the near-field in the simulation. An online coupling with another program predicting the near-field (e.g. CORMIX) is then used to insert the flow and density data in the far-field.

Another option, which will be the subject of this study, is the use of non-hydrostatic modelling in the near-field. Increased computational resources make the use of three-dimensional models

feasible. The idea is that the non-hydrostatic model may prevent instabilities and yield more accurate results, which may also help to increase the accuracy in the far-field.

It is stressed that the main focus is the prevention of instabilities, not to have physically correct predictions of the flow around the outfall. This would demand a totally different model, with for example a much finer grid and smaller time steps. Also turbulence modelling would be more important there. An advantage of this approach is that the near-field, also influencing the ambient flow around the outfall, does not have to be totally neglected. Furthermore, the use of non-hydrostatic modelling has a more general scope and can be beneficial in more situations.

Another aim of this study is to establish if the use of only a few pressure layers (less than the number of velocity layers) is enough to have the expected stabilising effect.

1.3 Objectives of this study

Summarising, the objectives of this study are:

- to get a better understanding of the effect of non-hydrostatic simulation on the near-field of positively buoyant jets. The focus is more on the forming of a stable upper layer than on the exactly physically correct results.
- to investigate if the use of only few pressure layers (less than the number of velocity layers) proves to be enough to obtain the desired stable non-hydrostatic results.

1.4 Outline of this report

Chapter 2 describes the physical processes involved in buoyant discharges. Also some often-used terms are introduced here.

In chapter 3 the equations governing the flow of fluids and the transport of heat and dissolved matter are explored. Special attention is given to the assumptions made in the process of deriving the necessary equations from more general principles of conservation.

Chapter 4 describes the commercial flow packages that are used in this study (DELFT3D-FLOW, CFX and CORMIX).

In chapter 5 a three-dimensional test case is simulated in Delft3D-FLOW, to see the problems and difficulties involved in the hydrostatic simulation of buoyant discharges. The simulations are made in CFX as well, because this package uses the full (non-hydrostatic) 3D equations.

In chapter 6 more 2DV simulations are made trying to see what the influence of different parameters is on the stability of the results in Delft3D. Also a CFX simulation is made.

In chapter 7 the effect of non-hydrostatic modelling on the occurrence of instabilities is examined.

Chapter 8 introduces a sigma-layered model suited to uncouple the number of computational pressure and velocity layers. The effect of lowering the number of pressure layers is examined.

In chapter 9 the conclusions of this study and recommendations for further research are described.

2 Physical processes involved in buoyant jets

2.1 Introduction

This chapter describes the physical processes involved in buoyant jets. Also some often-used terms will be introduced here.

When effluents are discharged, mixing with the ambient water occurs. The mixing behaviour of any jet is governed by the interaction of ambient conditions in the receiving water and by the discharge characteristics. In sections 2.2 and 2.3 the distinction between the ambient and the discharge conditions will be further explained.

Another important notion is splitting up the discharge region in a near-field and a far-field. This is done because different conditions and processes play a role in these regions. This distinction and the hydrodynamic processes that take place there will be examined in sections 2.4 and 2.5.

When doing a study on, for example, cooling water discharges, often most interest is taken in the temperature distribution in the receiving water body. However, to predict the temperature distribution, one needs to understand the hydrodynamic processes, as the transport of heat is dependent on the flow of the water and vice versa (cf. Figure 2-1).

Also, to predict the temperature distribution in the far-field, one needs to understand the hydrodynamic processes in the near-field. This is a region with complex phenomena and even if there is no interest in the temperature distribution there, in principle, it cannot be neglected in the hydrodynamic computations as the far-field is dependent on it.

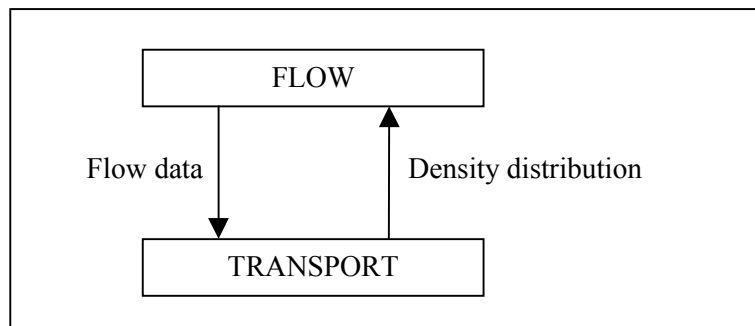


Figure 2-1: *Coupling of flow and transport.*

2.2 Ambient conditions

The ambient conditions in the receiving water body are described by geometric, kinematic and dynamic characteristics. Important geometric parameters include bathymetry, the plan shape and the vertical cross-sections.

Kinematic and dynamic characteristics are described by respectively the velocity and density distribution.

In many cases, the ambient conditions can be seen as steady state, because the time scale of the mixing process has usually a smaller order of magnitude than that of the changes in the conditions of the receiving water body. However in tidally influenced flows this assumption may be inappropriate.

2.3 Discharge conditions

The discharge conditions can be subdivided in the geometric and flux characteristics of the outfall installation. Important geometric parameters here are the location and orientation of the jet relative to the receiving water body and the shape and size of the discharge port. The flux characteristics are given by the effluent discharge flow rate, its momentum flux and by its buoyancy flux. The buoyancy flux represents the effect of the difference in density between the effluent and the ambient water. This difference, in combination with the gravitational acceleration, causes the effluent flow to rise (positive buoyancy) or to fall (negative buoyancy).

The hydrodynamic mixing process, caused by the jet, changes its character along the jet trajectory. Because of this rather strong change, the mixing process usually is conceptually split up in two regions: the near-field and the far-field.

Sometimes the mid-field is also distinguished. This region is defined as the region in which both near-field and far-field processes cannot be neglected.

Below, an extensive description of the near-field and far-field is given.

2.4 Near-field processes

The first region in the jet trajectory, the near-field, can be generally described as the region in which the initial discharge conditions like momentum flux, buoyancy flux and outfall geometry influence the jet trajectory and mixing. Here, mixing can be intense, causing rapid temperature changes.

Three important near-field processes are distinguished: submerged buoyant jet mixing, surface buoyant jet mixing and boundary interactions.

2.4.1 Submerged buoyant jet mixing

A submerged effluent flow at the discharge port can be conceptualised as a velocity discontinuity between the discharged fluid and the ambient fluid. This causes a shear stress, forcing the flow to become highly turbulent. The width of the zone of high turbulence intensity increases in the direction of the flow. This happens by means of incorporating less turbulent fluid from outside the jet and is called ‘entrainment’ (Figure 2-2a). In this manner internal concentrations (e.g. fluid momentum, dissolved mass (e.g. salt) or temperature) of the discharged flow become gradually diffused into the ambient field.

The flux at the outfall causing the velocity discontinuity can be of different nature: initial momentum flux and initial buoyancy flux.

If there is only a flux of momentum, one speaks of a pure jet (also called momentum jet or non-buoyant jet). In this case the effluent is injected with a high velocity into the ambient fluid.

In case of a pure plume (where initial momentum flux is absent) the initial buoyancy flux leads to a vertical acceleration, causing the velocity discontinuity.

In the general case, both types of fluxes are present. This is referred to as a buoyant jet (or forced plume).

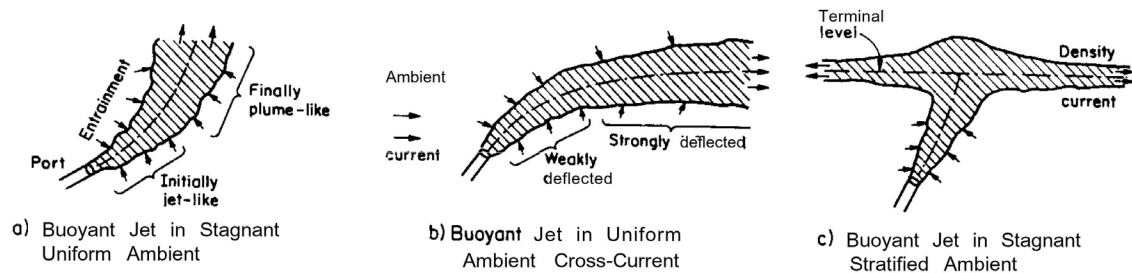


Figure 2-2: *Buoyant jet mixing flow patterns under different ambient conditions: (a) Stagnant ambient conditions, (b) Uniform ambient crosscurrent and (c) Stagnant stratified ambient conditions. [Jirka 1996].*

Density stratification and current are ambient conditions that play a role in the near-field. The ambient current deflects the jet in the direction of the current, thereby inducing additional turbulent mixing (Figure 2-2b).

Ambient density stratification can counteract the vertical acceleration, which is caused by the buoyancy. In this way the flow jet can become trapped between two layers of different density (Figure 2-2c).

2.4.2 Surface buoyant jet mixing

Positively buoyant surface jets are in some way the same as submerged buoyant jets. This is especially true at short distances from the outfall, where both horizontal and vertical spreading take place due to turbulent mixing.

After this stage, the buoyant damping suppresses the turbulence and associated vertical entrainment, forcing the warmer fluid to form a thin layer at the surface of the receiving water. By becoming increasingly thinner the jet also is forced to spread laterally.

In a stagnant ambient environment, the plume will continue to develop by means of buoyant spreading as is depicted in Figure 2-3a.

In the presence of an ambient cross flow, there are three scenarios. If the cross flow is weak compared to the initial momentum, the jet will not interact with the shoreline in the near-field (Figure 2-3b). When the cross flow is strong, the plume will attach to the downstream boundary (shore line), inducing a recirculation region (Figure 2-3c). The third scenario is when a high

initial buoyancy flux is combined with a weak cross flow. The buoyant spreading effects are so strong in this case that the plume also intrudes in upstream direction, staying close to the shoreline (Figure 2-3d).

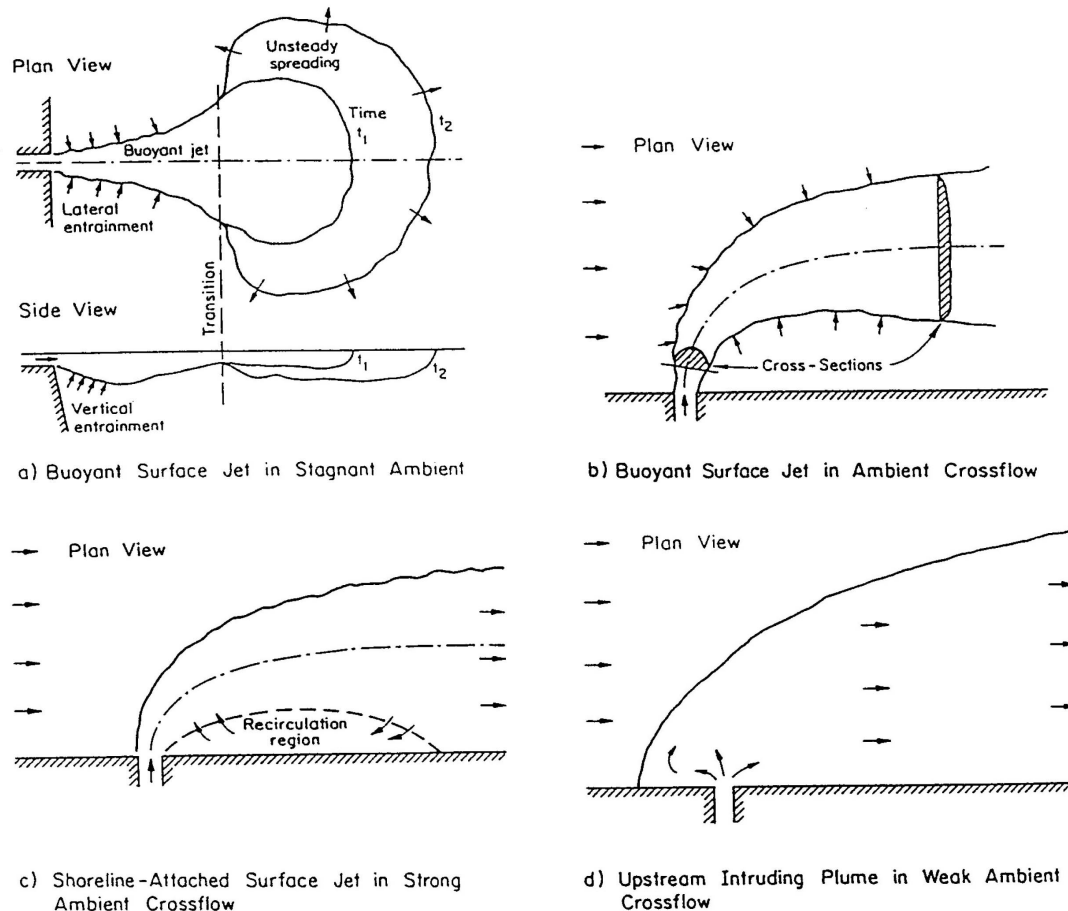


Figure 2-3: *Buoyant surface jet mixing flow patterns under different ambient conditions: (a) Stagnant ambient conditions, (b) Weak uniform ambient cross-current, (c) Strong stagnant stratified ambient conditions and (d) Weak uniform ambient cross flow with high buoyancy. [Jirka 1996].*

2.4.3 Boundary interaction processes

An ambient water body always has horizontal boundaries. Always present are the bottom and water surface, but sometimes also ‘internal boundaries’ might exist in the form of a pycnocline. This is a layer of sudden density change, possibly trapping the jet.

A few examples of surface boundary interactions are given in Figure 2-4. If the jet approaches the free surface gradually, due to cross flow little additional mixing will occur. Sufficient buoyancy will lead to a stable layer at the surface (Figure 2-4a). If the ambient flow is weak, the plume can

intrude in the upstream direction. On the other hand, if the buoyancy of the effluent is low, or the momentum high, unstable circulation can occur. Already mixed water can re-entrain due to these local circulations and full vertical mixing occurs (Figure 2-4b). In Figure 2-4c an intermediate case is shown, with local mixing and upstream spreading.

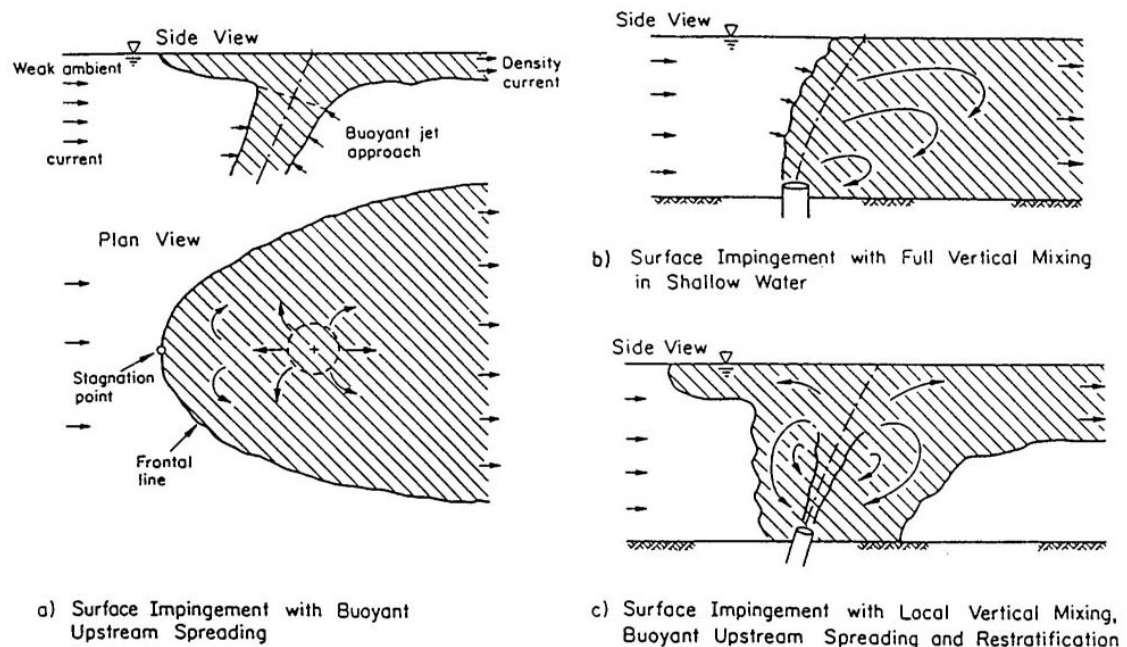


Figure 2-4: *Boundary interactions for submerged jets in a finite depth: (a) Stable situation with upstream intrusion, (b) Unstable situation with full vertical mixing and (c) Intermediate situation. [Jirka 1996].*

Another type of boundary interaction processes concerns the situation where submerged jets are discharged near the bottom or a wall. Two types of attachment can occur: wake attachment due to ambient cross flow (Figure 2-5a) or Coanda attachment forced by the entrainment demand of the jet itself (Figure 2-5b). In case of a Coanda attachment an area of low pressure can be found between the initial jet and the bottom.

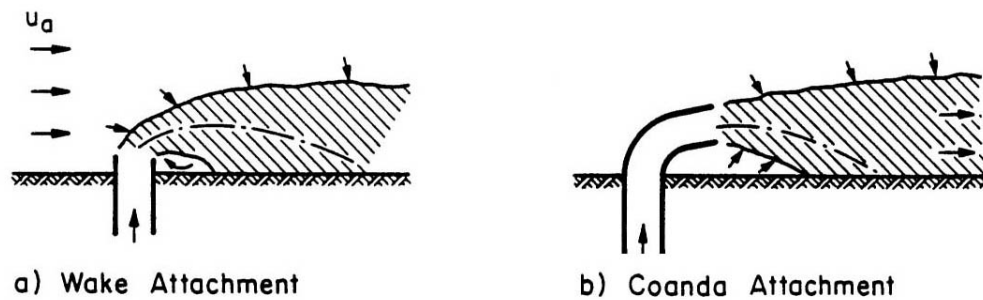


Figure 2-5: *Boundary interactions of jets discharging near boundaries: (a) Wake attachment (cross flow induced) and (b) Coanda attachment. [Jirka 1996].*

2.5 Far-field processes

As the jet travels further away from the source, the initial discharge conditions become less important. Now ambient characteristics will control trajectory and mixing of the turbulent plume. Important processes in the far-field are buoyant spreading and passive diffusion due to ambient turbulence. The transfer of excess heat to the atmosphere is also important in this region.

2.5.1 Buoyant spreading

Buoyant spreading is defined as the horizontal transverse spreading of the effluent flow, while the ambient current is advecting it downstream. The spreading is caused by the density difference between the plume and the surrounding fluid. When there is a large buoyancy effect the plume can spread rapidly in the direction perpendicular to the jet centreline trajectory, thereby decreasing its vertical thickness. In the frontal zone, some ambient fluid will entrain, although the mixing rate is usually small. In Figure 2-6 this spreading process is schematised at the water surface. Note that this can also occur at the bottom in case of negatively buoyant plumes or at pycnoclines when there is ambient density stratification.

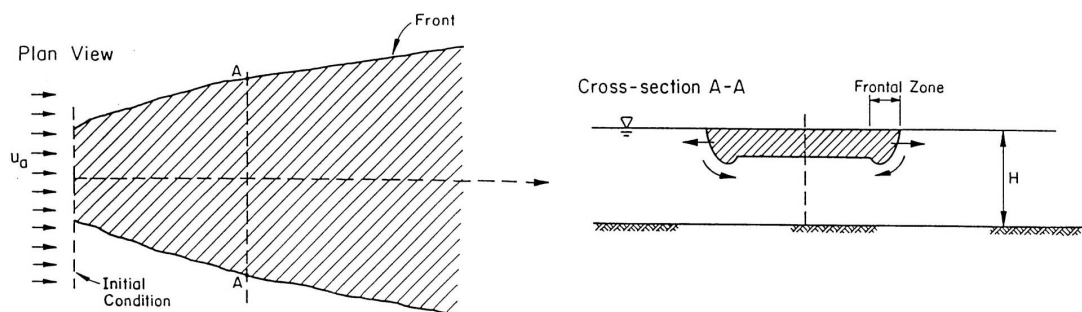


Figure 2-6: *Buoyant spreading processes along the water surface in the far-field region. [Jirka 1996].*

2.5.2 Passive ambient diffusion

Further in the far-field, when the buoyant spreading decreases, passive ambient diffusion becomes the main mixing mechanism. This means that the diffusion is caused by the existing turbulence in the ambient environment. The plume will grow in thickness and width until it reaches a bottom or vertical boundary. The strength of the ambient diffusion depends on the ambient conditions. In case of bounded flow (rivers, small estuaries) the spreading process can be described by constant diffusivity in both directions. However, in unbounded ambient regions, the plume growth will accelerate [Jirka 1996, p. 11]. See Figure 2-7 for a schematisation of this far-field process.

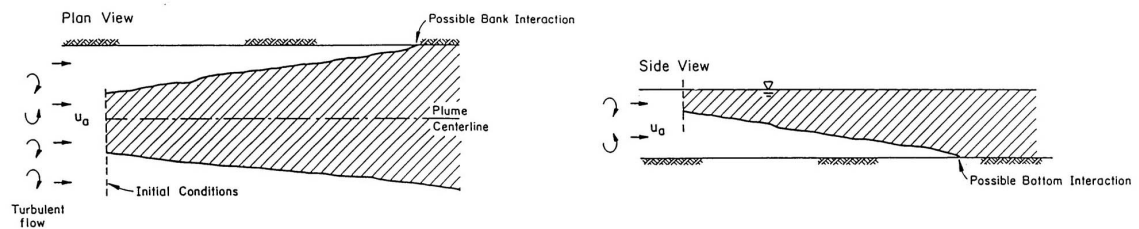


Figure 2-7: *Passive ambient diffusion processes with advection in the far-field region. [Jirka 1996].*

3 Governing equations

3.1 Introduction

Although the final interest may be in the distribution and transport of heat or dissolved mass, modelling the flow of the fluid cannot be omitted. This is because the transport of heat / dissolved mass is governed by the flow field. At the same time, the flow field is influenced by the distribution of temperature or dissolved mass.

Before a flow field can be modelled numerically, one needs a mathematical model of the physical processes involved. In this section a mathematical model is set up, starting with the principle of conservation and a set of assumptions.

The flow of the fluid (in this case water) is described by two conservation principles: (i) conservation of mass and (ii) conservation of momentum.

Also, some assumptions are made concerning the properties of the described fluid. The main ones are:

- Continuum
- Incompressibility ($\partial\rho/\partial p = 0$)
- Constant viscosity

More assumptions are made in this chapter. When used, it will be stated clearly in the text.

Furthermore two other physical processes need to be described, because they influence the properties of the fluid: transport of dissolved mass and transport of heat. These influences are taken into account in the momentum equations by means of the equation of state, which will also be given in this chapter.

In section 3.3 the treatment of turbulence will be described; in section 3.4 the equations are presented in the Cartesian co-ordinate system. In the last sections of this chapter some remarks will be made on the stability of density stratification (section 3.5), important dimensionless parameters (section 3.6) and subdividing the pressure (section 3.7).

3.2 The Equations

3.2.1 Conservation of mass

The principle of mass-conservation is represented by the continuity equation [Batchelor 1967, p. 74]:

$$\frac{\partial\rho}{\partial t} + \text{div}(\rho\mathbf{u}) = 0 \quad (3.1)$$

in which ρ is the fluid density [kg m^{-3}], t is the time [s] and $\mathbf{u}$ is the velocity vector [m s^{-1}]. The equation can also be written as

$$\frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \text{grad } \rho + \rho \text{div } \mathbf{u} = 0 \quad (3.2)$$

or

$$\frac{D\rho}{Dt} + \rho \text{div } \mathbf{u} = 0 \quad (3.3)$$

in which D/Dt is the substantial derivative operator, expressing a derivative following the motion of the fluid.

The substantial derivative operator is defined as

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + (\mathbf{u} \cdot \text{grad}). \quad (3.4)$$

3.2.2 Conservation of momentum

Under the assumption of compressible flow with constant viscosity [Batchelor 1967, p. 147 or Lamb 1974, p. 577, Art. 328], the principle of conservation of momentum is given by:

$$\rho \frac{D\mathbf{u}}{Dt} = \rho \mathbf{G} - \text{grad } p + \frac{1}{3} \mu \text{grad}(\text{div } \mathbf{u}) + \mu \nabla^2 \mathbf{u} \quad (3.5)$$

with p representing the fluid pressure [$\text{Pa} = \text{N m}^{-2}$], μ representing the dynamic viscosity (or briefly: viscosity) [Pa s] and $\mathbf{G}$ representing all the body forces per unit mass. Equation (3.5) is usually called the Navier-Stokes equation (of motion).

The quantity $\mathbf{G}$ can be written as [Batchelor 1967, p. 140]:

$$\mathbf{G} = \mathbf{g} + \Omega^2 \mathbf{r} - 2\boldsymbol{\Omega} \times \mathbf{u} \quad (3.6)$$

in which $\mathbf{g}$ is the gravitational acceleration vector [m s^{-2}], $\boldsymbol{\Omega}$ is the angular velocity vector [rad s^{-1}], Ω is the magnitude of the angular velocity [rad s^{-1}] and $\mathbf{r}$ is the position vector [m] normal to the southpole-northpole axis pointing to the position under consideration.

From now on we will assume that the Coriolis acceleration is negligible, i.e., $\mathbf{G} = \mathbf{g}$. This is justifiable because in the problems considered, $\Omega^2 \mathbf{r}$, as well as $2\boldsymbol{\Omega} \times \mathbf{u}$, are small compared to $\mathbf{g}$.

3.2.3 Boussinesq approximations

Before the Boussinesq approximations are introduced, it is useful to rewrite the conservation equations a bit.

Consider the Equilibrium State of static pressure in all directions. In this case all velocities and accelerations equal zero. The following relation defines the static pressure p_0 :

$$\rho_0 \mathbf{g} - \text{grad } p_0 = \mathbf{0}, \quad (3.7)$$

where ρ_0 is a constant reference density. Now the pressure and the density can be split up in two parts:

$$p = p_0 + \Delta p \quad (3.8)$$

$$\rho = \rho_0 + \Delta \rho \quad (3.9)$$

Substituting this into the momentum equation and neglecting the Coriolis acceleration (i.e., $\mathbf{G} = \mathbf{g}$) gives:

$$(\rho_0 + \Delta \rho) \frac{D\mathbf{u}}{Dt} = \rho_0 \mathbf{g} + \Delta \rho \mathbf{g} - \text{grad } p_0 - \text{grad } \Delta p + \frac{1}{3} \mu \text{grad}(\text{div } \mathbf{u}) + \mu \nabla^2 \mathbf{u} \quad (3.10)$$

Using relation (3.7) this can be shortened to:

$$(\rho_0 + \Delta \rho) \frac{D\mathbf{u}}{Dt} = \Delta \rho \mathbf{g} - \text{grad } \Delta p + \frac{1}{3} \mu \text{grad}(\text{div } \mathbf{u}) + \mu \nabla^2 \mathbf{u} \quad (3.11)$$

Rewriting the equation is useful to make clear what each term in the equation contributes to the flow. The static pressure relation causes no flow and is therefore subtracted.

After having rewritten the momentum equation, some approximations can be made.

In the first Boussinesq approximation the density variation $\Delta \rho$ in the momentum equation is assumed to be small compared to the reference density ρ_0 .

- 1st Boussinesq approximation: $(\rho_0 + \Delta \rho) \frac{D\mathbf{u}}{Dt} \longrightarrow \rho_0 \frac{D\mathbf{u}}{Dt}$

In words: it is assumed that the density differences play a role only in the external body force term. In the acceleration (local and advective) the density differences are neglected. After dividing by ρ_0 the momentum equation now reduces to:

$$\frac{D\mathbf{u}}{Dt} = \left(\frac{\Delta \rho}{\rho_0} \right) \mathbf{g} - \frac{1}{\rho_0} \text{grad } \Delta p + \frac{\mu}{3\rho_0} \text{grad}(\text{div } \mathbf{u}) + \frac{\mu}{\rho_0} \nabla^2 \mathbf{u} \quad (3.12)$$

The justification of this approximation lies in the fact that when $\Delta \rho / \rho_0$ is small, $\Delta \rho$ introduces only a small correction to the inertia term, whereas in the buoyancy term it is of primary importance. Neglecting $\Delta \rho$ here would lead to the omission of the entire body force term.

In the second Boussinesq approximation the substantial derivative of the density $D\rho/Dt$ in the continuity equation is assumed to be small compared to the other term.

- 2nd Boussinesq approximation: $\frac{D\rho}{Dt} + \rho \operatorname{div} \mathbf{u} = 0 \longrightarrow \operatorname{div} \mathbf{u} = 0$

With the second Boussinesq approximation the continuity equation is reduced to:

$$\operatorname{div} \mathbf{u} = 0 \quad (3.13)$$

The assumption that the term $D\rho/Dt$ is small is justified in Appendix A.

Substituting Equation (3.13) and the relation $\mu/\rho_0 = \nu$ into the momentum equation (3.12) gives

$$\frac{D\mathbf{u}}{Dt} = \left(\frac{\Delta\rho}{\rho_0} \right) \mathbf{g} - \frac{1}{\rho_0} \operatorname{grad} \Delta p + \nu \nabla^2 \mathbf{u}, \quad (3.14)$$

with ν representing the kinematic viscosity [$\text{m}^2 \text{s}^{-1}$].

For convenience the momentum equation is written also with the static pressure included:

$$\frac{D\mathbf{u}}{Dt} = \left(\frac{\rho}{\rho_0} \right) \mathbf{g} - \frac{1}{\rho_0} \operatorname{grad} p + \nu \nabla^2 \mathbf{u} \quad (3.15)$$

3.2.4 Conservation of energy

One of the transported quantities affecting the flow through the density is heat. Using the principle of conservation of energy and neglecting viscous dissipation, assuming incompressibility of the fluid and using the Boussinesq approximation, the following advection-diffusion equation for heat can be derived:

$$\frac{DT}{Dt} = \alpha \nabla^2 T \quad (3.16)$$

with thermal diffusivity $\alpha = \lambda/\rho_0 C_p$, where T is the temperature [K], λ is the thermal conductivity [$\text{W m}^{-1} \text{K}^{-1}$], ρ_0 is the reference density [kg m^{-3}] and C_p is the specific heat [$\text{J kg}^{-1} \text{K}^{-1}$].

A derivation of Equation (3.16) can be found in Appendix B. This transport equation for temperature is coupled to the momentum and continuity equations through the Equation of State (cf. section 3.2.5).

3.2.5 Equation of State

For an incompressible water body ($\partial\rho/\partial p = 0$), the density of the water is pressure-independent. However, the density of the fluid can still be dependent on other parameters. If only temperature plays a role in this ($\rho = \rho(T)$), the following equation of state can be used [AEA Technology 1999]:

$$\rho(T) = \rho_0 (1 - \beta(T - T_0)) \quad (3.17)$$

where β is the coefficient of thermal expansion [K^{-1}].

If we assume that the density depends on the temperature $[T]$ and the salinity $[s]$ ($\rho = \rho(s, T)$), an empirical relation for seawater given by Eckart [1958] can be used:

$$\rho = \frac{1000P_0}{\lambda + \alpha_0 P_0} \quad (3.18)$$

where:

$$\lambda = 1779.5 + 11.25T - 0.0745T^2 - (3.80 + 0.01T)s$$

$$\alpha_0 = 0.6980$$

$$P_0 = 5890 + 38T - 0.375T^2 + 3s$$

with the salinity s in [ppt], the water temperature T in [$^{\circ}\text{C}$] and the density ρ in [kg m^{-3}].

3.3 Turbulence treatment

Most free surface flows occurring in civil engineering practice have high Reynolds numbers ($Re > O(10^3)$; cf. section 3.6, Equation (3.47)), which makes the fluid motion turbulent. In the case of jets, this can easily be shown by assuming a characteristic velocity $U = 1.0 \text{ m s}^{-1}$, a characteristic length scale $L = 1.0 \text{ m}$ and a kinematic (molecular) viscosity of water $\nu = 1.0 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}$. Substituting this yields a Reynolds number of:

$$Re = \frac{UL}{\nu} = \frac{1.0 \text{ m s}^{-1} \cdot 1.0 \text{ m}}{1.0 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}} = 1.0 \cdot 10^6 \quad (3.19)$$

For low Reynolds numbers ($Re < O(10^3)$), the flow is laminar flow, i.e., the Navier-Stokes equations yield a stable solution. However in flows with a higher Reynolds number ($Re > O(10^3)$), the non-linear convective terms become so dominant with respect to the viscous terms that the solution becomes unstable. That is, infinitely small differences in boundary and initial conditions can yield significant differences in the solution.

Furthermore, although the equations presented in section 3.2 do govern turbulent flow, it is not feasible to solve these. Usually the grid is too coarse and the time step is too large to resolve the turbulent scales of motion. (The turbulent processes are ‘sub-grid’.) Because of this, the Navier-

Stokes equations are time-averaged. Time-averaged quantities are indicated by a bar over the symbol. For instance, the time-averaged (or time-smoothed) velocity is given by:

$$\bar{\mathbf{u}} = \frac{1}{t_0} \int_t^{t+t_0} \mathbf{u} dt \quad (3.20)$$

The instantaneous velocity $\mathbf{u}$ may then be written as the sum of the time-smoothed velocity $\bar{\mathbf{u}}$ and a velocity fluctuation $\mathbf{u}'$:

$$\mathbf{u} = \bar{\mathbf{u}} + \mathbf{u}' \quad (3.21)$$

Similar expressions can be written for the pressure p and the temperature T , which are also fluctuating:

$$p = \bar{p} + p' \quad (3.22)$$

$$T = \bar{T} + T' \quad (3.23)$$

Now, the equations that describe the time-smoothed velocity, pressure and temperature for an incompressible fluid can be derived. This is done by substituting Equations (3.21) to (3.23) into the continuity equation (3.13), the momentum equation (3.15) and the energy equation (3.16). The resulting equations are then time-averaged according to the method of Equation (3.20), yielding the so-called Reynolds averaged or Reynolds equations.

Continuity Equation (Reynolds averaged):

$$\text{div } \bar{\mathbf{u}} = 0 \quad (3.24)$$

Momentum equation (Reynolds averaged):

$$\frac{D\bar{\mathbf{u}}}{Dt} = \left(\frac{\rho}{\rho_0} \right) \mathbf{g} - \frac{1}{\rho_0} \text{grad } \bar{p} + \nu \nabla^2 \bar{\mathbf{u}} - \frac{1}{\rho_0} \text{div } \bar{\boldsymbol{\tau}}' \quad (3.25)$$

with the components of the quantity $\bar{\boldsymbol{\tau}}^{(t)}$, usually referred to as the Reynolds stresses [$\text{kg m}^{-1} \text{s}^{-2}$]:

$$\bar{\tau}_{xx}^{(t)} = \rho \overline{u'u'}; \quad \bar{\tau}_{xy}^{(t)} = \rho \overline{u'v'}; \quad \bar{\tau}_{xz}^{(t)} = \rho \overline{u'w'}; \quad \text{etc.} \quad (3.26)$$

Energy equation (Reynolds averaged):

$$\frac{D\bar{T}}{Dt} = \alpha \nabla^2 \bar{T} - \text{div } \bar{\mathbf{q}}^{(t)} \quad (3.27)$$

in which $\bar{\mathbf{q}}^{(t)}$ has the components:

$$\bar{q}_x^{(t)} = \overline{u'T'}; \quad \bar{q}_y^{(t)} = \overline{v'T'}; \quad \bar{q}_z^{(t)} = \overline{w'T'} \quad (3.28)$$

This way the additional quantities $\bar{\tau}^{(t)}$ and $\bar{\mathbf{q}}^{(t)}$ have been introduced. To solve the resulting system of equations, additional equations relating $\bar{\tau}^{(t)}$ and $\bar{\mathbf{q}}^{(t)}$ to the time-smoothed velocities and temperatures have to be found. Finding such expressions is called the ‘closure problem’. Such approaches are generally based on statistical methods, in which it is assumed that time-averages and spatial averages are equivalent to statistical ensemble averages. (This is the ergodic hypothesis.)

Usually, the eddy viscosity hypothesis is used to solve the closure problem. This hypothesis states that the Reynolds stresses (Equation (3.26)) can be linearly related to the mean velocity gradients in a manner analogous to the viscous term. Application yields the following expression for the Reynolds stresses [derived from Bird 1967, p. 88, by substituting the continuity equation (3.24)]:

$$\begin{aligned}\bar{\tau}_{xx}^{(t)} &= -2\mu^{(t)} \frac{\partial \bar{u}}{\partial x}, & \bar{\tau}_{xy}^{(t)} = \bar{\tau}_{yx}^{(t)} &= -\mu^{(t)} \left[\frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right] \\ \bar{\tau}_{yy}^{(t)} &= -2\mu^{(t)} \frac{\partial \bar{v}}{\partial y}, & \bar{\tau}_{yz}^{(t)} = \bar{\tau}_{zy}^{(t)} &= -\mu^{(t)} \left[\frac{\partial \bar{v}}{\partial z} + \frac{\partial \bar{w}}{\partial y} \right] \\ \bar{\tau}_{zz}^{(t)} &= -2\mu^{(t)} \frac{\partial \bar{w}}{\partial z}, & \bar{\tau}_{zx}^{(t)} = \bar{\tau}_{xz}^{(t)} &= -\mu^{(t)} \left[\frac{\partial \bar{w}}{\partial x} + \frac{\partial \bar{u}}{\partial z} \right]\end{aligned}\tag{3.29}$$

where $\mu^{(t)}$ is a turbulent coefficient of viscosity or eddy viscosity. Substituting these relations (3.29) into the Reynolds averaged momentum equation (3.25) and using the expression for the kinematic eddy viscosity $\nu^{(t)} = \mu^{(t)} / \rho_0$ yields:

$$\frac{D\bar{\mathbf{u}}}{Dt} = \left(\frac{\rho}{\rho_0} \right) \mathbf{g} - \frac{1}{\rho_0} \text{grad } \bar{p} + \text{div} \left(\nu^{(eff)} \text{grad } \bar{\mathbf{u}} \right)\tag{3.30}$$

with the effective viscosity $\nu^{(eff)} = \nu + \nu^{(t)}$.

Analogous to the eddy viscosity hypothesis, the eddy diffusivity hypothesis can be used to get an expression for the components of $\bar{\mathbf{q}}^{(t)}$:

$$\bar{\mathbf{q}}^{(t)} = -\alpha^{(t)} \text{grad } \bar{T}\tag{3.31}$$

in which $\alpha^{(t)}$ is the (thermal) eddy diffusivity [$\text{m}^2 \text{s}^{-1}$]. Substituting relation (3.31) into the Reynolds averaged energy equation (3.27) yields:

$$\frac{D\bar{T}}{Dt} = \text{div} \left(\alpha^{(eff)} \text{grad } \bar{T} \right)\tag{3.32}$$

with the effective thermal diffusivity $\alpha^{(eff)}$

$$\alpha^{(eff)} = \alpha + \alpha^{(t)}\tag{3.33}$$

The newly introduced eddy viscosity and eddy diffusivity are strongly dependent on the position in the flow. Several semi-empirical relations have been used to determine the eddy viscosity $\nu^{(t)}$ as a function of the local flow characteristics, e.g. the k - ϵ model, the Prandtl mixing length model.

In the k - ϵ model, the dynamic eddy viscosity $\nu^{(t)}$ is determined by linking it to the turbulent kinetic energy k [$\text{m}^2 \text{s}^{-2}$] and the turbulent energy dissipation rate ϵ [$\text{m}^2 \text{s}^{-3}$]. This is done with the following relation:

$$\nu^{(t)} = C_\mu \frac{k^2}{\epsilon} \quad (3.34)$$

where C_μ is an empirical constant. The turbulent kinetic energy k is determined by:

$$k = \frac{1}{2} \overline{\mathbf{u}'^2} \quad (3.35)$$

Both the energy dissipation rate and the turbulent kinetic energy are flow dependent quantities, which are calculated using a transport equation.

Eddy diffusivities are usually described by specifying a turbulent Prandtl number σ_t [-]

$$\sigma_t = \frac{\nu^{(t)}}{\alpha^{(t)}} \quad (3.36)$$

thereby linking the eddy diffusivity to the eddy viscosity. The Prandtl number is usually in the range of 0.5 to 1.0.

3.4 Equations in a Cartesian co-ordinate system

It can be convenient to write the governing equations in a Cartesian frame of reference. Consider a Cartesian co-ordinate system with x and y as horizontal co-ordinates, and z as vertical co-ordinate. (The z -axis is pointing opposite to the direction of the gravitational vector.)

$$\begin{aligned} \mathbf{x} &\hat{=} (x, y, z) \\ \mathbf{u} &\hat{=} (u, v, w) = \left(\frac{dx(t)}{dt}, \frac{dy(t)}{dt}, \frac{dz(t)}{dt} \right) \end{aligned} \quad (3.37)$$

From now on, the overbar used to represent the time smoothing of the parameters $\mathbf{u}$, p and T will be dropped. The Reynolds averaged equations governing turbulent flow can now be written as follows:

Continuity equation (Reynolds averaged; cf. Equation (3.24)):

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (3.38)$$

Momentum equation in the x-direction (Reynolds averaged; cf. Equation (3.30)):

$$\frac{Du}{Dt} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left(\mathbf{v}^{(eff)} \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mathbf{v}^{(eff)} \frac{\partial u}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mathbf{v}^{(eff)} \frac{\partial u}{\partial z} \right) \quad (3.39)$$

Momentum equation in the y-direction (Reynolds averaged; cf. Equation (3.30)):

$$\frac{Dv}{Dt} = -\frac{1}{\rho_0} \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left(\mathbf{v}^{(eff)} \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mathbf{v}^{(eff)} \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mathbf{v}^{(eff)} \frac{\partial v}{\partial z} \right) \quad (3.40)$$

Momentum equation in the z-direction (Reynolds averaged; cf. Equation (3.30)):

$$\frac{Dw}{Dt} = -\left(\frac{\rho}{\rho_0} \right) g - \frac{1}{\rho_0} \frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left(\mathbf{v}^{(eff)} \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left(\mathbf{v}^{(eff)} \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left(\mathbf{v}^{(eff)} \frac{\partial w}{\partial z} \right) \quad (3.41)$$

Transport equation for temperature (Reynolds averaged; cf. Equation (3.32)):

$$\frac{DT}{Dt} = \frac{\partial}{\partial x} \left(\alpha^{(eff)} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\alpha^{(eff)} \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(\alpha^{(eff)} \frac{\partial T}{\partial z} \right)$$

For all equations it holds that:

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} + w \frac{\partial}{\partial z} \quad (3.42)$$

3.5 Equilibrium and stability of stratification

Some notes will be made on the stability of stratification and the equilibrium situation to which a fluid tends to go to.

A stratified system is a system where the density varies in the vertical (smoothly or discontinuously), but is constant in horizontal planes. In a no flow situation this is an equilibrium state. The expression for hydrostatic pressure

$$p_h(z) = p_{atm} + g \int_z^{\zeta} \rho dz' \quad (3.43)$$

shows that the fluid is in rest only when the pressure as well as the density is constant in every horizontal plane. (In Equation (3.43), ζ represents the water level elevation [m].) This equilibrium stratification is stable when a heavier fluid lies below a lighter fluid. When the heavier fluid lies above the lighter fluid, the system is unstable (cf. Figure 3-1).

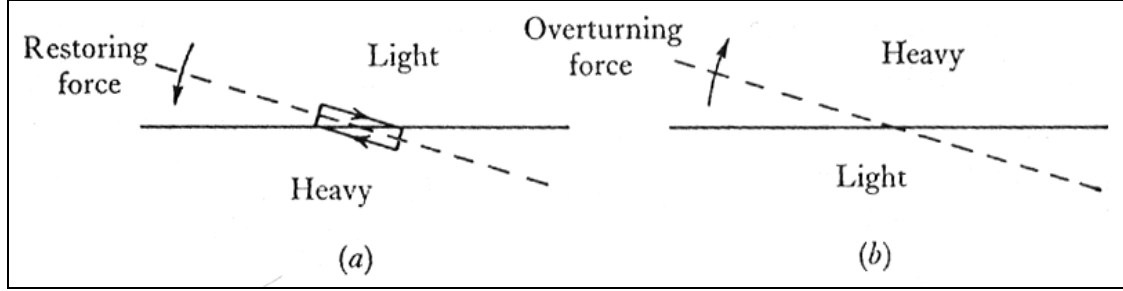


Figure 3-1: Displacements from hydrostatic equilibrium: (a) stable, and (b) unstable density distributions. [Turner 1973].

3.6 Important (dimensionless) parameters

An important overall parameter describing the entire flow is the overall or finite difference form of the Richardson number. When the Boussinesq and hydrostatic approximations are made and the motion is steady, the ratio of the buoyancy to the inertia terms is the only dimensionless number needed to specify an inviscid flow. Using the scales of velocity U and length L imposed by the boundary conditions, this can be written as

$$Ri_o = g' L / U^2 \quad (3.44)$$

with acceleration due to buoyancy $g' = g(\rho_a - \rho_o) / \rho_a$.

In hydraulic engineering it is more common to use the inverse square root of Ri_o :

$$F = U / \sqrt{g' L} \quad (3.45)$$

which is called the internal or densimetric Froude number.

By specifying the parameters used to represent the velocity and length scales in Equation (3.45), a common jet parameter (F_o) is obtained. The densimetric Froude number of the outfall (F_o) is given by

$$F_o = \frac{u_o}{\sqrt{\left(\frac{\Delta \rho_o}{\rho_a} g h_0 \right)}} \quad (3.46)$$

where u_o is the velocity of the effluent at the outfall, $\Delta \rho_o$ is the density difference at the outfall, ρ_a is the ambient density and h_0 is the reference dimension of the outfall. Usually the height of the outfall or the square root of the outfall port area is taken [Miller & Brighthouse 1984, p. 5].

When other physical effects like viscosity are taken into account, other parameters can be important. One of those is the Reynolds number,

$$Re = UL / \nu \quad (3.47)$$

where ν is the kinematic (molecular) viscosity. This dimensionless number represents a ratio of inertial and viscous terms in the momentum equations.

3.7 Subdividing the pressure

The purpose of this section is to give an overview of the various kinds of pressure used in this report. All pressures in this section have the unit [Pa = N m⁻²].

The total fluid pressure can be split in two parts: a hydrostatic part p_h and a hydrodynamic part q :

$$p = p_h + q \quad (3.48)$$

- The hydrodynamic (or non-hydrostatic) pressure q is the contribution to the total pressure due to acceleration of the fluid.
- The hydrostatic pressure is the part of the pressure that is independent of the acceleration of the fluid. The parameters determining this pressure are the atmospheric pressure p_{atm} , the water level ζ [m], the acceleration of gravity g [m s⁻²] and the density ρ [kg m⁻³] in the water column above the considered location. The hydrostatic pressure can be expressed as:

$$p_h(z) = p_{atm} + g \int_z^{\zeta} \rho dz' \quad (3.49)$$

See Appendix F.2 for more information about the derivation of this.

The hydrostatic pressure on its turn can be subdivided in four parts: the atmospheric pressure p_{atm} , the static pressure p_0 , the barotropic pressure p_{btr} and the baroclinic pressure p_{bcl} :

$$p_h = p_{atm} + p_0 + p_{btr} + p_{bcl} \quad (3.50)$$

- The atmospheric pressure p_{atm} represents the pressure of the atmosphere on the water surface. Because, at the surface, all other components of the total pressure are zero, the following holds:

$$p(x, \zeta, t) = p_{atm} \quad (3.51)$$

The atmospheric pressure is assumed to be constant. Because of the assumed incompressibility of the fluid, only gradients in the pressure govern the flow, not the magnitude of the pressure. This means that an arbitrary value can be taken for the atmospheric pressure (e.g. $p_{atm} = 0$). Atmospheric pressure is assumed to be constant.

- The static pressure p_0 is the pressure when the velocity and the acceleration of the fluid is zero. This static situation is defined more precisely in section 3.2.3.

$$p_0 = p_{atm} + \rho_0 g (\zeta_0 - z) \quad (3.52)$$

- The barotropic pressure p_{btr} is the component of the pressure due to a difference in water level compared to the static situation. The relation

$$p_{btr} = \rho_0 g (\zeta - \zeta_0) \quad (3.53)$$

in which ζ_0 is the water level [m] in the static situation represents this.

- The baroclinic pressure p_{bcl} is the component of the pressure due to variations in the density compared to the reference density ρ_0 :

$$p_{bcl} = g \int_z^\zeta (\rho - \rho_0) dz' \quad (3.54)$$

Sometimes the modified pressure is used in literature and software (e.g. CFX). This is the total pressure minus the static pressure (assuming no contribution of the atmospheric pressure):

$$p_{\text{mod}} = p - \rho_0 g (\zeta_0 - z) \quad (3.55)$$

In other words, the modified pressure is the sum of the dynamic pressure, the barotropic pressure and de baroclinic pressure.

4 Flow modelling software

4.1 Delft3D-FLOW

4.1.1 Introduction

Delft3D is a software package for 2D or 3D computations for coastal, river and estuarine areas. It can carry out simulations of free surface flow, sediment transport, waves, water quality, morphological developments and ecology. The Delft3D package consists of several modules, grouped around a common interface. Delft3D-FLOW is one of those modules.

The FLOW module is a multidimensional (2D-depth-averaged or 3D) hydrodynamic simulation program that calculates unsteady flow and transport phenomena with a free surface.

It aims at modelling flow phenomena of which the horizontal length scales are significantly larger than the vertical length scales.

In this section, some information about the governing equations and the assumptions used will be given. For a more extensive description of Delft3D-FLOW, it is referred to the ‘Delft3D-FLOW user manual’ [WL | delft hydraulics 1999].

4.1.2 Equations

In Delft3D-FLOW, the 2D (depth-averaged) or 3D non-linear shallow water equations are solved, which can be derived from the momentum equations (3.39) to (3.41) presented in section 3.4. The equations in section 3.4 were derived from the three-dimensional Navier-Stokes equations. Also a continuity equation, similar to the one presented in section 3.4 (Equation (3.38)), is solved in Delft3D. Some additional assumptions and approximations are necessary to come to the Delft3D equations:

- The Coriolis term that was neglected in chapter 3 is used in the Delft3D equations, however in a modified form (e.g. the term $\Omega^2 \mathbf{r}$ in Equation (3.6) is neglected).
- The hydrostatic pressure assumption is used (cf. section 4.1.3).
- Source terms are added in the continuity and momentum equations.
- The terms involving $\partial \bar{w} / \partial x$ and $\partial \bar{w} / \partial y$ in the Reynolds stresses (Equation (3.29)) are neglected.

The Delft3D equations can be solved on a finite difference grid using the following horizontal co-ordinates:

- Cartesian rectangular (x, y)
- Orthogonal curvilinear (ξ, η)
- Spherical (λ, ϕ)

In the vertical, sigma co-ordinates are used. This means that the vertical grid moves with the bottom and free surface using a fixed number of layers.

The sigma transformation starting from a Cartesian frame of reference with horizontal dimensions x [m] and y [m] and vertical dimension z [m], is expressed by:

$$\begin{aligned} t &= t' \\ x &= x' \\ y &= y' \\ z &= \sigma H(x, y, t) + \zeta(x, y, t) \Rightarrow \sigma = \frac{z - \zeta(x, y, t)}{H(x, y, t)} \end{aligned} \quad (4.1)$$

with:

$$\begin{aligned} H(x, y, t) &= \zeta(x, y, t) + d(x, y, t) \\ -d(x, y) &\leq z \leq \zeta(x, y, t) \Rightarrow -1 \leq \sigma \leq 0 \end{aligned}$$

in which t is the time [s], σ is the transformed vertical co-ordinate [-], H is the total water depth [m], ζ is the water level elevation [m] above a horizontal plane of reference and d is the water depth [m] below a horizontal plane of reference.

The shallow water equations based on a Cartesian horizontal grid and a sigma grid in the vertical are presented below showing respectively the horizontal x -direction (4.2), the horizontal y -direction (4.3) and the vertical σ -direction (4.4). Equation (4.5) represents the continuity equation.

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + \frac{\omega}{H} \frac{\partial u}{\partial \sigma} - f v = -\frac{1}{\rho_0} P_x + F_x + \frac{1}{H^2} \frac{\partial}{\partial \sigma} \left(\nu_v \frac{\partial u}{\partial \sigma} \right) + M_x \quad (4.2)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + \frac{\omega}{H} \frac{\partial v}{\partial \sigma} + f u = -\frac{1}{\rho_0} P_y + F_y + \frac{1}{H^2} \frac{\partial}{\partial \sigma} \left(\nu_v \frac{\partial v}{\partial \sigma} \right) + M_y \quad (4.3)$$

$$\frac{\partial p}{\partial \sigma} = -\rho g H \quad (4.4)$$

$$\frac{\partial \zeta}{\partial t} + \frac{\partial (Hu)}{\partial x} + \frac{\partial (Hv)}{\partial y} + \frac{\partial \omega}{\partial \sigma} = Q \quad (4.5)$$

In the above equations, ω is the vertical velocity [m s^{-1}] relative to the sigma grid (cf. Appendix E), f is the Coriolis parameter, ν_v is the vertical eddy viscosity [$\text{m}^2 \text{s}^{-1}$], P_x and P_y represent the pressure gradient terms, F_x and F_y represent the unbalance of horizontal Reynolds stresses, M_x and M_y take into account the contributions due to external sources or sinks of momentum and Q represents the contributions due to the discharge or withdrawal of water (mass).

These equations formulated in the more general orthogonal curvilinear co-ordinates can be found in the user manual of Delft3D-FLOW [WL | delft hydraulics 1999].

Pressure gradient

The only terms where density differences are taken into account are the pressure gradient terms P_x and P_y , which can be expressed as:

$$\frac{1}{\rho_0} P_x = g \frac{\partial \zeta}{\partial x} + \frac{gH}{\rho_0} \int_{\sigma}^0 \left(\frac{\partial \rho}{\partial x} + \frac{\partial \sigma}{\partial x} \frac{\partial \rho}{\partial \sigma} \right) d\sigma' \quad (4.6)$$

$$\frac{1}{\rho_0} P_y = g \frac{\partial \zeta}{\partial y} + \frac{gH}{\rho_0} \int_{\sigma}^0 \left(\frac{\partial \rho}{\partial y} + \frac{\partial \sigma}{\partial y} \frac{\partial \rho}{\partial \sigma} \right) d\sigma' \quad (4.7)$$

Sinks and sources of mass and momentum

In addition to discharging water at the computational boundaries, Delft3D-FLOW has also an option to add a source or sink in the simulation area. This is achieved by adding a mass source term Q in the continuity equation. This term can be written as:

$$Q = H(q_{in} - q_{out}) \quad (4.8)$$

in which q_{in} and q_{out} are respectively the local sources and sinks of water per unit of volume [s^{-1}]. At the surface the effects of precipitation and evaporation can be taken into account.

When the discharged water also gives rise to a flux of momentum, additional terms are added in the horizontal momentum equations:

$$M_x = q_{in}(\hat{U} - u) \quad (4.9)$$

$$M_y = q_{in}(\hat{V} - v) \quad (4.10)$$

with $\hat{U}$ and $\hat{V}$ being the velocity components in x- and y-direction of the discharged water.

4.1.3 Hydrostatic pressure assumption

More attention will be given to the assumed hydrostaticity of the flow, because of the importance of this for this subject.

Dunsbergen [1994] distinguishes three characteristic relations important in the justification of the shallow water assumption:

- $H/L \ll 1$ The characteristic horizontal length scale is much larger than the water depth (H).

- $W = O(HU/L)$ The characteristic vertical velocity component is small in comparison with the characteristic horizontal velocity component.
- $|\tau|/\rho gH \ll 1$ The characteristic shear stress $|\tau|$ introduced by the main driving force of the fluid flow is small in comparison with the characteristic hydraulic pressure.

If these conditions are fulfilled, the momentum equation in the direction of the gravitational vector (z - or σ -direction) reduces to the hydrostatic pressure relation. This is represented in the following relation.

$$\frac{Dw}{Dt} = -\left(\frac{\rho}{\rho_0}\right)g - \frac{1}{\rho_0} \frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left(v^{(eff)} \frac{\partial w}{\partial x} \right) + \frac{\partial}{\partial y} \left(v^{(eff)} \frac{\partial w}{\partial y} \right) + \frac{\partial}{\partial z} \left(v^{(eff)} \frac{\partial w}{\partial z} \right) \longrightarrow \frac{\partial p}{\partial z} = -\rho g \quad (4.11)$$

The resulting equations are the so-called Shallow Water Equations. The reduction in (4.11) does not mean that the term Dw/Dt and the viscous terms are taken equal to zero. This is the case only in a no flow situation. What can be said of both terms is that they are not used for determining the pressure.

Strictly speaking, conservation of momentum is not accounted for in the z -direction. Therefore, Dw/Dt is not influenced by the driving forces in the z -direction, only by the condition of continuity.

If the hydrostatic pressure relation (the second part of Equation (4.11)) is transformed to sigma co-ordinates, the Delft3D equation (4.4) is obtained.

4.1.4 Turbulence modelling

The terms F_x and F_y in respectively Equation (4.2) and Equation (4.3) represent the unbalance of horizontal Reynolds stresses. They can be expressed by:

$$F_x = \frac{\partial}{\partial x} \left[2v_H^{(eff)} \left(\frac{\partial u}{\partial x} + \frac{\partial u}{\partial \sigma} \frac{\partial \sigma}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[v_H^{(eff)} \left(\frac{\partial u}{\partial x} + \frac{\partial u}{\partial \sigma} \frac{\partial \sigma}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial v}{\partial \sigma} \frac{\partial \sigma}{\partial y} \right) \right] \quad (4.12)$$

$$F_y = \frac{\partial}{\partial x} \left[v_H^{(eff)} \left(\frac{\partial u}{\partial x} + \frac{\partial u}{\partial \sigma} \frac{\partial \sigma}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial v}{\partial \sigma} \frac{\partial \sigma}{\partial y} \right) \right] + \frac{\partial}{\partial y} \left[2v_H^{(eff)} \left(\frac{\partial v}{\partial y} + \frac{\partial v}{\partial \sigma} \frac{\partial \sigma}{\partial y} \right) \right] \quad (4.13)$$

in which $v_H^{(eff)}$ is the effective horizontal viscosity, which is assumed to be a superposition of three parts: a part due to '2D-turbulence', a part due to 3D-turbulence and a part due to molecular viscosity:

$$v_H^{(eff)} = v^{(mol)} + \max \left(v_H^{(t,2D)}, v^{(t,3D)} \right) \quad (4.14)$$

The molecular viscosity is incorporated in the code of Delft3D ($\nu^{(mol)} = 1.0 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}$), the ‘2D’-viscosity $\nu_H^{(t,2D)}$ can be specified by the user (as a constant) and a user-specified turbulence model computes the 3D part. For a 3D simulation, four types of turbulence closure models are available to determine the ‘3D’ eddy viscosity $\nu^{(t,3D)}$:

- Constant eddy (turbulent) viscosity
- Algebraic Eddy viscosity closure Model (AEM)
- k - L turbulence closure model
- k -epsilon (k - ϵ) turbulence closure model (cf. section 3.3)

For 3D shallow water flow, the difference between the characteristic horizontal and vertical length scales gives rise to an anisotropic eddy viscosity. Therefore the vertical (effective) eddy viscosity $\nu_V^{(eff)}$ is defined differently from the horizontal viscosity:

$$\nu_V^{(eff)} = \max\left(\nu^{(t,B)}, \nu^{(mol)} + \nu^{(t,3D)}\right) \quad (4.15)$$

The molecular viscosity and ‘3D’ eddy viscosity are isotropic. The variable $\nu^{(t,B)}$ represents a background value for the eddy viscosity, which the user can specify as a constant.

The expressions for the terms F_x (Equation (4.12)) and F_y (Equation (4.13)) can be derived from Equation (3.25) presented in chapter 3, by neglecting the terms involving $\partial \bar{w} / \partial x$ and $\partial \bar{w} / \partial y$ in the relation that defines the Reynolds stresses (Equation (3.29)). Also a sigma transformation is applied to these terms. (See section 3.3 for general information about turbulence treatment.)

4.1.5 Boundary conditions

Open boundaries

Open boundaries are almost always necessary to restrict the computational area. These boundaries exist only in the mathematical model and have no physical meaning, but without open boundaries, large areas without interest would also have to be modelled.

Because the incoming characteristics at an open boundary are never exactly correct, the outgoing characteristics will reflect and propagate as a disturbance into the simulation area. To reduce those reflections, so-called weakly reflecting boundary conditions may be applied.

The open boundaries available in Delft3D-FLOW are:

- Water level boundary
- Velocity boundary
- Discharge boundary
- Riemann invariant

The first three boundaries can be made weakly reflecting by specifying a reflection coefficient α . The Riemann invariant type boundary condition is already non-reflective for waves.

For the velocity, discharge and Riemann invariant type of boundary condition, the flow is assumed to be perpendicular to the open boundary. Other configurations are not (yet) possible.

Kinematic boundary conditions

The kinematic boundary conditions at the bottom ($z = -d$) and the free surface state that there is no flow through these boundaries. This is denoted as:

$$\omega|_{z=-d} = 0, \quad \omega|_{z=\zeta} = 0 \quad (4.16)$$

where ω is the vertical velocity relative to the σ -co-ordinate system. Because the σ -isolines move with the free surface and bottom, a zero ω -velocity means that there is no flow through these boundaries. These boundaries are implemented in the model as default and do not have to be prescribed when building a model.

Turbulence boundaries

Boundary conditions for the transport equations used in the selected turbulence model cannot be prescribed. The underlying assumption is that the turbulence will be generated inside the computational domain.

4.2 CFX

4.2.1 Introduction

CFX is a software package for the prediction of laminar and turbulent flow and heat transfer. It also contains additional models for multi-phase flows, combustion and particle transport. Both steady state and transient problems can be solved. Besides the information in this section, more information can be found in ‘Manual CFX-4.4’ [AEA Technology 1999].

CFX is built up in three parts:

- The front-end module (pre-processor)
Here, the problem is specified in a single data file, using a set of commands and keywords. This includes geometry and grid generation. The module translates the command file to a form designed for execution. At the same time an error check is performed.
- The solution module (solver)
This solves the discretised representation of the problem, using a finite volume approach.
- The graphics module (post-processor)
In this module, graphical output can be made using the data output files written by the solution module at specified times.

The equations used are the Navier-Stokes equation, a continuity equation and an equation for energy. To be able to simplify the equations where this is appropriate, various approximations can be made such as the Boussinesq approximation and the assumption of incompressible flow.

For solving the system of equations a finite volume method is used, with the possibility of choosing between various iteration methods.

The stopping mechanism (convergence criterion)

To end the iterations when a sufficiently accurate solution is reached, a convergence criterion is needed. The default criterion in CFX is a test on the *mass source residual* to see if it has fallen below a tolerance set in the command file. The mass source residual [kg s^{-1}] is the absolute value of the summation of mass fluxes through all cell faces.

A guideline for a sufficiently low threshold is 0.01 - 0.1 % of the total in going mass flux in the computational domain.

4.2.2 Buoyancy model

Buoyancy can be included in CFX in two different ways. The first option is to consider the flow to be compressible. However, for free surface problems, this is not efficient and necessary. The second option is to assume incompressible flow and to use the Boussinesq approximation (cf. section 3.2.3). The Boussinesq approximation results in a constant density ρ_0 in the momentum equations, except in the body force (buoyancy) term (the first right hand term in Equation (3.14)). In this term, if $\rho = \rho(T)$, the density is represented by [AEA Technology 1999]:

$$\rho(T) = \rho_0 (1 - \beta(T - T_0)) \quad (4.17)$$

where β is the coefficient of thermal expansion [K^{-1}] and T_0 is the buoyancy reference temperature.

4.2.3 Turbulence modelling

When the flow is turbulent, CFX solves the Reynolds averaged equations (cf. section 3.3). The averaging gives rise to the quantities $\nu^{(eff)}$ and $\alpha^{(eff)}$, respectively the effective viscosity and the effective diffusivity.

If the Boussinesq approximation is used in CFX, the (isotropic) effective viscosity μ_{eff} is defined by:

$$\nu^{(eff)} = \nu + \nu^{(t)} \quad (4.18)$$

where ν is the molecular kinematic viscosity and $\nu^{(t)}$ is the turbulent eddy viscosity.

The effective (isotropic) diffusivity $\alpha^{(eff)}$ is then defined by:

$$\alpha^{(eff)} = \alpha + \alpha^{(t)} \quad (4.19)$$

where α is the molecular kinematic diffusivity and $\alpha^{(t)}$ is the turbulent eddy diffusivity.

The default turbulence closure model (to determine the turbulent viscosity as a function of the flow characteristics) is the k - ϵ model (cf. section 3.3).

A user specified Prandtl number $\sigma_t = \nu^{(t)} / \alpha^{(t)}$ (default value: $\sigma_t = 1.0$) couples the turbulent diffusivity to the turbulent viscosity.

4.2.4 Boundary conditions

Various types of boundary conditions can be used in CFX. An overview with a short description is given in this section.

- **Inlet boundary:**
An inlet boundary in CFX is a boundary where the values of variables are specified. This is known as a Dirichlet boundary condition. In most cases the specified variables are the velocity and the temperature, but turbulence quantities can also be prescribed at inlets.
- **Outlet boundary:**
At an outlet boundary or mass flow boundary, Neumann boundary conditions are imposed on all transported variables. This means that the normal components of their gradients are specified, not their values. All transported quantities are given a zero gradient normal to the boundary, except for the velocity, which is given a constant gradient. Because of global mass continuity (in incompressible flow) the total outflow of mass is imposed to be equal to the sum of the flow of mass through the inlet boundaries.
- **Pressure boundary:**
At a pressure boundary the values of the modified pressure are specified at the boundary (Dirichlet boundary condition). The modified pressure is the pressure with the static part removed. Zero gradients normal to the boundary are imposed on the velocity and other transported quantities (Neumann boundary conditions).
- **Wall boundary:**
At wall boundaries a zero velocity through the boundary is imposed. Wall boundary conditions allow tangential velocity or tangential shear stress (or a combination of both) to be specified.

- Symmetry boundary:
At the symmetry plane (boundary) all variables are mathematically symmetric, thus allowing no advection and diffusion across the boundary, except the component of velocity normal to the boundary and the Reynolds shear stress and Reynolds flux involving the normal velocity which are all anti-symmetric.

4.2.5 Methods to improve convergence rate

If a computation shows a poor convergence rate or even diverges, some modification in the solution strategy can be made. The methods used are explained in this section.

Under-relaxation factors

Using an under-relaxation factor, the amount by which a variable would change if its transport equation were solved iteratively is reduced. For every variable a different factor can be specified.

The under-relaxation factor (URF) has a range of $0 < \text{URF} \leq 1$. The smaller the URF, the more under-relaxation is employed. The working of the factor is shown for a variable u :

$$u^{n+1} = u^n + \text{URF} \cdot \Delta u \quad (4.20)$$

Default values depend on the solution algorithm that is used. A value of 1 means that no under-relaxation is applied.

False time step factors

An alternative to setting under-relaxation factors is using false time steps. When using this method, each iteration can be seen as a time step. Non-stationary terms ($\partial/\partial t$) in the equations are included even if only a steady-state solution is required. When the steady state solution is obtained these terms will approach zero.

A drawback of this method is that some knowledge is required of the time scales involved.

4.3 CORMIX

4.3.1 Introduction

The Cornell Mixing Zone Expert System is a software system for the analysis and prediction of discharges into diverse water bodies. It was developed by the U.S. Environmental Protection Agency (U.S. EPA) and Cornell University. A flow classification system is at the heart of CORMIX. It provides an expert knowledge database that distinguishes among the many hydrodynamic flow patterns that a jet may exhibit. As examples, it includes jets attaching to the bottom, jets vertically mixing due to instabilities in shallow water, jets becoming trapped

internally due to density stratification, and jets intruding upstream against the ambient current due to buoyancy, and many others. Theoretically based hydrodynamic criteria using length scale analysis and empirical knowledge from laboratory and field experimentation, are applied in a systematic fashion to identify the most appropriate flow classification for a particular analysis situation.

4.3.2 Length scales

In case of buoyant surface jets discharging into unstratified water CORMIX distinguishes the following length scales.

- Jet/plume transition length scale L_M :

$$L_M = M_o^{3/4} / J_o^{1/2} \quad (4.21)$$

in which M_o is the kinematic momentum flux [$\text{m}^4 \text{s}^{-2}$] and J_o is the kinematic buoyancy flux [$\text{m}^2 \text{s}^{-3}$]. More information on these quantities is given at the end of this subsection.

Interpretation: The extent of the initial jet region before an unsteady surface spreading motion takes over from strong mixing. At the start the jet mixing is characterised by the initial momentum, later this changes to buoyancy induced lateral spreading.

- Jet/cross flow length scale L_m :

$$L_m = M_o^{1/2} / u_a \quad (4.22)$$

in which M_o is the kinematic momentum flux [$\text{m}^4 \text{s}^{-2}$] and u_a is the ambient velocity [m s^{-1}].

Interpretation: The distance over which a discharging jet intrudes into the ambient cross flow before it gets strongly deflected. At start only the initial momentum plays a role in establishing the jet trajectory. Later on, the flow regime is only influenced by the ambient velocity.

- Plume/cross flow length scale:

$$L_b = J_o / u_a^3 \quad (4.23)$$

in which J_o is the kinematic buoyancy flux [$\text{m}^2 \text{s}^{-3}$] and u_a is the ambient velocity [m s^{-1}].

Interpretation: A measure of the tendency for upstream intrusion for a strongly buoyant discharge.

Notes on the length scales:

$M_o = U_o Q_o$	Kinematic momentum flux [$\text{m}^4 \text{s}^{-2}$]
$J_o = g'_o U_o$	Kinematic buoyancy flux [$\text{m}^2 \text{s}^{-3}$]
$Q_o = U_o a_o$	Source discharge volume flux [$\text{m}^3 \text{s}^{-1}$]

$g'_o = g(\rho_a - \rho_o)/\rho_a$	Discharge buoyancy or acceleration due to buoyancy at the outfall [m s^{-2}]
a_o	Outfall port area [m^2]
u_a	Ambient velocity [m s^{-1}]
U_o	Discharge velocity [m s^{-1}]
ρ_o	Fluid density at the outfall
ρ_a	Ambient fluid density

4.3.3 Notes on the use of CORMIX

The basic idea of the simulation methodology is to make a sequence of relatively simple simulation modules which, when executed together, predict the trajectory and dilution characteristics of a complex flow. Each of the simulations uses the final values of the previous module as initial conditions. This way, the near-field as well as the far-field are modelled.

CORMIX can be used only to predict steady state solutions. Irregular time varying discharge situations pose a problem to the use of CORMIX, because in this situation steady state solutions do not exist. Problems can also arise when the surrounding bathymetry and flow patterns are difficult to schematise with the few parameters that CORMIX uses (e.g. estuaries).

More detailed information about CORMIX can be found in the CORMIX user manual [Jirka 1996].

5 Test case Point Beach Power Station

5.1 Introduction

To investigate the problems and difficulties that might be involved in the hydrostatic simulation of buoyant discharges a test case is simulated in Delft3D-FLOW (hydrostatic) and CFX (non-hydrostatic). This makes it possible to see what effects can be attributed to the hydrostaticity of the simulation. The results of the comparison can be found in section 5.3.

In section 5.4, the sensitivity of the simulated jet to the location of the open boundary, the initial direction of the discharge and the discharge method will be investigated.

In section 5.5, a comparison of the temperature distribution around the jet centreline will be made.

5.2 Outline of the case

5.2.1 General outline

From Miller and Brighthouse [1984, p. 123] a case is taken concerning a surface cooling water discharge in Lake Michigan. This case, the Point Beach Power Station, is taken because it has been used before in different simulation packages. Apart from the field data, results of analytical predictions made by Miller and Brighthouse are also available.

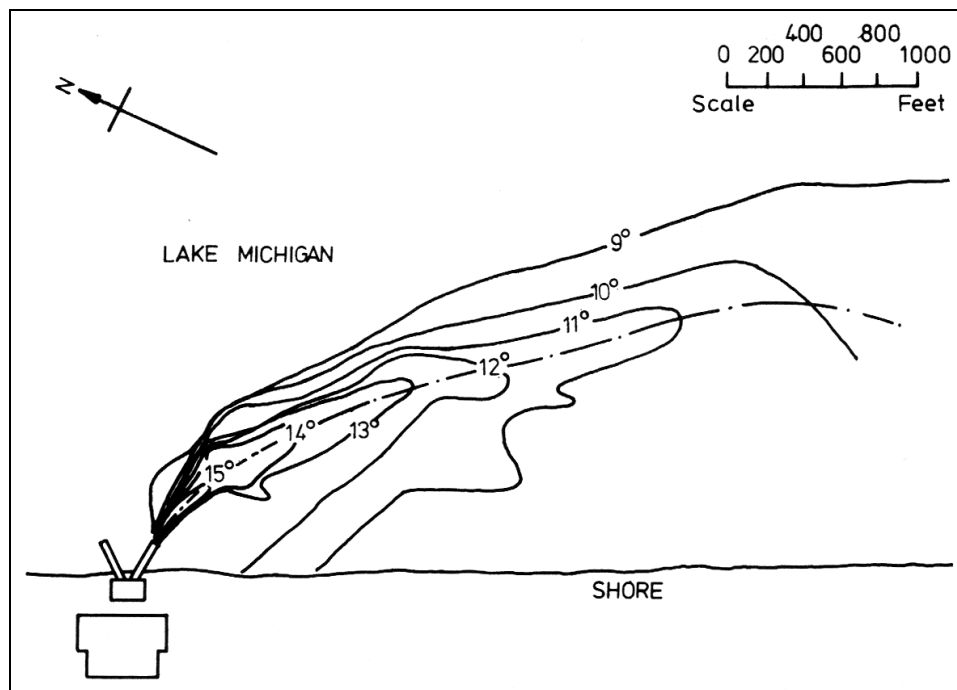


Figure 5-1: *Buoyant surface jet at the Point Beach Power Station, May 20, 1971.*

Point Beach Power Station on Lake Michigan has two discharge units, but at the time considered (May 20, 1971) only one was operating. The warmer cooling water is discharged into the shallow ambient water at an angle of 60° to the coastline. The ambient water has a small but significant current. Figure 5-1 shows the outline of the case together with the measurement data. These are limited to temperature, so the actual plume dynamics are not known.

The known parameters of the discharge situation are:

- Discharge $Q_o = 25.1 \text{ m}^3 \text{ s}^{-1}$
- Discharge velocity $u_o = 0.57 \text{ m s}^{-1}$
- Height of outfall $h_o = 4.2 \text{ m}$
- Width of outfall $b_o = 10.7 \text{ m}$
- Ambient temperature $T_a = 8.2^\circ\text{C}$
- Discharge temperature $T_o = 17.5^\circ\text{C}$
- Ambient current velocity $u_a = 0.09 \text{ m s}^{-1}$

There are no detailed data available concerning the exact bathymetry in the discharge area. In previous computations, the height of the outfall (4.2 m) is taken as the (uniform) ambient water depth. This is done here as well.

The densimetric Froude number of the outfall is:

$$F_o = \frac{u_o}{\sqrt{((\Delta\rho_o/\rho_a)gh_o)}} = \frac{0.57}{\sqrt{0.0016 \times 9.81 \times 4.2}} = 2.61$$

5.2.2 The grid

To model the discharge in Lake Michigan, a grid is made to represent the receiving water body. To be able to make a fair comparison between Delft3D-FLOW and CFX, the same grid is used for the two computational models.

As the coastline is straight in the area of discharge a rectilinear, horizontal grid is made with the following characteristics:

- Number of grid points in longshore direction 90 (m-direction)
- Number of grid points in cross-shore direction 35 (n-direction)
- Expansion factor 1.14
- Smallest cell $\Delta m \times \Delta n = 5.24 \text{ m} \times 5.98 \text{ m}$
- Largest cell $\Delta m \times \Delta n = 265.21 \text{ m} \times 246.28 \text{ m}$
- Total grid size $5458.31 \text{ m} \times 3170.14 \text{ m}$ (m $\times$ n)

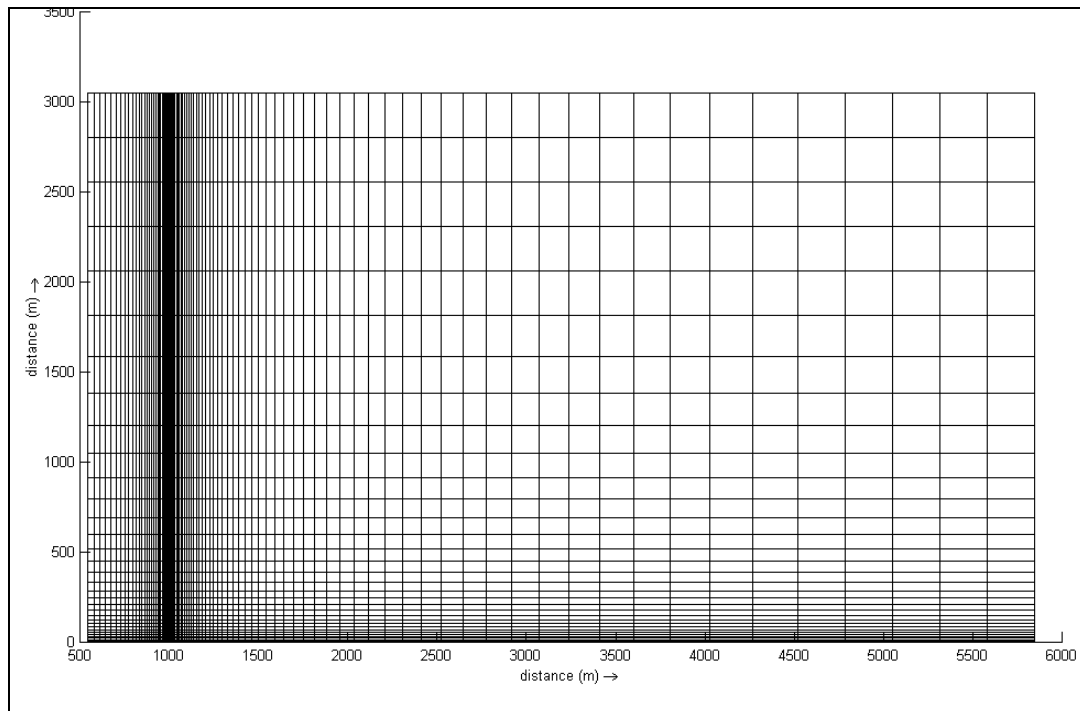


Figure 5-2: *Horizontal hydrodynamic grid used in CFX and Delft3D-FLOW computations. (Horizontal: m-direction; Vertical: n-direction.)*

To save computational time, the grid is taken non-uniform with the finest cells around the outfall. From the place of the outfall the grid widens with an expansion factor of 1.14. Only at the six outermost cells, the grid cell dimensions remain the same.

The smallest cells are half the size of the outfall width. The centre of the outfall will be placed at the 1000 m co-ordinate in the m-direction (longshore). In the vertical, 10 layers are applied, with a uniform thickness of 0.42 m.

5.2.3 Boundary conditions

The imposed boundary conditions in Delft3D are:

- | | |
|------------------------|--|
| <i>Lower boundary:</i> | The shore is represented by imposing no flow perpendicular to the shore at the boundary line. This means it can be seen as a vertical wall. (No actual data of the exact bathymetry were available.) |
| <i>Left boundary:</i> | At the left boundary a current of 0.09 m s^{-1} parallel to the shoreline is imposed as boundary condition. In Delft3D the depth averaged velocity is taken 0.09 m s^{-1} with a logarithmic distribution in the vertical. No turbulence is imposed at the boundary. |
| <i>Right boundary:</i> | Weakly reflective water level boundary (0.0 m) with a reflection coefficient α of 556.4 s^2 . |

<i>Upper boundary:</i>	Weakly reflective water level boundary (0.0 m) with a reflection coefficient α of 323.2 s ² .
<i>Bottom boundary:</i>	No flow through the cell faces perpendicular to the bottom. In horizontal direction a Chézy bottom coefficient C of 65 m ^{1/2} s ⁻¹ is applied.
<i>Surface boundary:</i>	Because Delft3D is a free-surface model, no special attention has to be given to this boundary. The free surface is implemented in the model by the kinematic boundary condition and a constant pressure at the surface. No temperature or wind boundary condition is applied.

The imposed boundary conditions in CFX are:

<i>Lower boundary:</i>	Wall boundary with no flow perpendicular to the shore and no shear stresses tangential to the wall (free slip).
<i>Left boundary:</i>	Inlet boundary (0.09 m s ⁻¹). This velocity is taken constant over the vertical. It is assumed that a logarithmic profile will (at least partially) be established where the jet fluid enters the ambient water. For reasons of similarity with Delft3D, turbulence is not imposed at the boundary. because this was not possible in Delft3D
<i>Right boundary:</i>	Outlet boundary / mass flow boundary. The net mass flow through the right and upper boundary is equal to the mass flow through the inflow boundaries.
<i>Upper boundary:</i>	Outlet boundary / mass flow boundary. The net mass flow through the right and upper boundary is equal to the mass flow through the inflow boundaries.
<i>Bottom boundary:</i>	Wall boundary. Zero velocity normal and tangential to the wall (smooth, $C \rightarrow \infty$) is specified.
<i>Surface boundary:</i>	Symmetry plane.

An alternative recommendation for applying mass flow boundaries (right and upper boundary) could be imposing a pressure boundary (thereby imposing the modified pressure). This would fit better with the Delft3D boundaries as a water level boundary is applied there. By using the hydrostatic pressure assumption in Delft3D, imposing a water level fixes the hydrostatic pressure at all points in the vertical.

For the CFX simulation it should be noted that the flow options ‘incompressible flow’ and ‘buoyant flow’ are turned on. For Delft3D this is default in the package.

In both CFX and Delft3D, the molecular viscosity is $\nu = 1.0 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}$ and the k - ϵ model is used to determine the eddy viscosity. In addition, in Delft3D a background (minimum) horizontal eddy viscosity of $\nu_{H,2D} = 1.0 \cdot 10^{-3} \text{ m}^2 \text{ s}^{-1}$ is applied. The other background values for viscosity and diffusivity are taken to be zero. [Note: additional runs without a background viscosity showed that the minimum horizontal background viscosity had no influence on the results.]

In the Delft3D runs, a Chézy bottom coefficient C of $65 \text{ m}^{1/2} \text{ s}^{-1}$ is applied. In the CFX runs, the bottom is taken to be hydraulically smooth. Additional runs in D3D with a Chézy bottom coefficient C of $500 \text{ m}^{1/2} \text{ s}^{-1}$ showed some influence on the results. However, these differences are within the margins of error to be expected from this type of simulation.

5.2.4 Implementation of the jet

In Delft3D the jet is brought into the system by use of a source term in the continuity and momentum equations. Another option, simulating the jet by using a velocity boundary condition, is presented in section 5.4. This alternative could not be used here, because Delft3D only allows velocities perpendicular to the boundary surface to be imposed at a boundary. (While an angle of 60° is demanded here.)

In CFX the jet is implemented as an inlet boundary, by prescribing velocity and temperature. Two components of the velocity are prescribed: a component normal to the boundary and one tangential to it. This is done in a way that the resulting velocity has an angle of 60° with the boundary and a magnitude of 0.57 m s^{-1} .

In both CFX and Delft3D no turbulence quantities were prescribed at the start (boundary or source) of the jet.

5.2.5 Computations and available data

Data already available:

- Field measurements from 1971
- Semi-empirical calculations by M&B [Miller and Brighthouse, 1984]

Computations made within the framework of this study:

- Delft3D-FLOW (hydrostatic)
- CFX4.4 (non-hydrostatic)
- CORMIX (expert-model)

5.3 The results

Plots of the results of the Delft3D and CFX computations can be found in Figure 5.18 to 5.21 in the back of this report.

5.3.1 Trajectory of the plume centreline

To enable a good comparison of the different models special attention is given to the trajectory of the plume centreline and the centreline temperature decay. Such data are available from measurements as well.

In Figure 5-3, all data of the centrelines of the computed/measured plumes are brought together. Also a line at a 60° angle with the coast is shown, because this is the initial direction of the discharge.

For the determination of the centrelines of the Delft3D and CFX simulations, only the top layer temperature distribution is considered. This is justified because the positive buoyancy makes the plume confined to the upper layers, with the highest temperatures at the top.

It is unknown at what depth the measurement data is gathered, but it is assumed that it is the temperature at the surface.

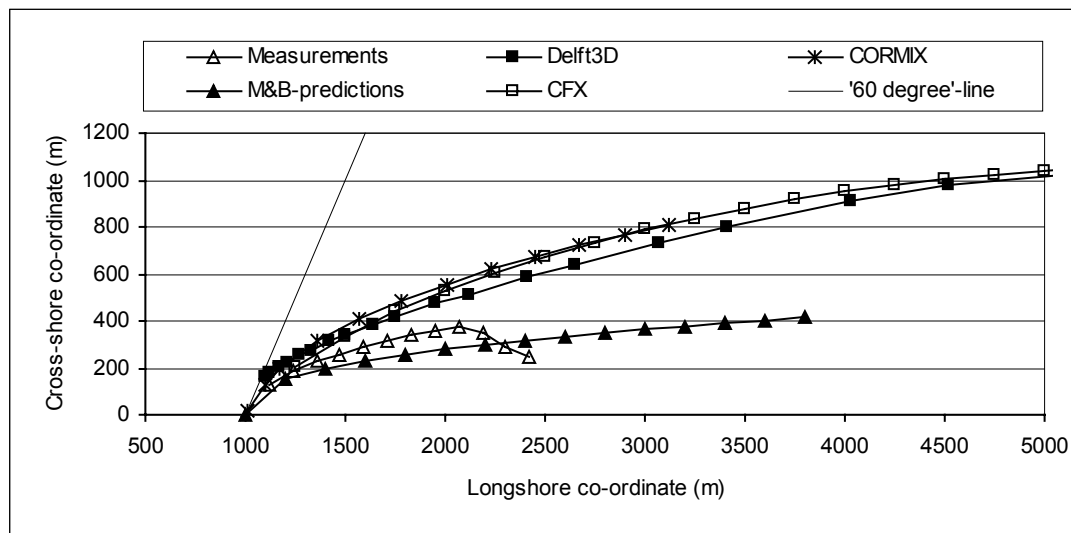


Figure 5-3: Comparison of the trajectory of the plume centrelines for various simulations of the Point Beach Power Station cooling water discharges.

From the measurements, the predictions and the CORMIX computations, specific data of the centreline trajectory were available. For the Delft3D and CFX data, the centreline had to be fitted through the temperature field of the uppermost layer. Cross sections of the plume at the upper layer (Delft3D) or the surface (CFX) are taken and the temperature-weighted centre of mass is determined, using a Matlab script. This centre is taken as a point on the centreline trajectory.

It should be noted that the cross sections of the plume are taken normal to the coast, whereas normal to the trajectory would be correct. This can lead to inaccuracies near the outfall, but further downstream, where the plume is already deflected, the difference in method disappears.

It is due to the fact that especially the Delft3D data does not have bell-shaped cross sections, that the centre of 'mass' is used and not the location of the highest temperature in a cross section. When the temperature cross sections are approximately bell shaped, the highest temperature

coincides with the centre of ‘mass’. In the Delft3D results maxima appear near the coast, but most of the excess heat is located further from the shore. The ‘centre of mass’ criterion is more stable because all data in a cross section are used, whereas the maximum temperature is only one data point. (To illustrate: temperature cross sections are shown in Figure 5-16 and 5-17 at the end of this chapter.)

Conclusions from Figure 5-3

The first thing that catches the eye when examining the resulting centreline trajectories is that the Delft3D, CFX and CORMIX plume trajectories are much less deflected than the measured plume. This may suggest that the differences with the measurement data are caused by inaccuracies in the bathymetry and other modelling assumptions. Other reasons could be that in the simulations there is no temperature exchange with the atmosphere and wind influences are assumed to be absent. (This is done because there are no relevant data available.)

A third reason could be that the discharge period is not known. Therefore the simulations are done until a steady state is reached, notwithstanding the fact that it is not known if this was the case in the measurements as well. (However, from the simulation data it can be seen that the discharge period does not have a great influence on the trajectory.)

Also striking is the small difference between the DELFT3D, CFX and CORMIX results, which is even more notable considering that CORMIX (an expert model for jets and plumes) is a model that works completely different from the more general flow packages DELFT3D and CFX.

The good agreement between the Delft3D and CFX trajectories suggests that the hydrostatic pressure assumption does not have much influence on the far-field results. (Note that this assumption is only used in the Delft3D package, CFX uses a momentum equation in all three dimensions.) However, the hydrostatic assumption may still have an influence on the spreading around the centreline.

5.3.2 Centreline temperature decay

In Figure 5-4, the excess temperature along the centreline relative to the initial excess temperature ($\Delta T/\Delta T_0$) is plotted. This quantity is related to the dilution of the excess temperature, being defined as $1-(\Delta T/\Delta T_0)$.

CORMIX gives the relative excess temperature ($\Delta T/\Delta T_0$) along the centreline in the data output. The relative centreline excess temperature of the measurements and the predictions made by Miller & Brighthouse was available in a graph. Note that the measurement data are available for only the first kilometre after the outfall. For Delft3D and CFX the temperature at the centreline is taken to calculate the relative excess temperature. Note that this is not necessarily the same as the highest temperature in a cross section of the plume.

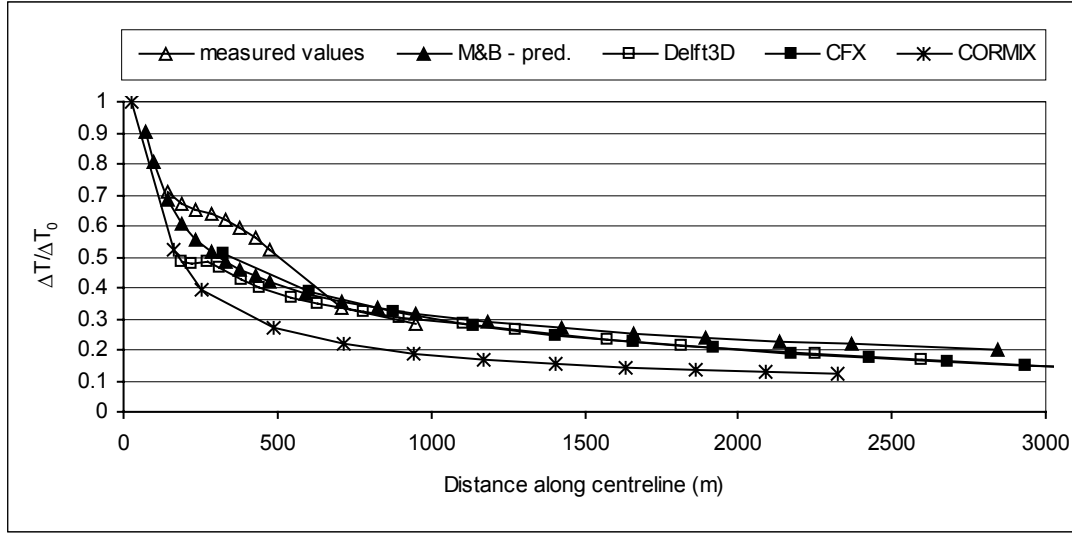


Figure 5-4: *Excess temperature along the jet trajectory relative to the initial excess temperature ($\Delta T/\Delta T_0$).*

Conclusions from Figure 5-4

Again, the agreement between the CFX and Delft3D lines is very good. Also the M&B-predictions fit very well with these simulations. The CORMIX relative excess temperature line on the other hand is considerably lower at every distance along the centreline.

Also striking is that in the first 500 metres, the measured excess temperature is higher than that in all of the calculations. However, further downstream the measured excess temperature approaches that of the Delft3D and CFX predictions. For the region further downstream (>1000 m from the outfall) there were no measurement data available.

Another thing that catches the eye in Figure 5-4 is that the start of the Delft3D line (at about 300 m downstream) is about horizontal where a steep line is expected. This may be attributed to the inaccuracy in determining the jet centreline trajectory around the outfall. (The cross section that is used for determining the centre of mass of the excess temperature is taken normal to the shoreline, instead of normal to the trajectory.)

5.3.3 Vertical temperature profile

In Figure 5-5, the vertical distribution of temperature is plotted at different locations along the centreline. See Figure 5-6 for the horizontal position of the temperature distributions.

It is clear that the vertical temperature distribution of the CFX computations is smoother than that of the Delft3D computations. This can be explained by the use of the 3D Navier-Stokes equations in CFX, whereas Delft3D uses the hydrostatic pressure assumption. Therefore the vertical velocities in Delft3D are overestimated, and the warmer, lighter water will be more easily transported to the top layer.

Another observation from Figure 5-5 is that the excess temperature in the lower layers is 0.2 to 0.3 °C for the CFX simulation. In the Delft3D results this approaches zero, which was also what was expected.

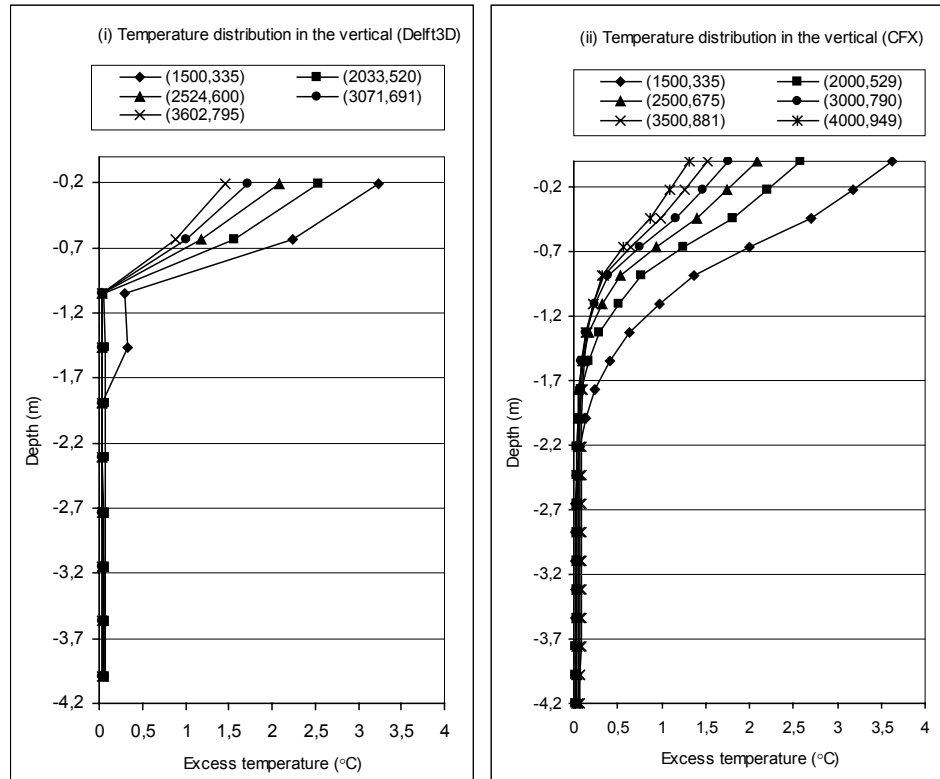


Figure 5-5: The vertical distributions of temperature taken along the plume centreline: (i) Delft3D, (ii) CFX. The legend gives the horizontal location of the profiles, also plotted in Figure 5-6.

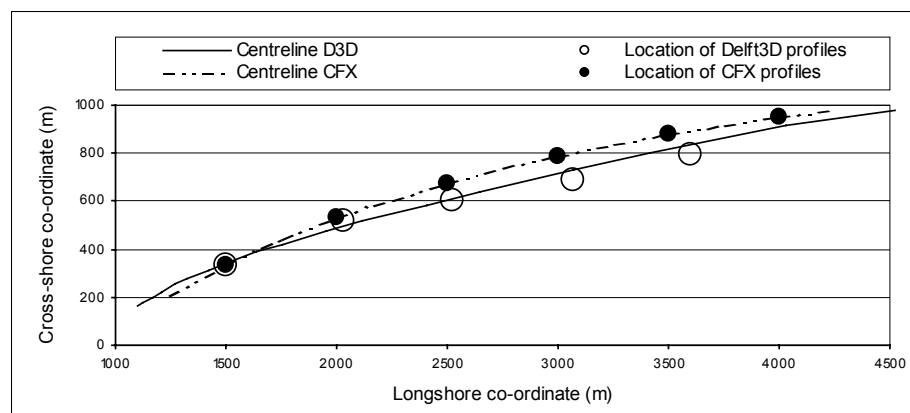


Figure 5-6: Horizontal position of the vertical profiles in Figure 5-5.

5.3.4 Plume thickness

To be able to compare the plume thickness in the various simulations, it has to be defined first. The criterion used in CORMIX is the $1/e$ ($\approx 36.8\%$) threshold, which is the depth at which the excess temperature is 36.8 % of the maximum excess temperature in the considered vertical. For CORMIX, the plume thickness data are automatically given in the output, for CFX and Delft3D this is computed in an Excel sheet.

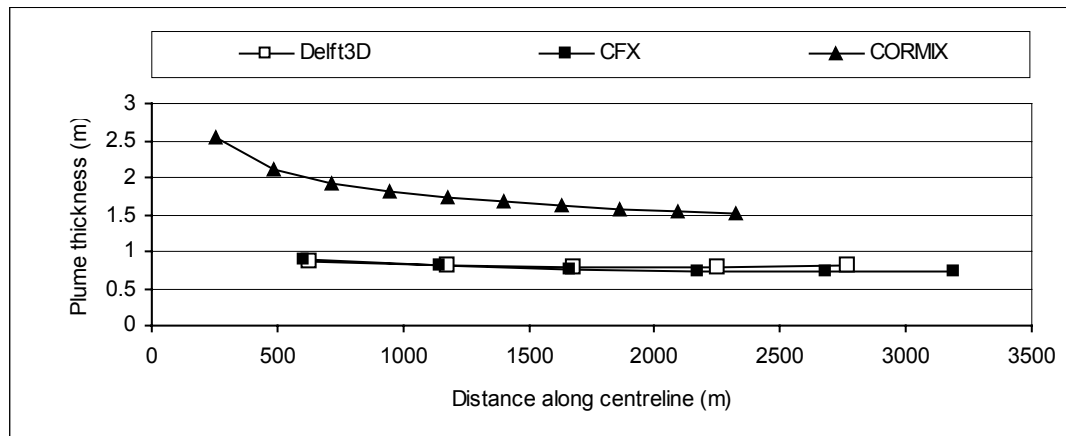


Figure 5-7: The plume thickness at the centrelines of CFX, Delft3D and CORMIX.

From Figure 5-7 it can be seen that the plume thickness in the Delft3D and CFX computations are almost the same, with a thickness of slightly less than 1 metre. The CORMIX plume thickness is considerably larger, going from 2.5 m at 250 m from the outfall to about 1.5 m further downstream. No measurement data about the plume thickness were available.

5.3.5 The near-field

In Figure 5-8, plots are presented from the direct surroundings of the discharge (Delft3D simulation). What catches the eye from the first plot (Figure 5-8 *i*) is that the initial direction of the jet (60° to the coast) is not recognisable in the vectors. From the point of discharge the vectors go in all possible directions, and only after intruding into the ambient water body deflection takes place. The immediate spreading can cause the jet to intrude less far into the ambient water. When we look at the magnitude of the horizontal velocity (cf. Figure 5-8 *ii*), two main directions can be distinguished. The highest and widest in the direction perpendicular to the coast and a smaller ‘plume’ almost parallel to the shoreline. Expected was a compact jet at an angle of 60° to the coast, widening in the direction of the flow. In the temperature distribution (Figure 5-8 *iii*), it can also be noticed that the warmer effluent spreads immediately in all horizontal directions after being discharged. Further from the outfall, deflection takes place. Another aspect that catches the eye in the third plot is the peak in temperature almost parallel to the shore. The last plot in Figure 5-8, (*iv*), shows a vertical distribution of the temperature. The cross section is taken at the place of the outfall, normal to the coast. The lighter (warmer) water is confined to the upper layers, almost

immediately after being discharged. Already at 25 m from the coast, no excess temperature is seen in the lower layers. In the CORMIX calculations, it is predicted that the plume detaches from the bottom only at 160 m (along the trajectory) from the outfall. Although this cross-section is not taken along the jet centreline trajectory, it seems that the vertical transport is overestimated.

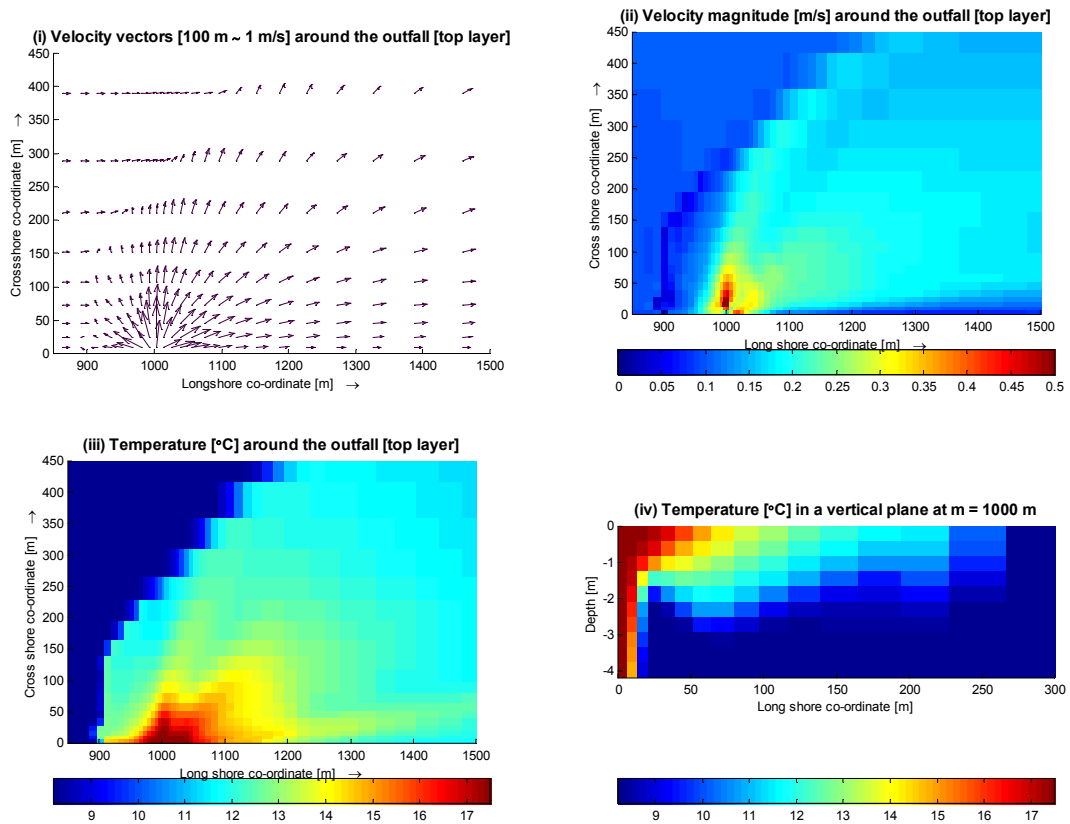


Figure 5-8: *Plots focusing on the direct surroundings of the outfall (Delft3D); (i) Horizontal velocity vectors in the top layer, (ii) Horizontal velocity magnitude in the top layer, (iii) Temperature distribution in the top layer, (iv) Temperature distribution in a vertical plane (normal to the shore, starting at the outfall).*

5.4 Sensitivity analysis

After analysing the results of the Delft3D simulation, it is decided to further examine the influence of two factors on the results: the open boundaries and the method of bringing the discharge in the computational domain.

5.4.1 Open boundaries

Figure 5-9 clearly shows the influence of the upper water level boundary on the velocity in the ambient field. It looks as if the plume has a tendency to attach to this boundary. The possible influence of the open boundaries was already anticipated by making the grid considerably larger than the area of interest, covering about 0 – 2 km downstream from the outfall.

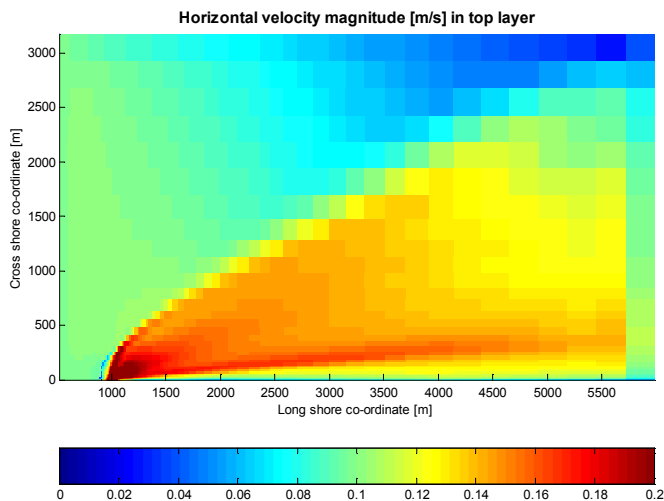


Figure 5-9: *The horizontal velocity magnitude in the top layer (Delft3D).*

However, to check whether the influence of the boundaries extends to the area of interest, the grid is enlarged in both horizontal directions.

Extending the initial grid with five grid cells in the longshore direction and five grid cells in the cross-shore direction does this. Therefore the total grid size becomes 6784.35 m × 4401.54 m. (This was 5458.31 m × 3170.14 m.) The size of the extra grid cells is equal to the outermost cell (expansion factor 1).

5.4.2 Discharge method

The influence of the method of discharge is measured by comparing the two possible methods.

The method initially used is adding a source term in the continuity and momentum equation. The other method – which is used in the CFX computations as well – is bringing the effluent into the computational area by means of a velocity boundary (CFX name: inlet boundary).

However, in Delft3D, it is only possible to impose a velocity perpendicular to the boundary cell face on a boundary. Velocities with components parallel to the cell face are not allowed, making it impossible to implement the 60°-angle discharge as a boundary.

However, to be able to check the influence of the type of boundary, the discharge situation is changed by giving the jet an initial angle of 90° with the coast. This situation is computed for

both a 'source term' discharge and a 'boundary condition' discharge. Plots of the Delft3D and CFX results can be found in Figure 5-22 to 5-28 in the back of the report.

5.4.3 Results of the sensitivity analysis

Results: Influence of open boundaries on centreline trajectory

Figure 5-10 shows the influence of the grid size for all discharge configurations in Delft3D. It can be concluded that the size of the grid still plays a (minor) role in the trajectory. This is because the open boundaries influence the results in an artificial way. The further the open boundaries, the less influence should be noticed.

As expected, the extension of the grid causes the centreline trajectory to move a little more to the shore, especially further downstream. However, in the area of interest (trajectory < 2000 m), the influence is negligible. Should there be interest in the results further downstream, it is advised that a larger grid be used.

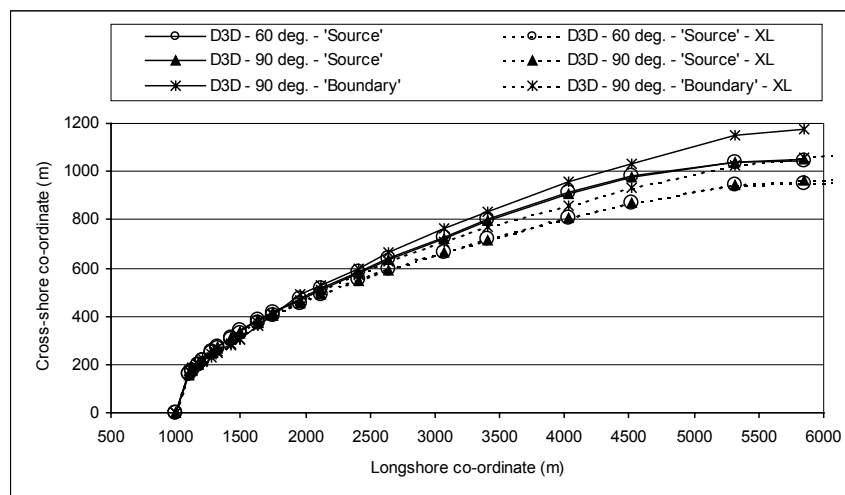


Figure 5-10: *Jet centreline trajectories compared for different grids in Delft3D. The lines marked with 'XL' represent the results using the enlarged grid.*

Results: Influence of open boundaries on centreline decay

The influence of the grid size on the centre line temperature decay in Delft3D is given in Figure 5-11. It is shown there that the position of the open boundaries has no significant influence on the centreline dilution. The centrelines for both grid sizes show no discernible differences. This holds for all three types of discharge configurations.

This outcome is understandable because the dilution and spreading of the jet is a much more local process compared to the establishment of the trajectory. Therefore influence of the remote boundaries is much less present.

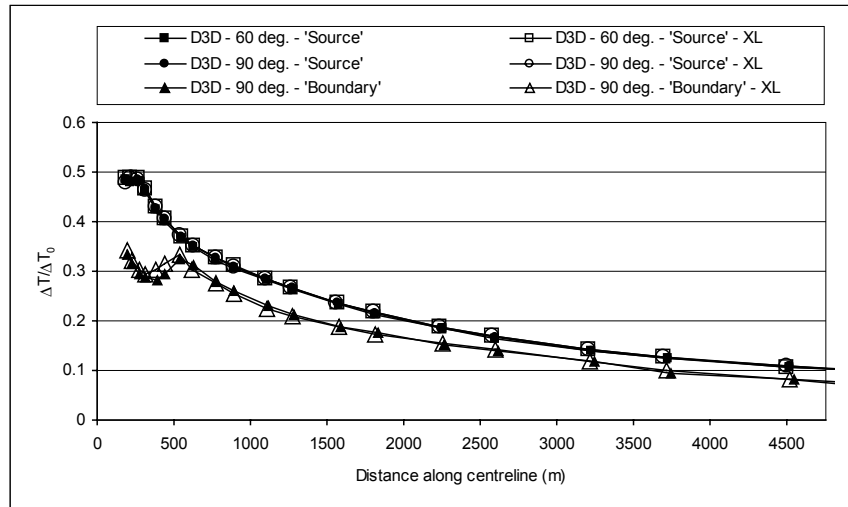


Figure 5-11: Plot showing the influence of an enlarged grid on the centreline decay. The lines marked in the legend with 'XL' represent the results using the enlarged grid.

Results: Influence of method of discharge on centreline trajectory

Figure 5-12 shows the jet centreline trajectories for all of the tested discharge situations in Delft3D. Also the trajectories of CFX simulations with an angle of 60° and 90° are added, to be able to further corroborate the effects of the changing of initial angle of discharge.

As can be seen in the figure, in the Delft3D simulations, the effect of changing the initial angle is negligible. The difference between the 60°- and 90°-angled Delft3D source term jets is hardly distinguishable in the graph. The CFX jets on the other hand, start with a small difference in trajectory, but at the end of the simulated domain this difference has disappeared.

The jet injected into the water body by boundary condition (Delft3D), starts following the same path as the other Delft3D jets. Only when reaching the far-field, this jet is deflected less than the others are. However, these differences in path are within the margins of accuracy that can be expected for this kind of simulations. Therefore the conclusion can be drawn that the method of discharge has no significant influence on the trajectory in this configuration. Furthermore, the trajectory in both Delft3D and CFX is not sensitive to changes in the initial discharge angle. In Delft3D the effect of changing the initial discharge angle it is not even visible at all.

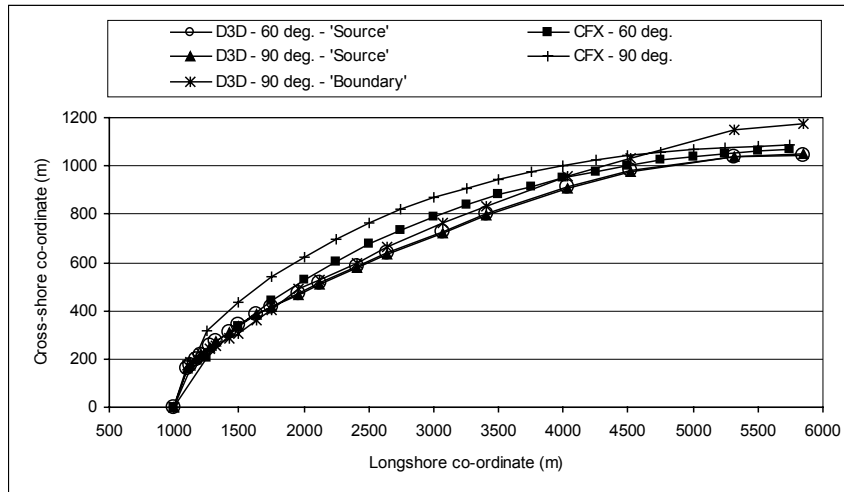


Figure 5-12: *Jet centreline trajectories for different types of discharge (Delft3D and CFX).*

Results: Influence of method of discharge on centreline temperature decay

The influence of the method of discharge on the temperature decay can be seen in Figure 5-13. In Delft3D the changing of initial direction has no influence on the temperature decay at all. In CFX, the centre line temperature decay becomes a little larger (lower excess temperature) when the angle is increased, but this is not significant, especially in the far-field.

The discharge method has also a noticeable, but insignificant effect on the relative centreline excess temperature ($\Delta T/\Delta T_0$). The decay is slightly higher if the jet is simulated with a boundary condition and this result fits very well with the CFX result at 90°.

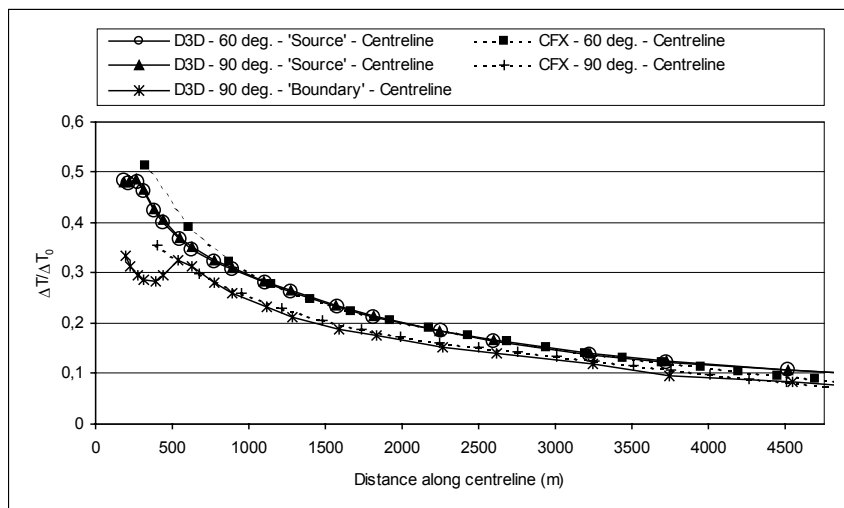


Figure 5-13: *Plot showing the influence of the method of discharge on the relative excess temperature ($\Delta T/\Delta T_0$).*

Results: Influence of method of discharge on near-field (Delft3D)

Figure 5-14 shows that the velocity vectors of the boundary type discharge in Delft3D have a much more distinct direction. Especially when looking at the velocity magnitude a momentum forced jet can be seen starting at an angle of 90° to the shore, slowly deflecting in the direction of the cross flow. The jet is much more compact and keeps a higher maximum velocity further downstream.

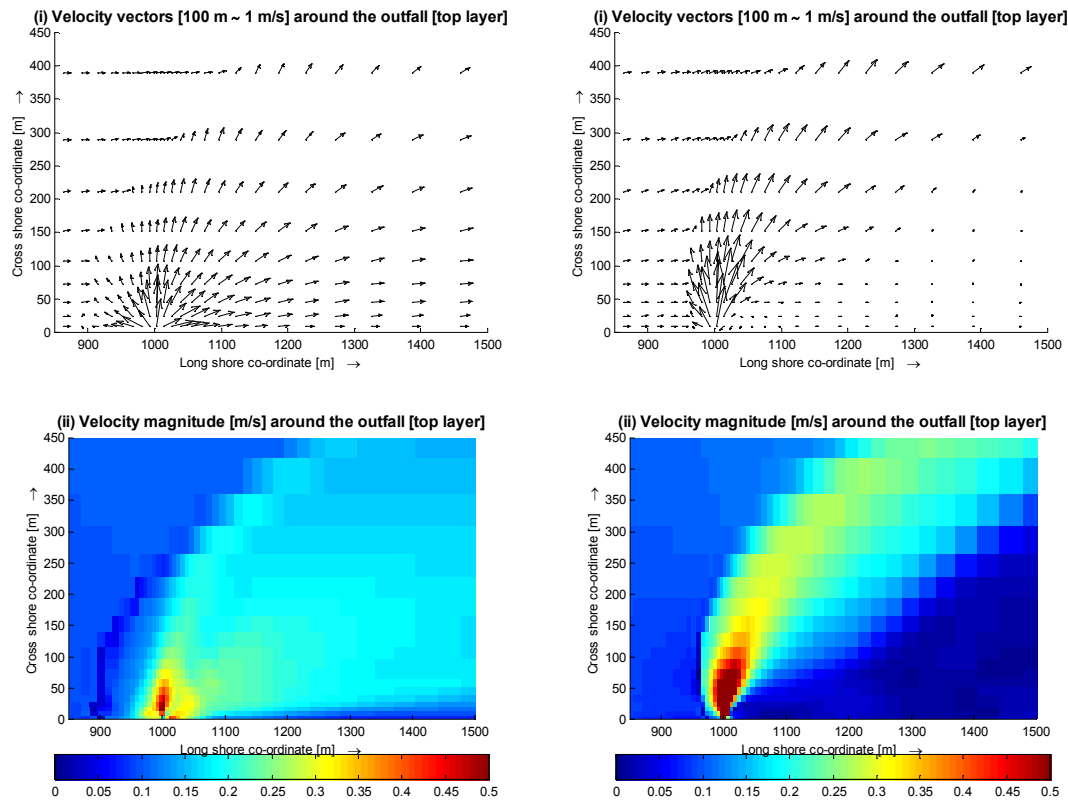


Figure 5-14: *Plots comparing the direct surroundings of the outfall; (i) Horizontal velocity vectors in the top layer; discharge by source term, (ii) Horizontal velocity vectors in the top layer; discharge by boundary condition, (iii) Velocity magnitude in the top layer; discharge by source term, (iv) Velocity magnitude in the top layer; discharge by boundary condition.*

5.5 The temperature distribution around the centreline

In the graphs previously shown, the temperature at the centreline is taken to calculate the relative excess temperatures. However, if the maximum temperature in the cross section of the jet is taken instead, the results show something different. These results are presented in Figure 5-15.

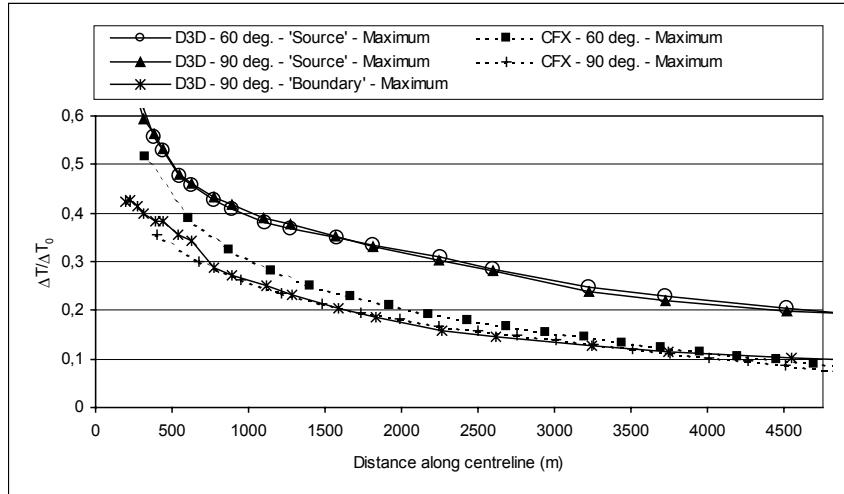


Figure 5-15: *Plot showing the influence of the method of discharge on the centreline excess temperature decay relative to the initial excess temperature, using the maximum temperature in a cross section to calculate this.*

Comparing the results in Figure 5-15 with those in Figure 5-13 gives rise to the following remarks. Using the ‘maximum temperature’ dilution criterion the results remain roughly the same for the CFX computations and the Delft3D computation using a velocity boundary condition. The dilutions for the Delft3D source term simulations on the other hand are much higher with this criterion. From this, the conclusion can be drawn that there is a significant difference between the two methods of discharge.

The difference between the centreline criterion and the maximum temperature criterion also suggest that the temperature distribution around the centreline is not very regular or bell-shaped.

To make a more detailed comparison, further attention is given to the temperature distribution around the jet trajectory centreline. Two cross sections are taken: one around the 2000 m longshore co-ordinate and the other around the 4000 m longshore co-ordinate.

These temperature distributions around the centreline are plotted in Figure 5-16 and 5-17. The vertical line is located at the ‘centre of mass’ of the temperature distribution. The height of the vertical line is the maximum temperature in the profile.

The first thing that catches the eye is the local peak in temperature near the shore, which only occurs using the source term discharge. The difference between the two directions in the Delft3D simulations is virtually non-existent. In CFX the difference is larger, but this diminishes further along the centreline.

The Delft3D discharge, using a boundary condition, fits much better with the CFX results. Also it does not have the strange peak near the shore.

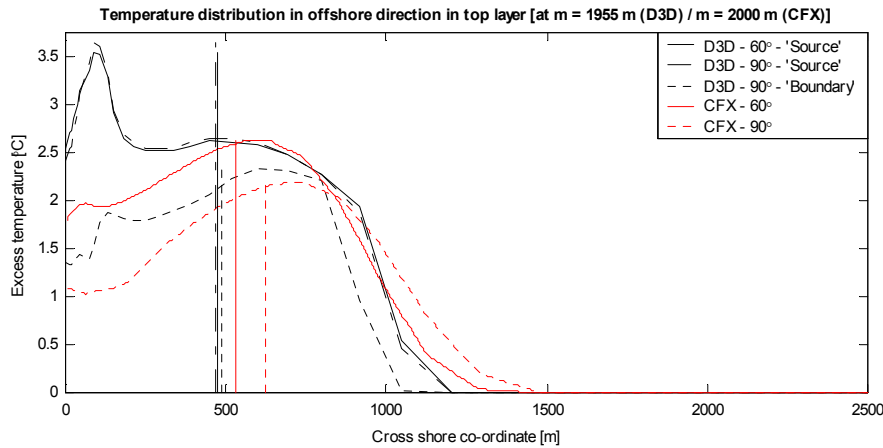


Figure 5-16: *Excess temperature distribution in a cross section perpendicular to the coast at m = 1955 m (Delft3D) / m = 2000 m (CFX). The vertical lines represent the 'centre of mass' under the graphs.*

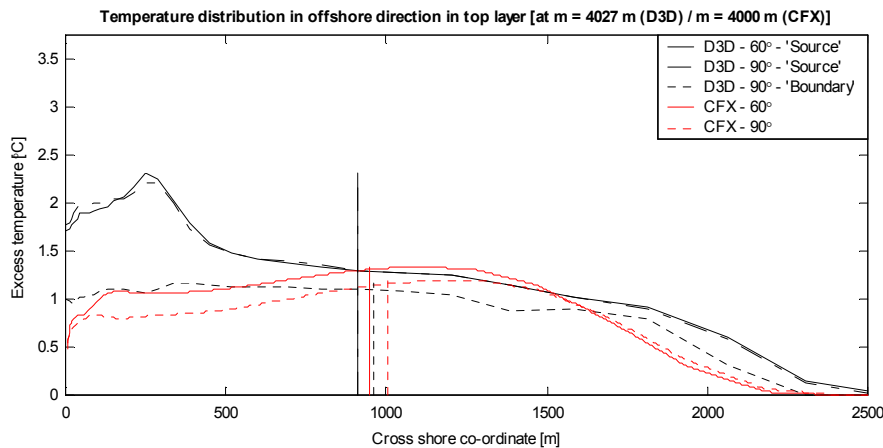


Figure 5-17: *Excess temperature distribution in a cross section perpendicular to the coast at m = 4027 m (Delft3D) / m = 4000 m (CFX). The vertical lines represent the 'centre of mass' under the graphs.*

Another striking thing that the temperature distributions show is that the excess temperature in the Delft3D simulation using a boundary to simulate the jet is lower at every point of the cross sections than that of the other Delft3D simulations. This does not mean that heat is not conserved. It should be noted that the cross sections only involve the top layer. Only when all layers are considered conclusions about this can be drawn. Moreover, to draw conclusions about conservation of heat, the velocity perpendicular to the cross section should also be considered.

5.6 Conclusions

In this chapter, a buoyant jet was simulated in both Delft3D (hydrostatic) and CFX (non-hydrostatic) to see if the influence of the hydrostatic assumption on the results. From this, the following conclusions can be drawn:

- There is a good agreement between the Delft3D and CFX results concerning the jet trajectories, the centreline temperature decay and the plume thickness
- The Delft3D and CFX jet trajectories correspond well with the jet trajectory predicted by CORMIX.
- The vertical temperature distribution in the CFX results is smoother than in the Delft3D results.

Also in this chapter, the sensitivity of the simulated jet to the initial direction of the discharge and the discharge method is investigated. From this, the following conclusions can be drawn:

- In Delft3D, the trajectory of the jet is not influenced by the initial discharge direction, if the outfall of the jet is modelled as a source term in the continuity and momentum equations. In CFX there is a difference, however this diminishes in the far field.
- The results in the near-field improve, using a velocity boundary condition instead of a source term to model a jet outfall in Delft3D. The horizontal velocity is much less diffused and shows a much more distinct initial direction.

Comparison of the temperature distribution around the jet centreline showed:

- The use of a velocity boundary to model a jet outfall in Delft3D yields a temperature distribution around the jet centreline that is in better agreement with the results in CFX (compared to using a source term in Delft3D).

Overall it can be concluded that there is not much difference in the far-field results due to the hydrostatic assumption.

6 2DV jets with Delft3D and CFX

6.1 Introduction

From chapter 5, it was concluded that a hydrostatic simulation of the near- and far-field yielded similar results in the far-field, compared to the non-hydrostatic simulation (CFX). In this chapter more attention will be given to the near-field and the influence of this on the far-field.

To further see if instabilities can be found when simulating jets, more simulations have been made. A simple configuration is chosen with only two dimensions to save computer time. In this way many different situations can be examined in a short time.

A simulation focussing on the near-field is chosen, although Delft3D does not claim to solve this correctly, due to the hydrostatic assumption. However, this is also the region where instabilities can be expected.

In section 6.2, the results of a 2DV simulation of a buoyant jet in Delft3D will be presented. In section 6.3, a similar jet will be simulated in CFX, to be able to make a comparison with the result of the non-hydrostatic Delft3D model.

In section 6.4, more jets will be simulated in Delft3D, to show the sensitivity of the stability to various parameters.

In section 6.5, attention is given to the (dynamic) pressure distribution of a non-hydrostatic jet.

6.2 Delft3D buoyant jet simulation

6.2.1 Outline of the case

A 2DV (two-dimensional: one horizontal dimension, one vertical dimension) channel is chosen with a depth of 10 m and a length of 30 m. At 30 m from the outfall a water level (undisturbed level) is imposed, with reflection coefficient α of 3.06 s².

At the left vertical boundary, the jet and the co-flowing ambient current are modelled by imposing the velocity. The direction of the imposed velocities is perpendicular to the boundary.

The buoyant jet initiates between 1.0 m and 2.0 m above the bottom.

Some important parameters of the discharge situation are:

- Discharge (per metre) $q_o = 5.0 \text{ m}^3 \text{ s m}^{-1}$
- Jet velocity $u_o = 0.5 \text{ m s}^{-1}$
- Height of outfall $h_o = 1.0 \text{ m}$
- Ambient temperature $T_a = 10.0^\circ\text{C}$
- Discharge temperature $T_o = 20.0^\circ\text{C}$ ($\Delta T_o = 10.0^\circ\text{C}$)
- Ambient current velocity $u_a = 0.01 \text{ m s}^{-1}$ (co-flowing)

6.2.2 Implementation

To be able to model this situation, an equidistant grid (ignoring small changes in the vertical due to surface elevation) is made with two dimensions. In the third dimension the grid consists of one grid cell with a width of 1.0 m. Because zero flow through the frictionless side of the channel is imposed, the grid is considered as 2D.

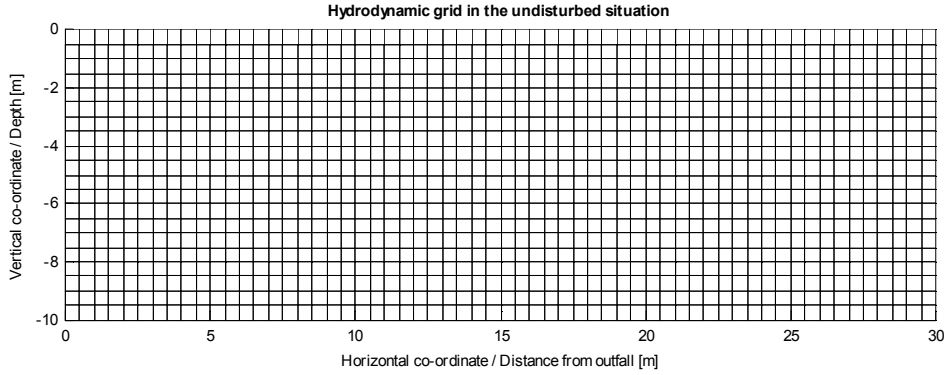


Figure 6-1: *Hydrodynamic 2DV grid in the undisturbed situation (sigma co-ordinates).*

The grid characteristics are:

- No. of grid points in horizontal direction 63
- No. of grid points in vertical direction 20
- Cell size $\Delta x \times \Delta z = 0.5 \text{ m} \times 0.5 \text{ m}$
- Total grid size $30 \text{ m} \times 10 \text{ m} (x \times z)$

The k - ϵ model is used to determine the (turbulent) eddy viscosity. In addition, a minimum horizontal background eddy viscosity $\nu_H^{(t,2D)} = 0.1 \text{ m}^2 \text{ s}^{-1}$ is applied. The additional vertical eddy viscosity is set to zero, as well as the additional diffusivity in both directions. (This means that the minimum (molecular) value of $1.0 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}$ is used.)

The time step Δt used for running this model is $0.0125 \text{ min} = 0.75 \text{ s}$. This gives a numerical Courant number C_{fl} of

$$C_{fl} = \sqrt{gh} \frac{\Delta t}{\Delta x} = \sqrt{9.81 \cdot 10} \frac{0.75}{0.5} = 14.9$$

which should be low enough to yield stable results. Also, a test with a considerably lower Courant number showed no different results.

The total running time is 40 minutes, which is sufficient to converge to a steady state solution.

6.2.3 Results

Results of the simulation are presented Figure 6-2; they can also be found at larger scale in Figure 6-5 in the back of this report.

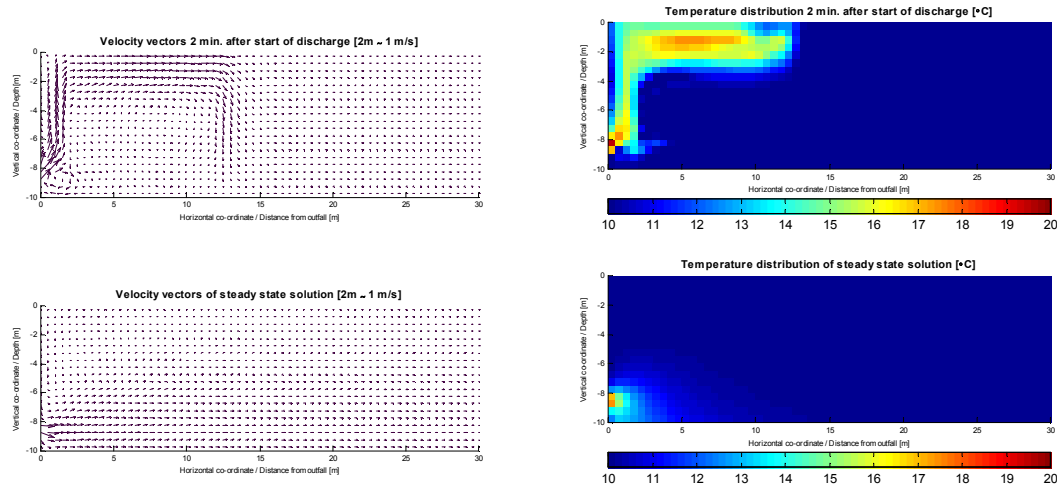


Figure 6-2: *Results of a 2DV buoyant jet simulation in Delft3D: (i) Velocity vectors at $t = 2$ min, (ii) Temperature distribution at $t = 2$ min, (iii) Velocity vectors of the steady state solution, (iv) Temperature distribution of the steady state solution. (Numbering from left to right.)*

The results in the first minutes after the start of the jet are acceptable, in that the warmer jet water comes to the surface and starts flowing downstream. However, looking at the details the results are not as they can be expected in physical reality. The buoyant jet water comes to the surface immediately when entering the ambient water body. In reality the jet would at first mainly be driven by the flux of momentum, driving it in horizontal direction. Only later on, the buoyancy would take over and drive the flow to the free surface. This would give a much smoother trajectory.

Also the front of the plume travelling in downstream direction seems very steep with high velocities pointing downward.

Another odd detail is that the highest temperatures cannot be found in the top layer, but slightly lower.

The steady state results are even more unrealistic. The warmer water that is present stays at the bottom. Also it does not reach the other side of the channel, showing that the excess heat is not conserved. A weak wake is formed directly above the outfall reaching to the free surface. However this circulation of water cannot be found in the plot of the temperature, which is strange because the excess temperature should move along the streamlines of the water particles.

6.3 CFX buoyant jet simulation

To be able to distinguish the effects of the hydrostatic assumption in the Delft3D run, the same case is implemented in CFX. The dynamic (molecular) viscosity is set to a value of $\mu = 0.001 \text{ Pa s}$. With a reference density of 1000.0 kg m^{-3} , this gives a kinematic viscosity $\nu = 1.0 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}$, which is the same as in the Delft3D runs. Also similar to the Delft3D runs is the use of the k- ϵ model to supply the turbulent viscosity. The only difference is that in this (CFX) simulation, no additional horizontal eddy viscosity is applied.

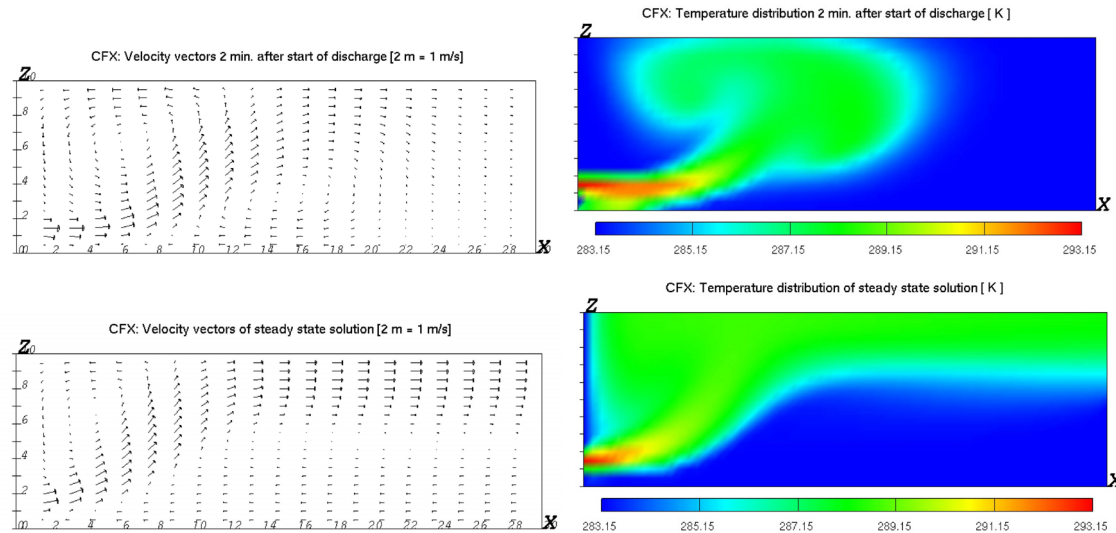


Figure 6-3: *Results of a 2DV buoyant jet simulation in CFX: (i) Velocity vectors at $t = 2 \text{ min}$, (ii) Temperature distribution at $t = 2 \text{ min}$, (iii) Velocity vectors of the steady state solution, (iv) Temperature distribution of the steady state solution. (Numbering from left to right.)*

Unlike the Delft3D run, these (CFX) results show a stable solution (cf. Figure 6-3 and 6-13). Also the CFX results after 2 minutes are considerably different from the Delft3D results. The jet starts horizontally, which is to be expected given the initial momentum flux. A little further downstream the buoyancy also starts to show its effects, bending the trajectory in a more upward direction. At the front of the plume circulation can be seen in both directions. The steady state solution shows a fully mixed area above the first metres of the jet. This is due to the re-entrainment of effluent. The re-entrainment is intensified by the wake present in the upper left corner of the simulated water body.

6.4 Sensitivity analysis

To get a broader idea of what instabilities and other problems can arise when modelling buoyant discharges more runs have been made in Delft3D. This might also give some clues about the importance of the various parameters for the problems experienced in the first run of this chapter.

The varied parameters are presented in Table 6-1. In each run only one parameter is changed compared to the reference model (first run).

Table 6-1: *Test runs with reference number and changed parameters. In the last column the new densimetric Froude number of the outfall F_o is given.*

Run	Changed parameter	New value	Old value	F_o
1	Reference model	-	-	4.23
2	Jet velocity	$u_o = 1.0 \text{ ms}^{-1}$	$u_o = 0.5 \text{ ms}^{-1}$	8.47
3	Jet velocity	$u_o = 0.25 \text{ ms}^{-1}$	$u_o = 0.5 \text{ ms}^{-1}$	2.11
4	Temperature difference	$\Delta T_o = 20.0^\circ\text{C}$	$\Delta T_o = 10.0^\circ\text{C}$	2.55
5	Temperature difference	$\Delta T_o = 5.0^\circ\text{C}$	$\Delta T_o = 10.0^\circ\text{C}$	6.67
6	Ambient current velocity	$u_a = 0.1 \text{ ms}^{-1}$	$u_a = 0.01 \text{ ms}^{-1}$	4.23
7	Ambient current velocity	$u_a = 0.0 \text{ ms}^{-1}$	$u_a = 0.01 \text{ ms}^{-1}$	4.23
8	Add. hor. eddy viscosity	$\nu_H^{(t,2D)} = 0.001 \text{ m}^2 \text{ s}^{-1}$	$\nu_H^{(t,2D)} = 0.1 \text{ m}^2 \text{ s}^{-1}$	4.23

The results of these Delft3D runs can be found in Figure 6-6 to 6-12.

Looking at these results, a few things can be noted:

- All jets have steep fronts.
- High vertical velocities are present in the front reaching to far below the plume.
- High vertical velocities ($>0.5 \text{ m s}^{-1}$) exist where the jet enters the ambient water body. (Except for the case with the high ambient velocity (Run 6, Figure 6-10).) Theory and CFX simulations suggest that, at the start, momentum flux should dominate leading to a horizontal outflow. These results suggest that the horizontal momentum flux is not simulated well.
- Most jets have the same ‘strange’ steady state solution. Only the runs with the low jet velocity (Run 3, Figure 6-7), high temperature difference (Run 4, Figure 6-8) and lower viscosity (Run 8, Figure 6-12) have a more or less stable buoyant upper layer. Run 3 and 4 (cf. Figure 6-7 and 6-8) simulate the jets with the lowest densimetric Froude numbers at the outfall. This suggests that to make a stable discharge, the Froude number should be lower than 3. This would be a rather strict criterion, considering that many jets found in the literature have higher Froude numbers at the outfall (roughly ranging from 1 to 10).
- At the start jets have a tendency to attach to the bottom or the vertical boundary.
- The case with the low additional horizontal eddy viscosity (Run 8, Figure 6-12) has a stable steady state solution. However, during a long time after the start of discharging, small vortices occur at the density interface. These are transported through the open boundaries.

Explanation

Where the angle of the density interface or the horizontal density gradient is large, the hydrostatic assumption leads to non-physical phenomena. In most cases (where a stable buoyant layer exists) this means an overestimation of the vertical velocities.

This is easy to explain considering that conservation of momentum in the vertical is not accounted for. Vertical momentum can be generated freely (without inertia); only the continuity equation governs the vertical flow.

Not only are the vertical velocities high, it is also striking that the high vertical velocities can be found immediately behind the outfall. This is not what is expected, as theory says that, close to the outfall, momentum flux will dominate the flow leading to a flow in the direction of the discharge velocity. Only later, when the momentum is more diffused, the buoyancy force will take over. This is because the effects of the buoyancy need some time to be noticeable, as the fluid starts with a zero vertical velocity (when no vertical momentum flux is present). Normally, it will take some time to gradually accelerate the fluid particles by the buoyancy force, thereby increasing the effect on the fluid velocity and trajectory.

However, using the hydrostatic assumption, the vertical acceleration is not restricted by inertia, leading to immediate buoyancy effects after the outfall, and consequently an earlier vertical deflection and high velocities.

[Note: The high vertical velocities are made *possible* by the absence of a vertical momentum balance. By no means does this mean that this absence will always lead to the high vertical velocities in other discharge situations, as the vertical motion is still influenced by the demand of continuity.]

6.5 Modified pressure

As the impact of the hydrostatic pressure assumption seems so large in the previous near-field simulations, a closer look is taken at the modified pressure distribution in the (non-hydrostatic) CFX run.

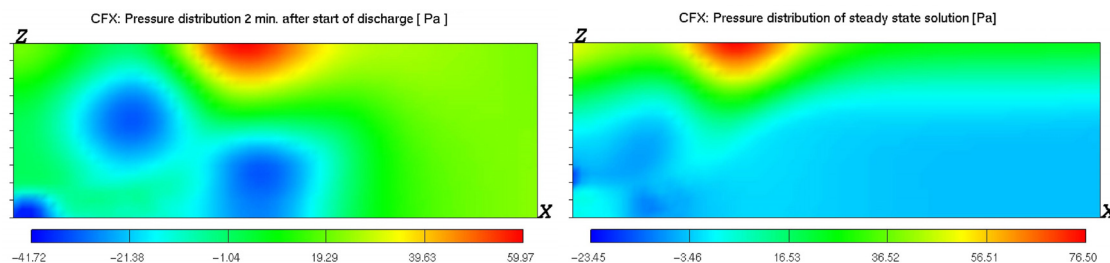


Figure 6-4: Results of a 2DV buoyant jet simulation in CFX: (i) Modified pressure [Pa] distribution at $t = 2 \text{ min}$, (ii) Modified pressure [Pa] distribution of the steady state solution.

In Figure 6-4, the modified pressure at 2 minutes after the start of discharging and the steady state is plotted. The modified pressure is defined as (cf. Equation (3.55))

$$p_{\text{mod}} = p - \rho_0 g (\zeta_0 - z) \quad (6.1)$$

This is the total pressure p minus the static pressure (assuming no contribution of the atmospheric pressure). In other words, the modified pressure is the sum of the dynamic pressure, the barotropic pressure and de baroclinic pressure (cf. section 3.7).

In the plots several low-pressure areas can be found. Two minutes after the start of discharge, the lowest pressures can be found directly under the outfall and behind the circulating jet fronts. In the steady state solution, there is less underpressure. (Note that the colour scales in both plots are different.) Only above and under the start of the jet slightly lower pressures exist.

The plots show that the highest pressure can be found where the plume reaches the surface. Note that the CFX model uses a fixed symmetry line to represent the water surface. The (barotropic) pressure gradients at the top should be seen as a water level gradient. The presence of the high barotropic pressure shows that the water level gradient plays an essential role in establishing the flow pattern. It drives the effluent in the horizontal directions and drives the circulation in the left upper corner.

The modified pressure seems very low, having a range of about 100 Pa in the whole area, while the static pressure at the bottom amounts to about 100 kPa, which is a thousand times larger. However, in incompressible flow the absolute value of the pressure is not relevant, only gradients in the dynamic pressure can influence the flow. This becomes clear when we consider Equation (3.14), which is repeated here:

$$\frac{D\mathbf{u}}{Dt} = \left(\frac{\Delta\rho}{\rho_0} \right) \mathbf{G} - \frac{1}{\rho_0} \text{grad } \Delta p + \nu \nabla^2 \mathbf{u} \quad (6.2)$$

The only pressure in this equation is Δp , which is defined as the modified pressure.

6.6 Conclusions

In this chapter, 2DV simulations are made in both Delft3D and CFX to see if the hydrostatic assumption can cause instabilities in the near-field (and thereby also affecting the far-field).

It can be concluded that in all hydrostatic simulations, the jet behaves physically unrealistic. This is shown by the unrealistically high vertical velocities ($>0.5 \text{ m s}^{-1}$) where the jets enter the ambient water body. In reality, at the beginning, the jet would be driven mainly by the flux of momentum, driving it in horizontal direction. CFX results confirm this and show that there should be a wake present above the outfall. Furthermore the density fronts moving in downstream direction are too steep with too high velocities pointing downward (compared with the non-hydrostatic results in CFX).

While the hydrostatic results are not physically realistic, this does not necessarily mean that the results are unstable, that is, not resulting in a warmer upper layer in the far field. Stable steady state results in Delft3D occur only when the densimetric Froude number at the outfall is low ($F_0 < 3$).

7 2DV model with pressure correction

7.1 Introduction

Tests in Delft3D (described in the previous chapter) showed that a hydrostatic simulation could yield results where positively buoyant jet water does not come to the surface. To further investigate the effect of the hydrostatic pressure assumption in this, more simulations are made. This is done using a program that can switch between a hydrostatic and a non-hydrostatic mode by switching the pressure correction off and on. Because the other parameters and model characteristics remain unchanged, a good insight can be given in the exact effects of the hydrostatic pressure assumption.

Numerous effects play a role in simulating buoyant jets, being a combination of forced convection and buoyancy driven convection. Because this is a rather complex combination of physical phenomena, first, a closer look is taken at the simulation of an exchange flow (section 7.2). This is a purely buoyancy driven phenomenon, of which some analytical data are available such as the front speed and the slope of the nose.

Second, an intrusive gravity current is simulated (section 7.3). Of this case some analytical information is available concerning the intrusion speed and the layer thickness.

Finally, in section 7.4, a simulation of the jet considered in the previous chapter will be described.

The computational model

The model that is used is a pressure corrected fixed layer (Cartesian) model. It uses a staggered grid. The equations are solved using a fractional step method, thereby dividing a time step in three phases. In the first step a hydrostatic solution is obtained. With the now available information of the (intermediate) velocity field the mass distribution for the new time step is determined. Therefore a finite volume method is used. The third and last stage in a time step is the calculation of the pressure correction, for which a Poisson equation has to be solved. The program uses a conjugate gradient method to solve the Poisson equations.

For more detailed information concerning the program, the reader is referred to Bijvelds [1995].

7.2 Lock exchange

7.2.1 Model set-up

A closed basin with a length L of 15.0 m and an initial depth H of 2.5 m is considered, which at time $t = 0$ is divided into two areas with different densities:

$$\rho = \begin{cases} 1010.0 \text{ kg m}^{-3} & \text{if } 0 \leq x < L/2 \\ 1000.0 \text{ kg m}^{-3} & \text{if } L/2 \leq x \leq L \end{cases} \quad (7.1)$$

All velocities are zero at $t = 0$. The grid size is equal in both directions, being $\Delta x = \Delta z = 0.1 \text{ m}$. The time step Δt is 0.015 s , which yields a Courant number $C_{fl} = 0.74$. The Courant number is taken this low, because of the explicit treatment of the mass transport. Other adjusted parameters include a reference density ρ_0 of 1005.0 kg m^{-3} and a Chézy bottom coefficient C of $100 \text{ m}^{1/2} \text{ s}^{-1}$. Both the viscosity and the diffusivity are set to zero.

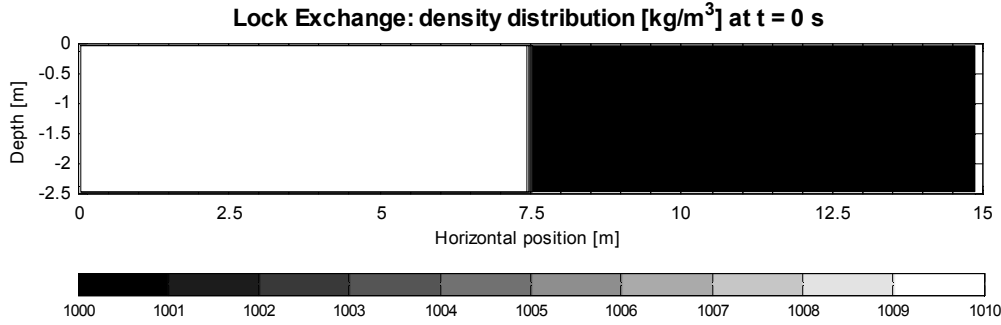


Figure 7-1: Initial density distribution in the lock exchange case.

7.2.2 Expected phenomena

When time starts running, a baroclinic pressure gradient will induce gravity currents at the bottom as well as the free surface. The heavier water will tend to intrude under the lighter fluid, whereas the light water intrudes along the free surface over the heavier water. This will result in two fronts moving in opposite direction.

Theoretical front speed and angle

The speed of the front of the density current can be deduced by considering the energy budget of the system (assuming no viscosity).

At $t = 0$ the total amount of potential energy per unit volume is equal to $\frac{1}{2} gH(\rho_1 + \rho_2)$.

Assuming a frictionless flow, with no mixing between the layers, the system must consist of two layers with a different density and of equal thickness (cf. Turner 1973, p. 71). The potential energy per unit volume of the transient state equals $\frac{3}{4} gH\rho_1 + \frac{1}{4} gH\rho_2$. Because of conservation of energy, the net change in potential energy per unit volume, $\frac{1}{4} gH\Delta\rho_{1,2}$, must be equal to the total kinetic energy per unit volume, $\frac{1}{2} \rho_0 u_f^2$, hence the front speed u_f is given by

$$u_f = \frac{1}{2} \sqrt{\frac{gH\Delta\rho_{1,2}}{\rho_0}} = \frac{1}{2} \sqrt{g'H} \quad (7.2)$$

with $g' = g(\Delta\rho_{1,2}/\rho_0)$.

In this specific case, this amounts to a theoretical front speed of $u_f = 0,247 \text{ m s}^{-1}$.

Furthermore, the slope of the nose at the stagnation point (water surface and bottom) is 60° to the horizontal according to theoretical analysis [Turner 1973, p. 71].

7.2.3 Results

Both a hydrostatic and a non-hydrostatic run are made, with the previously described set-up. The results of these runs can be found in Figure 7-6 to 7-12 in the back of this report.

Figure 7-6 and 7-7 show the velocity vectors for various times during the run. It shows clearly that the differences in velocities can be found mainly in the density fronts. The vertical velocities found in the fronts of the hydrostatic results are much higher than those in the non-hydrostatic results are. Furthermore, the hydrostatic vertical velocities are also very concentrated in the fronts, whereas in case of the non-hydrostatic run, the vertical velocities can also be found around the front.

Figure 7-8 and 7-9 show the density distribution for various times during the run. In both the runs diffusion of density can be seen, although this is not explicitly built in in the mathematical model. An explanation for this is numerical diffusion, caused by the discretisation of the equations. The main difference between the two runs is the steepness of the density front. In case of the hydrostatic run the fronts are almost vertical, whereas in the non-hydrostatic run this amounts to about 60° to the horizontal.

In Figure 7-10 and 7-11 the vertical velocities are plotted separately, using contour lines. These plot show more clearly what could already be noted from the vector plot: the vertical velocities in case of the hydrostatic computation are much higher than the non-hydrostatic velocities and the vertical flow is much more concentrated in the (vertical) density front. The maximum non-hydrostatic vertical velocities are less than 0.1 m s^{-1} , whereas in the hydrostatic case the vertical velocities can be higher than 0.7 m s^{-1} .

From Figure 7-12 it can be noted that the high dynamic pressures (and pressure gradients) can be found mainly around the fronts.

7.2.4 Comparison with theory

To be able to compare the results with the available theory, the front speed and the front angle are determined for the hydrostatic as well as the non-hydrostatic run. The front position and angle are determined by fitting a third order polynomial through the 1005 kg m^{-3} -iso-line of the density. The fitting is done to get a more realistic result; only using the last two data points of the iso-line gave very unstable results. The polynomial is chosen to be of the third order, because this gave the best fit where the front borders the horizontal boundary (bottom or free surface). This is the location where the angle is determined.

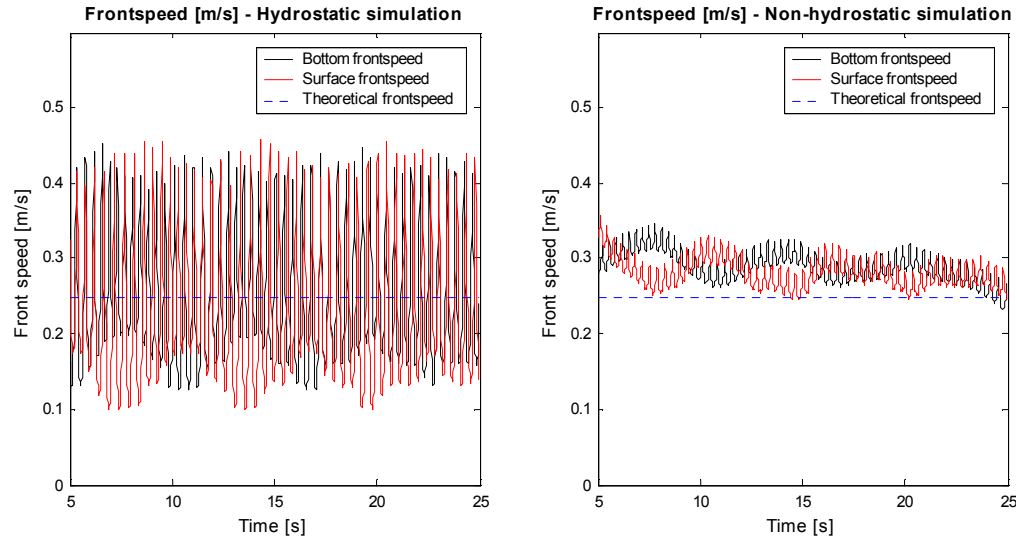


Figure 7-2: *Front speed in the time range of 5 to 15 s for the bottom and the surface front (hydrostatic and non-hydrostatic).*

Figure 7-2 shows the front speeds as a function of time. The time frame starting from 5 s is chosen because in the first 5 seconds, the speed still has to develop starting from zero. Because there is a large (cyclic) variation during the plotted time frame, the averages are determined and shown in Table 7-1. The results can also be found in Figure 7-3, showing the travelled distance of the fronts plotted against time.

Table 7-1: *Average and standard deviation of front speed in the time frame of 5 - 25 s.*

	Avg. front speed [m s^{-1}]	$u_f / \sqrt{g'H}$ [-]	Std. Dev. [m s^{-1}]
Bottom front - Hydrostatic	0.242	0.490	0.081
Surface front - Hydrostatic	0.227	0.459	0.090
Bottom front - Non-hydrostatic	0.289	0.584	0.020
Surface front - Non-hydrostatic	0.280	0.568	0.019
Theory	0.247	0.500	-

In both in the hydrostatic as well as in the non-hydrostatic case, the bottom front speed is larger than the surface front speed. This is not in agreement with measurements, showing the higher front speed on the surface. [Turner 1973, p. 72] quotes multiplying constants $u_f / \sqrt{g'H}$ of 0.47 for the underflow and 0.59 for the overflow.

A possible explanation for this deviation of theory might be an inaccuracy in representing the free surface in the fixed layer model.

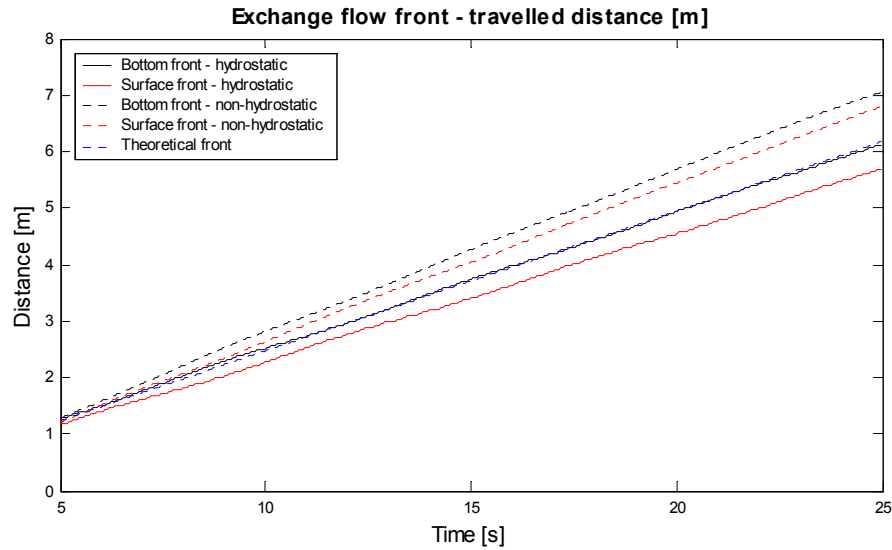


Figure 7-3: *Travelled distance of the bottom and surface fronts plotted against the time. (Hydrostatic, non-hydrostatic and theoretical results.)*

The angle of the nose at the stagnation point fits well with theory for the non-hydrostatic run. As can be seen in Figure 7-4, both the bottom and the surface nose develop an angle of 60° to the horizontal. In the hydrostatic case, the slope is unrealistically steep at about 80° to the horizontal.

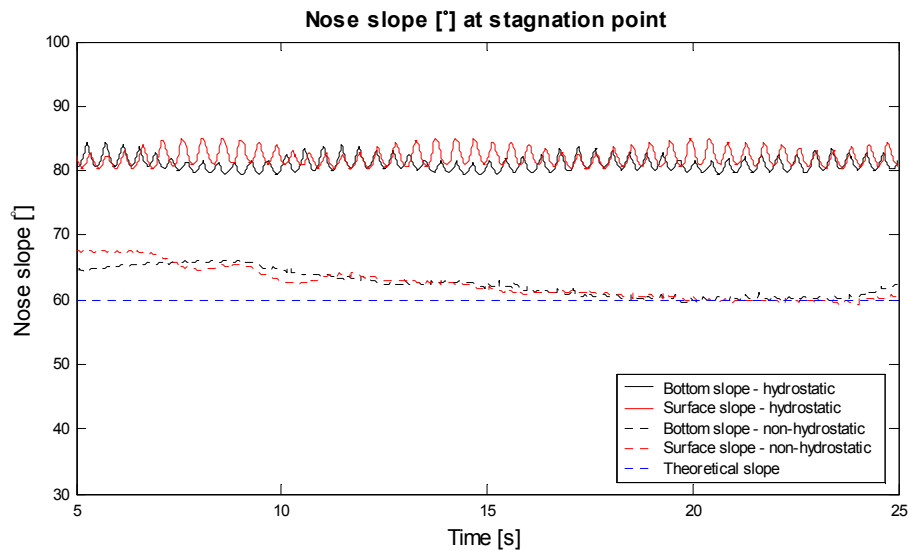


Figure 7-4: *Slope of the bottom and surface nose to the horizontal at the stagnation point. (Hydrostatic, non-hydrostatic and theoretical results.)*

7.3 Intrusive gravity current

7.3.1 Model set-up

Again a closed basin with a length L of 15.0 m and a depth H of 2.5 m is considered, this time divided (at time $t = 0$) into four areas with different densities:

$$\rho = \begin{cases} \rho_1 = 1010.0 \text{ kg m}^{-3} & \text{if } 3.0 \text{ m} \leq x < 15 \text{ m} \wedge -2.5 \text{ m} \leq z \leq -1.65 \text{ m} \\ \rho_2 = 1000.0 \text{ kg m}^{-3} & \text{if } 3.0 \text{ m} \leq x < 15 \text{ m} \wedge -0.85 \text{ m} \leq z \leq 0.0 \text{ m} \\ 1010.0 \text{ kg m}^{-3} - (\rho_1 - \rho_2) \frac{1.65 \text{ m} + z}{0.8} & \text{if } 3.0 \text{ m} \leq x < 15 \text{ m} \wedge -1.65 \text{ m} < z < -0.85 \text{ m} \\ \frac{1}{2}(\rho_1 - \rho_2) = 1005.0 \text{ kg m}^{-3} & \text{if } 0.0 \text{ m} \leq x < 3.0 \text{ m} \end{cases} \quad (7.3)$$

This density distribution is symmetrical with respect to the centre line at half depth. The other parameters are kept unchanged compared to the lock exchange case (e.g. $\Delta t = 0.015 \text{ s}$, $\Delta x = \Delta z = 0.1 \text{ m}$).

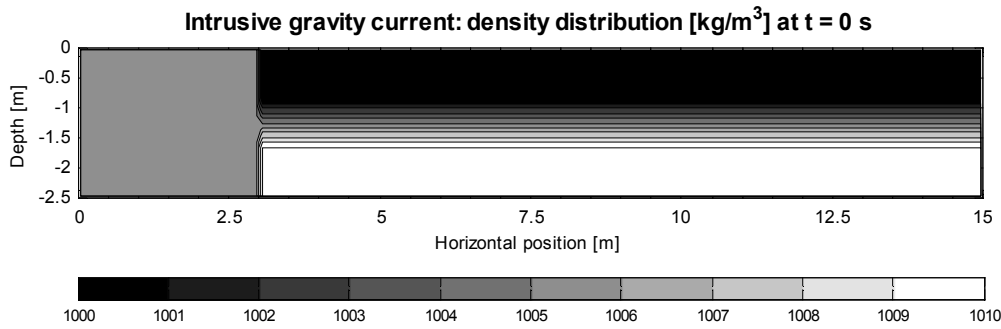


Figure 7-5: Initial density distribution in intrusive gravity current case.

7.3.2 Expected phenomena

When time starts running, a baroclinic pressure gradient will induce gravity currents, moving the fluid on the left-hand side of the basin to intrude between the layers on the right. Furthermore, the density fronts at the surface and the bottom will move to the left.

Theoretical intrusion speed

Ignoring viscous effects and mixing, a theoretical expression for the intrusion speed u_i can be derived:

$$u_i = 0.2637 \sqrt{g'H \left[1 - 4\alpha_0 \left(\frac{\delta_0}{H} \right) + 8\alpha_1 \left(\frac{\delta_0}{H} \right)^2 \right]} \quad (7.4)$$

with δ_0 the initial thickness of the density interface layer and profile coefficients α_0 and α_1 (cf. Bijvelds 1995). Furthermore, according to this theory, the intruding layer thickness h_i is given by:

$$h_i = 0.347H + 0.653\delta_0 \quad (7.5)$$

In this specific case, the initial interface layer thickness δ_0 is 0.8 m and the profile coefficients α_0 and α_1 amount to respectively 1/4 and 1/24. Substituting this in Equation (7.4) yields a theoretical intrusion speed of $u_i = 0.110 \text{ m s}^{-1}$. The theoretical intruding layer thickness h_i amounts to 1.39 m.

7.3.3 Results

Both a hydrostatic and a non-hydrostatic run are made, with the previously described set-up. The results of these runs can be found in Figure 7-13 to 7-19. Again it is clear from the results that the differences between the two runs lie mainly in the shape of the density fronts. Using the hydrostatic assumption the fronts remain steeper.

It is also observed again that the bottom front moving left runs slightly faster than the surface front moving to the left.

7.3.4 Comparison with theory

To be able to compare the results with the available theory, the intrusion speed and the intruding layer thickness for both the hydrostatic and the non-hydrostatic run are determined (cf. Table 7-2).

Table 7-2: *Intrusion speed and layer thickness of the intrusive gravity current case.*

	Intrusion speed u_i (m s ⁻¹)	Intruding layer thickness h_i (m)
Hydrostatic	0.133	1.6
Non-hydrostatic	0.138	1.6
Theoretical	0.110	1.39

The computed intrusion speed is in both cases somewhat higher than the maximum theoretical intrusion speed. It remains unclear what causes this difference. The measured intruding layer thickness has a maximum of around 1.6 m. This is higher than the theoretical layer thickness, which can be explained by numerical (artificial) diffusion which causes mixing. Using a finer grid can reduce this effect.

7.4 Buoyant momentum jet

7.4.1 Model set-up

The same set-up as in chapter 6 is chosen to be able to make a comparison. A short description is repeated here; for the full description of the case see section 6.1.1.

A 2DV channel is taken with a depth of 10 m and a length of 30 m. At 30 m from the outfall a water level (undisturbed level) is imposed, with reflection coefficient α of 3.06 s^2 .

At the left vertical boundary, the jet and the co-flowing ambient current are modelled by imposing the velocity. The direction of the imposed velocities is perpendicular to the boundary.

The buoyant jet initiates between 1.0 m and 2.0 m above the bottom. Some important parameters of the discharge situation are:

- Discharge (per metre) $q_o = 5.0 \text{ m}^3 \text{ s m}^{-1}$
- Jet velocity $u_o = 0.5 \text{ m s}^{-1}$
- Height of outfall $h_o = 1.0 \text{ m}$
- Ambient temperature $T_a = 10.0^\circ\text{C}$
- Discharge temperature $T_o = 20.0^\circ\text{C}$ ($\Delta T_o = 10.0^\circ\text{C}$)
- Ambient current velocity $u_a = 0.01 \text{ m s}^{-1}$ (co-flowing)

Not all parameters could be taken the same as in chapter 6. The time step had to be decreased for this simulation because of the numerical handling of advection. This demands a Courant number lower than one, whereas in the Delft3D simulation, a value of $C = 15$ was sufficient. Also, the horizontal background viscosity is changed to a lower value ($\nu_H^{(t,2D)} = 0.001 \text{ m}^2 \text{ s}^{-1}$). This is done to diminish the effect of the viscous terms compared to the other terms (for example the buoyancy term). The current model uses a different (more basic) turbulence model; an algebraic Prandtl mixing length model, which makes the viscous effects difficult to compare.

This viscosity change makes the discharge situation more comparable to run 8 of the Delft3D 2DV runs (section 6.4) and the 2DV simulation carried out in CFX (section 6.3). However, in this chapter it will still be referred to as run 1.

The discharge and initial ambient temperature are converted to a density because the 2DV model works only in terms of density. The conversion is done by means of the empirical relation presented in Equation (3.18), with an assumed salinity of zero. Note that the relation between the density and the temperature of the water is not linear.

For the ease of reading an overview of the runs made is given in Table 7-3.

Table 7-3: *Test runs with reference number and changed parameters. In the last column the densimetric Froude number of the outfall is given. (2DV fixed layer model).*

Run	Changed parameter	New value	Old value	F_o
1	Reference model	-	-	4.23
2	Jet velocity	$u_o = 1.0 \text{ m s}^{-1}$	$u_o = 0.5 \text{ m s}^{-1}$	8.47
3	Jet velocity	$u_o = 0.25 \text{ m s}^{-1}$	$u_o = 0.5 \text{ m s}^{-1}$	2.11
4	Temperature difference	$\Delta\rho = 3.912 \text{ kg m}^{-3}$ ($\Delta T_o = 20.0^\circ\text{C}$)	$\Delta\rho = 1.422 \text{ kg m}^{-3}$ ($\Delta T_o = 10.0^\circ\text{C}$)	2.55
5	Temperature difference	$\Delta\rho = 0.573 \text{ kg m}^{-3}$ ($\Delta T_o = 5.0^\circ\text{C}$)	$\Delta\rho = 1.422 \text{ kg m}^{-3}$ ($\Delta T_o = 10.0^\circ\text{C}$)	6.67
6	Ambient current velocity	$u_a = 0.1 \text{ m s}^{-1}$	$u_a = 0.01 \text{ m s}^{-1}$	4.23
7	Ambient current velocity	$u_a = 0.0 \text{ m s}^{-1}$	$u_a = 0.01 \text{ m s}^{-1}$	4.23

7.4.2 Results

The results of the runs described in Table 7-3 can be found in Figure 7-20 to 7-40 in the back of the report.

First a comparison between the hydrostatic and non-hydrostatic results of run1 (cf. Figure 7–20 to 7–26) is made:

- Both runs have roughly the same front speed, the hydrostatic front being slightly slower, which is similar to the results in the lock exchange case.
- There is a steeper density front in case of the hydrostatic simulation.
- Both runs reach a stable steady state solution where the lighter water is confined to the upper layer.
- The layer thickness in the far-field is about the same if the density contour line at 1/3 of the initial density difference is taken as a measure for the layer thickness.
- The main differences can be found in the near-field. In the hydrostatic computation, the discharged buoyant water immediately goes to the surface with vertical speeds of over 0.5 m s^{-1} , whereas in the hydrodynamic run, the water first moves in a horizontal direction. Only further from the outfall the buoyancy of the water takes over from the horizontal momentum and the lighter water flows to the surface. This process generates a wake between the jet and the vertical above the outfall.
- The vertical velocities change much more smoothly in horizontal direction if using the pressure correction. This is clearly shown in Figure 7-19, where the contour lines of the vertical velocity are rounder in shape than the contour lines in Figure 7-20, showing the hydrostatic result.

Comparison of run1 (hydrostatic) with run8 of the Delft3D runs (Figure 6-12) shows that the front speed in this model is a lot higher. This may be attributed to the difference in turbulence model. While the horizontal background (eddy) viscosity is the same in both models ($v_H^{(t,2D)} = 0.001 \text{ m}^2 \text{ s}^{-1}$), the turbulence model is different. Delft3D uses the k- ϵ model to determine the eddy viscosity, while the 2DV program presented in this chapter uses the Prandtl mixing length model.

Overall the results have the same characteristics, showing high vertical velocities immediately downstream of the outfall.

Compared with the run in CFX, the results are quite similar. After 2 minutes the velocity and density distribution show the same patterns, with the jet in the 2DV model being more developed in the horizontal direction than in CFX. The steady state solutions look even more similar, with a comparable layer thickness at 30 m from the outfall. The stagnation point (the point at the surface where the horizontal velocity is zero) of the 2DV pressure corrected run is at about 10 m from the outfall; the CFX result shows it at slightly less than 9 m from the outfall.

Dynamic pressure

The highest dynamic pressures can be found in the region immediately downstream of the discharge. This coincides with the region with the highest vertical velocities. This shows that the hydrodynamic pressure (gradients) is necessary to suppress unrealistically high vertical accelerations and velocities.

Other runs

Runs 2 to 7 made with this 2DV pressure corrected model can be found in Figure 7–27 to 7–32. Below, the differences in output between the hydrodynamic and hydrostatic results of those runs will be analysed.

Run2: The higher jet velocity makes the hydrostatic results a lot worse. The hydrodynamic results are as expected; a wake is formed above the jet and a stable lighter upper layer is formed with velocities away from the outfall. In the hydrostatic results, the lighter water stays confined to the bottom with irregular, high, vertical velocities occurring.

Thus, high Froude numbers make pressure correction necessary.

Run3: Both the hydrostatic as the pressure corrected results have a stable solution. The hydrostatic simulation shows more diffusion with a heavier upper layer.

Run4: Again both solutions are stable. The upper layer of the hydrodynamic run is thinner with higher velocities.

Run5: In the hydrostatic run, a strange irregular pattern of high vertical velocities occurs. However, near the end of the simulation domain, a stable layer exists, with the warmer water confined to the upper layer.

- Run6: Both the hydrostatic and the non-hydrostatic results show a lighter water layer positioned at the bottom. In both cases this is the steady state solution. From this it can be concluded that this is not caused by neglect of the hydrodynamic terms. Possibly, a better turbulence model will yield totally different results. However, it is also possible that this result is similar to the physical reality. The high ambient velocity delivers more water to the jet than it can entrain. Therefore the buoyant jet is forced down to accommodate the forced convection of the ambient water.
- Run7: The results of run7 are similar to those of run1. Only difference in the hydrostatic results is that the first vertical column of vectors points downward in run1, whereas in run 7 it points upwards. In the hydrodynamic results, the ambient flow forces the stagnation point of the jet about 3 m more downstream (from 7 to 10 m). This caused by an ambient flow of 0.01 m s^{-1} , which is not much compared to the jet velocity of 0.5 m s^{-1} . In both the hydrostatic and the non-hydrostatic runs, the ambient flow influences the upper layer density. The forced ambient flow is mixed with the lighter jet water, thereby causing an upper layer with a higher density.
- Run6b: Run 6 is repeated, this time with a higher background viscosity ($\nu_H^{(t,2D)} = 0.1 \text{ m}^2 \text{ s}^{-1}$). The results for the pressure corrected run are almost equal to those of run6. In the hydrostatic results, it can be seen that instabilities arise even a long time after the start of the discharge (cf. $t = 600 \text{ s}$ and $t = 1200 \text{ s}$). This is only seen in the hydrostatic computation. From this it can be concluded that some instabilities can arise caused by a combination of hydrostatic computation and high viscosity. It should be noted that the turbulence model is rather primitive.

7.5 Conclusions

In this chapter the effect of the hydrostatic pressure assumption was investigated by simulating in both the hydrostatic and the non-hydrostatic mode of a 2DV program, without changing other parameters.

First, two purely buoyancy driven cases were simulated: the lock exchange problem and an intrusive gravity current. The main improvements using the non-hydrostatic pressure correction (compared to hydrostatic simulation) were:

- More realistic (lower) vertical velocities in the density fronts. Also the spatial change in velocity was much smoother.
- The angle of the density fronts fits with theory. This was not the case in the hydrostatic results where the fronts were too steep (almost vertical).

The main differences were found at locations with high horizontal pressure gradients (density fronts). This was also where the highest dynamic pressures were located. It should also be noted that the horizontal front speeds were almost unaffected by the pressure correction.

Second, various buoyant jets were simulated. Non-hydrostatic simulation proved necessary to prevent physically incorrect results. All simulated jets started with a horizontal momentum flux, which should result in a horizontal jet trajectory near the outfall, gradually moving to the surface. This was only the case in the non-hydrostatic simulations; in the hydrostatic simulations the jet stayed at the bottom, or moved to the surface immediately downstream of the outfall. Furthermore, the same improvements as in the purely buoyancy driven cases were noticeable here: more realistic (lower) vertical velocities with a smoother spatial distribution and a more realistic (less steep) angle of the density front.

In hydrostatic simulations accompanied by a high densimetric Froude number ($F_\theta > 5$) or a strong ambient (co-) flow, the warmer jet water did not come to the surface. In the case of the strong ambient (co-) flow, non-hydrostatic simulation did not improve the situation. In the case of the high densimetric Froude number, pressure correction made the warmer water to come to the surface and form a stable layer.

8 2DV hydrodynamic sigma model with pressure interpolation

8.1 Introduction

In this chapter the effect of using less pressure layers compared to the number of velocity layers will be examined. Therefore another model is used, which was built to allow uncoupling of the pressure and velocity layers. The pressure is interpolated in the vertical to be able to solve the horizontal momentum equations in the intermediate velocity layers. The model is based on the σ -transformed equations, which has the advantage of a fixed number of layers and an efficient treatment of the free surface and bottom.

As the existing model only accommodated a constant density, the model had to be extended. This involved adding a transport equation and a baroclinic pressure term in the horizontal momentum equation. The original model is described in [Van Reeuwijk 2002]. More information about the assumptions, derivation and extension of the model can be found in Appendix F of this report. A drawback of this hydrodynamic sigma layered model is that it is not able to simulate weakly reflective alpha boundaries. Also, it has no turbulence model incorporated.

Because this hydrodynamic model is different than the fixed layer model in chapter 7, the same experiments of the previous chapter (lock exchange and intrusive gravity current) are repeated here to test the performance of the model. The various jets are simulated again as well, using this program to see if the results in chapter 7 are model dependent.

In section 8.5, the effect of diminishing the number of pressure layers on the stability of the results is examined. This is done for the buoyant jets, as well as the lock exchange case and the intrusive density current.

8.2 Lock exchange - sigma layer model

8.2.1 Model set-up

The same lock exchange case as in the previous chapter is used here. For the exact description of the set-up, see section 7.2.1 (see Figure 7-1 for a graphic overview of the initial conditions).

The expected phenomena are described in section 7.2.2. Summarised, for this case, the maximum theoretical front speed amounts to $u_f = 0,247 \text{ ms}^{-1}$. Furthermore, the slope of the nose at the stagnation point (water surface and bottom) is 60° to the horizontal according to theoretical analysis [Turner 1973, p. 71].

8.2.2 Results

With this model, only a non-hydrostatic run can be made, of which the results can be found in Figure 8-13 to 8-16.

Figure 8-14 shows somewhat smoother results in the density distribution compared to the previously used model. Figure 8-15 shows somewhat higher maximum vertical velocities. (For example at $t = 18\text{ s}$ the maximum vertical velocity in the fixed layer model is less than 0.11 m s^{-1} , compared to less than 0.13 m s^{-1} in the sigma layer model.) Otherwise, the results look similar (in pattern) to the results in section 7.2.

8.2.3 Comparison with theory

Again, to be able to compare the results with the available theory and with the previous results, the front speed and the front angle are determined. The front position and angle are determined by fitting a third order polynomial through the 1005 kg m^{-3} -iso-line of the density. The fitting is done to get a more realistic result; only looking at the last two data points of the iso-line gave very unstable results. The polynomial is chosen to be of the third order, because this gave the best fit near the stagnation point of the front.

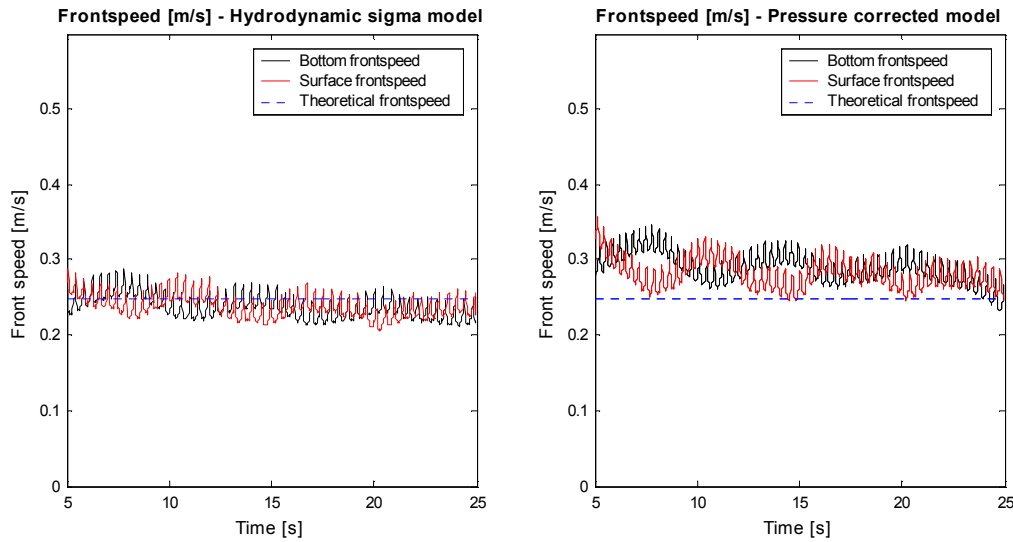


Figure 8-1: *Front speed in the time range of 5 to 15 s for the bottom and the surface front. (Using the hydrodynamic sigma layer model and the pressure corrected (non-hydrostatic) fixed layer model.)*

Figure 8-1 shows the front speeds as a function of time. The time frame starting from 5 s is chosen because in the first 5 seconds, the speed still has to develop starting from zero. Again there is a large (cyclic) variation during the plotted time frame, with on first sight the same frequencies as in the fixed layer model. The averages of the front speed are determined and shown in Table 8-1 where also the data of the other model can be found.

Table 8-1: *Average and standard deviation of front speed in the time frame of 5 - 25 s. (For the hydrodynamic sigma model and the pressure corrected fixed layer model.)*

	Avg. front speed [m s^{-1}]	$u_f / \sqrt{g'H} [-]$	Std. Dev. [m s^{-1}]
Bottom front - Sigma model	0.237	0.480	0.015
Surface front - Sigma model	0.237	0.480	0.014
Bottom front - Fixed layer	0.289	0.584	0.020
Surface front - Fixed layer	0.280	0.568	0.019
Theory	0.247	0.500	-

The results show that concerning the front speeds, this model performs well when compared to the model used in chapter 7 and the theoretical results. The front speeds are a little lower than the theoretical front speeds. This can be explained by the existence of numerical diffusion. Unlike the fixed layer model, the sigma model produces identical front speeds for the bottom and the surface. This can be explained by a better and easier handling of the free surface in sigma layer models.

Considering the theoretical angle of the nose at the stagnation point, the agreement with the results of the sigma layer model is not as good as the agreement with the results of the fixed layer model of chapter 7. As can be seen in Figure 8-2, both the bottom and the surface nose of the sigma layer run develop an angle going from about 55° to 50° to the horizontal.

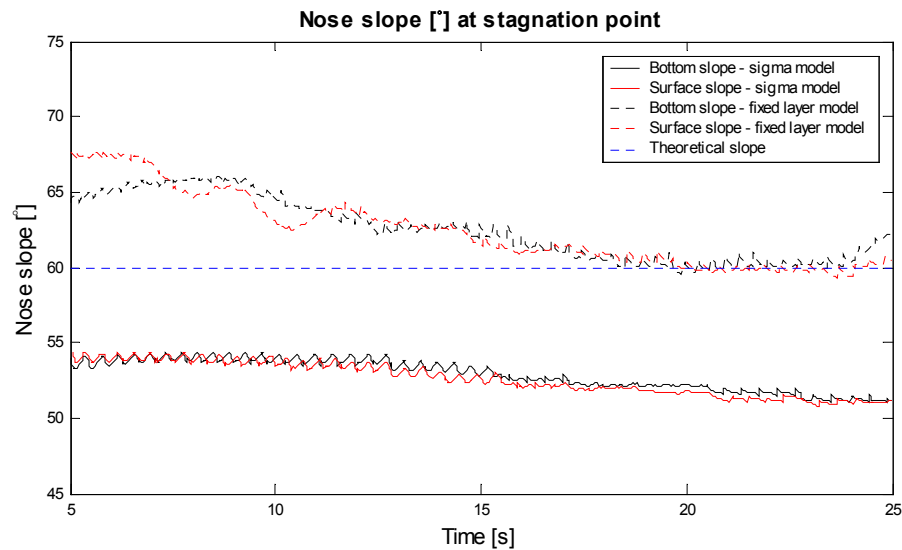


Figure 8-2: *Slope of the bottom and surface nose to the horizontal at the stagnation point (Hydrodynamic sigma layer model, fixed layer model (cf. section 7.2) and theoretical results).*

8.3 Intrusive gravity current - sigma layer model

8.3.1 Model set-up

Again, the model set-up is taken the same as with the fixed layer model. For the exact set-up see section 7.3.1.

When time starts running, a baroclinic pressure gradient will induce gravity currents, moving the fluid on the left-hand side of the basin to intrude between the layers on the right. Furthermore, the density fronts at the surface and the bottom will move to the left.

The theoretical intrusion speed amounts to $u_i = 0,110 \text{ ms}^{-1}$. The theoretical intruding layer thickness h_i is to 1.39 m.

8.3.2 Results

With this model a computation is made using the previously described set-up. The results of this run can be found in Figure 8-17 to 8-20.

Figure 8-18 shows somewhat smoother results in the density distribution around the fronts moving in the negative direction, compared to the previously used model. Also, it can be observed that the 1005 kg m^{-3} -isoline remains almost perfectly horizontal, whereas in the previous results this is clearly not the case. Figure 8-19 shows somewhat higher maximum vertical velocities in the intruding front. (For example, at $t = 45 \text{ s}$ the maximum vertical velocity in the fixed layer model is less than 0.0025 m s^{-1} , compared to less than 0.035 m s^{-1} in the sigma layer model.) Otherwise, the results look similar (in pattern) to the results in section 7.3.

8.3.3 Comparison with theory and fixed layer model

Again, to get be able to compare the results with the available theory and with the results of section 7.3, the intrusion speed and the intruding layer thickness are determined (cf. Table 8-2).

Table 8-2: *Intrusion speed and layer thickness of the intrusive gravity current case. (For the hydrodynamic sigma model and the pressure corrected fixed layer model.)*

	Intrusion speed u_i (m s^{-1})	Intruding layer thickness h_i (m)
Sigma layer model	0.104	1.6
Fixed layer model	0.138	1.6
Theoretical	0.110	1.39

The intrusion speed of the sigma layer model fits well with the theoretical prediction, being only slightly lower. The small difference is most likely caused by numerical diffusion, which lowers the propagation speed.

The intruding layer thickness corresponds well with that of the fixed layer model. The deviation from the theoretically expected value can also be explained by numerical diffusion. Using a finer grid should diminish this effect.

8.4 Buoyant momentum jet

8.4.1 Model set-up

The same set-up as in chapter 7 is chosen to be able to make a comparison (see section 7.4.1). Again, the Courant number is taken just lower than one. Originally the model did not have a turbulence model. However, to increase stability, a viscous term with constant viscosity is added to the horizontal momentum equations. The viscosity is taken to be $\nu = 0.001 \text{ m}^2 \text{ s}^{-1}$.

Instability was especially seen in the water level causing the runs to ‘explode’. It is tried to switch of the water level gradient terms in the equation, hoping for a stable and still acceptable result. However this attempt only showed the importance of the barotropic pressure term in the horizontal momentum, working as a driving force of the wakes present above the outfall.

An overview of the runs made is given in Table 7-3. The column ‘new value’ represents the value of a parameter that is changed compared to the reference run.

8.4.2 Results

The results of the runs made with the sigma-layered hydrodynamic model presented in this chapter can be found in Figure 8–21 to 8–30 in the back of the report.

This program improves the stability of the results compared to the hydrostatic model. This was expected after the results of the pressure-corrected simulations in section 7.4, which were more stable than the hydrostatic results. However the results of the pressure-corrected model differ from the results of this model. Generally two differences can be seen:

- The jets attach more to the boundaries. This can be the bottom (as in run 1, 2 and 5) or the vertical boundary (run 3 and 4). In case of attachment to the bottom this is only in the first meters after the outfall, after this (within 10 m) the warmer water goes to the surface as expected.
- The jets develop slower after the start of the discharge (cf. run 1 to 5).

Furthermore, in some other aspect the results are not as expected. For example in run1 the direction of the wake above the outfall changes (at around $t = 600 \text{ s}$), which seems physically incorrect. Also in the dynamic pressure there are relatively large oscillations (cf. Figure 8-24; $t = 40 \text{ s}$). This can probably be explained by instabilities caused by the explicit free surface handling. Much more primitive turbulence handling than in the model used in chapter 7 plays a role in this as well.

Another important factor causing unstable results is the fact that this model does not use the weakly reflective alpha boundary used in the pressure corrected model (right hand open boundary). This means that the outgoing short waves introduced in a cold start will be reflected at the boundaries and stay within the computational domain. Only the low constant viscosity and the numerical diffusion introduce slight damping.

Changing the ambient flow velocity shows again the enormous influence this has on the stability of the flow.

- Making the ambient velocity 0.0 m s^{-1} (instead of the previously used value of 0.01 m s^{-1}) causes the flow to attach to the vertical boundary, which was not the case when the small ambient flow was present. In the runs with the pressure-corrected model, the differences in results were less rigorous, with the stagnation point at the surface moving a little to the vertical boundary.
- Simulating with an ambient flow of 0.1 m s^{-1} , the warmer jet water eventually comes to the surface in the sigma layered runs. In the run with the model of chapter 7 this was not the case. There a stable layer attached to the bottom, staying there in the rest of the computational domain.

8.5 Pressure interpolation

One of the features of this program is the ability to use less pressure layers than velocity layers. This can save a fair amount of computer time, which increases exponentially with the number of pressure points. In this section, it is examined if using less pressure layers still yields stable results. This is done for the lock exchange problem, the intrusive gravity current and two buoyant momentum jets.

8.5.1 Lock Exchange

The first problem that is computed with less pressure layers is the lock exchange, also used in sections 7.2 and 8.2. This time only five pressure layers are used with again 25 layers for the velocity. The other input is unchanged compared to the set-up described in section 8.2.

As can be seen from Figure 8-3, the results are quite good. The difference between using 25 or 5 layers is hardly noticeable.

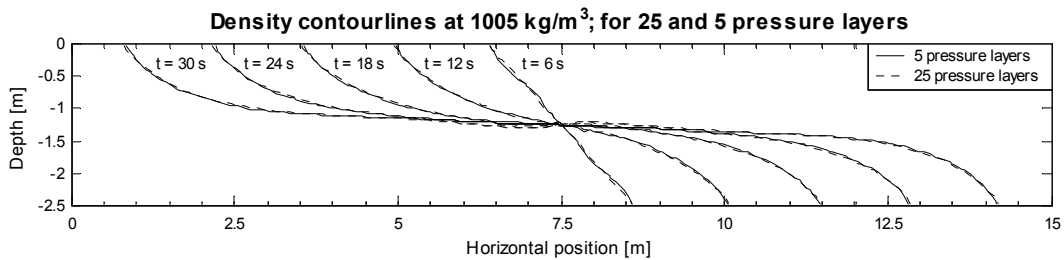


Figure 8-3: *Density contour lines for the Lock Exchange case at $t = 6; 12; 18; 24; 30$ s for both 25 and 5 pressure layers.*

Using only three pressure layers the results deteriorate slightly, especially at half the depth where the contour lines are more or less horizontal with 25 pressure layers (cf. Figure 8-4). The fronts are still reasonably smooth with a slightly higher front slope.

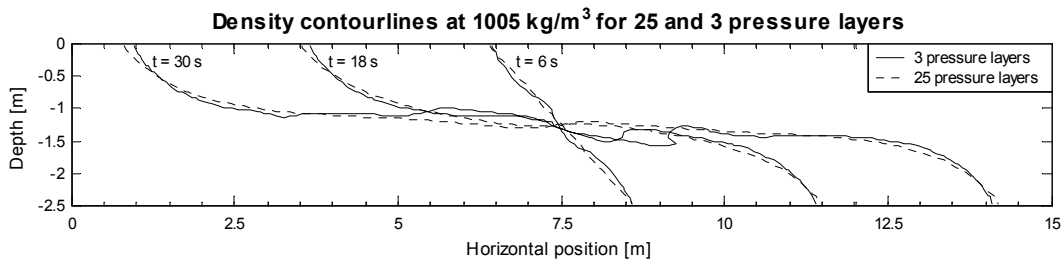


Figure 8-4: *Density contour lines for the Lock Exchange case at $t = 6; 12; 18; 24; 30$ s for both 25 and 3 pressure layers.*

With only two pressure layers, the simulation explodes before $t = 10$ s. This is probably caused by instabilities in the water level.

8.5.2 Intrusive gravity current

The second case that is used is the intrusive gravity current from sections 7.3 and 8.3. Only five pressure layers are used with again 25 layers for the velocity. The other input is unchanged compared to the set-up described in section 8.2.

Also here the results are quite good (cf. Figure 8-3). The difference between using 25 or 5 layers is hardly noticeable in the fronts moving in the positive direction.

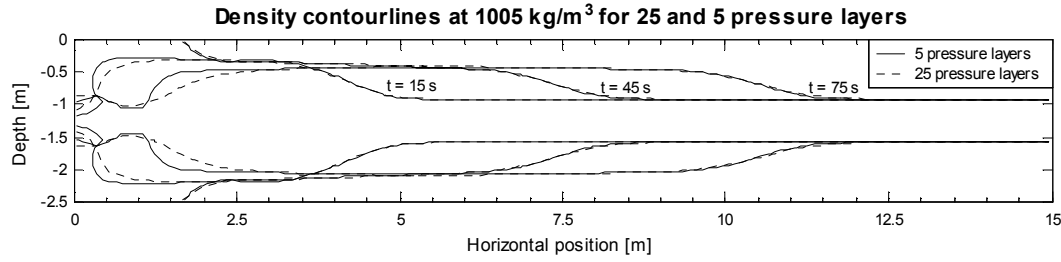


Figure 8-5: *Density contour lines for the Intrusive Gravity Current case at $t = 15; 45; 75$ s for both 25 and 5 pressure layers.*

With only three layers the computation stops before $t = 45$ s. However, the results before this still look quite good (cf. Figure 8-6). Probably oscillations in the water level have caused the instability in the simulation.

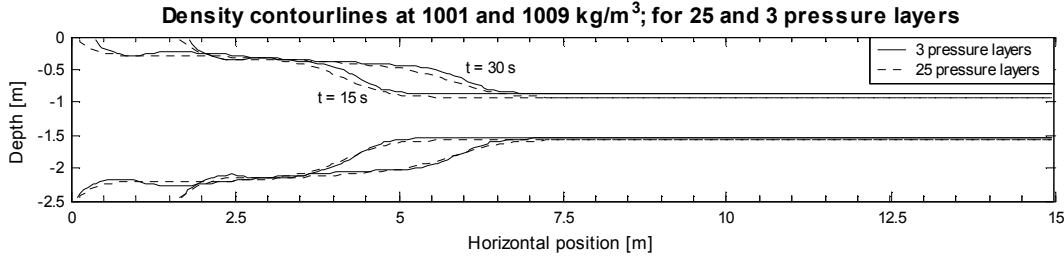


Figure 8-6: *Density contour lines for the Intrusive Gravity Current case at $t = 15; 45; 75$ s for both 25 and 3 pressure layers.*

8.5.3 Buoyant momentum jets - run1

The reference model (run1) from chapter 7 and 8 is also used to test the effect of using less pressure layers. While the original set-up used 20 pressure and velocity layers, the model also gave results with 10 and 7 pressure layers. As can be seen from Figure 8-7 to 8-9, the difference in results between the original simulation with 20 layers and the one with 10 layers is negligible. However, with only 7 pressure layers the results differ considerably. The lighter jet water goes to the surface immediately after being discharged, just like in the hydrostatic simulation (cf. Figure 7-21). The front on the other hand looks similar to the front using more pressure layers, as can be seen clearly in Figure 8-8. Also a stable steady state is reached (plotted in Figure 8-9) with a thin layer of lighter water at the free surface.

Using five layers, the simulation explodes around $t = 100$ s.

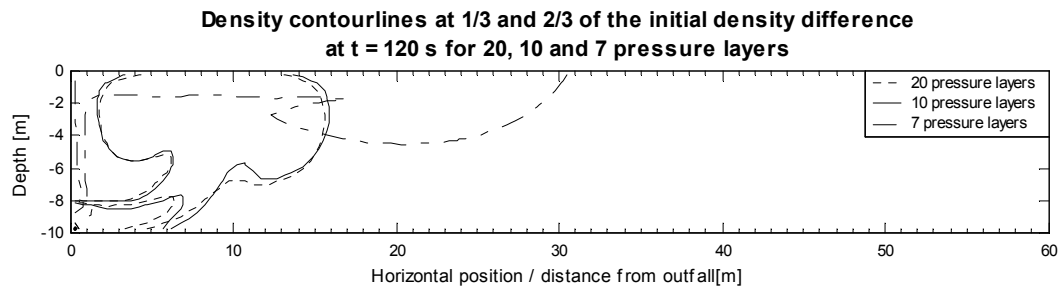


Figure 8-7: *Density contour lines at 1/3 and 2/3 of the initial density difference at $t = 120$ s for 20, 10 and 7 pressure layers. [Run1].*

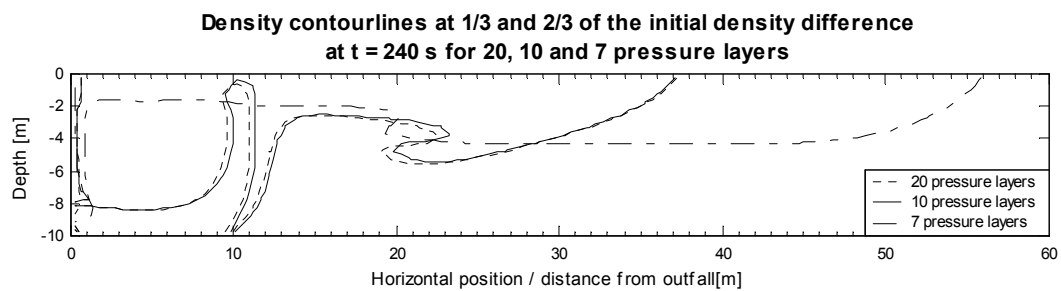


Figure 8-8: *Density contour lines at 1/3 and 2/3 of the initial density difference at $t = 240$ s for 20, 10 and 7 pressure layers. [Run1].*

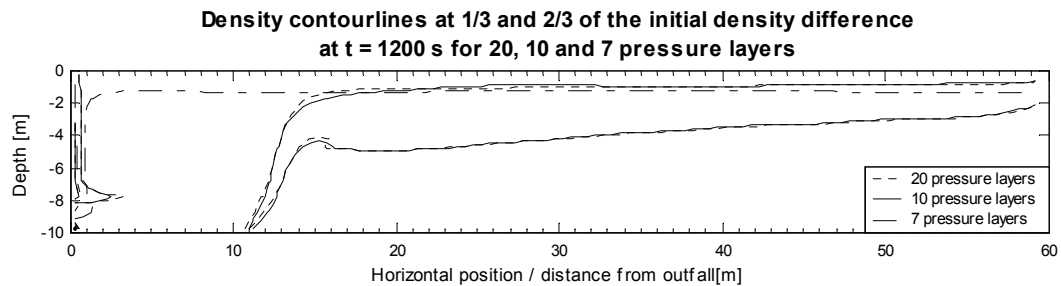


Figure 8-9: *Density contour lines at 1/3 and 2/3 of the initial density difference at $t = 1200$ s for 20, 10 and 7 pressure layers. [Run1].*

8.5.4 Buoyant momentum jets - run2

Another simulation is made, this time with the set-up of run2 of chapter 7 and 8. Run 2 is characterised by a higher outfall velocity than in the previous run (1.0 m s^{-1} compared to 0.5 m s^{-1} in the previous (reference) run). This set-up is taken because the simulated jet has the

highest densimetric Froude number of the outfall of all runs ($F_o = 8.47$). Also, this run yielded an irregular vector field in the hydrostatic computation.

The set-up is tested using 20, 10, 7 and 5 pressure layers and in all cases 20 velocity layers. With 10 pressure layers the computation exploded at around $t = 70$ s. Using only 5 pressure layers, the computation stopped at around $t = 50$ s. Again, presumably instabilities in the water level caused these runs to stop.

In Figure 8-10 the density contour lines of the results at $t = 60$ s can be seen. With 20 and 10 layers, the results are similar in shape. With only 7 pressure layers, the results look different. The lighter jet water flows to the surface immediately after being injected in the ambient water body. This is similar to the results of some hydrostatic simulations. The front of the jet at the free surface however has a shape that does not look like the typical hydrostatic fronts, being less steep.

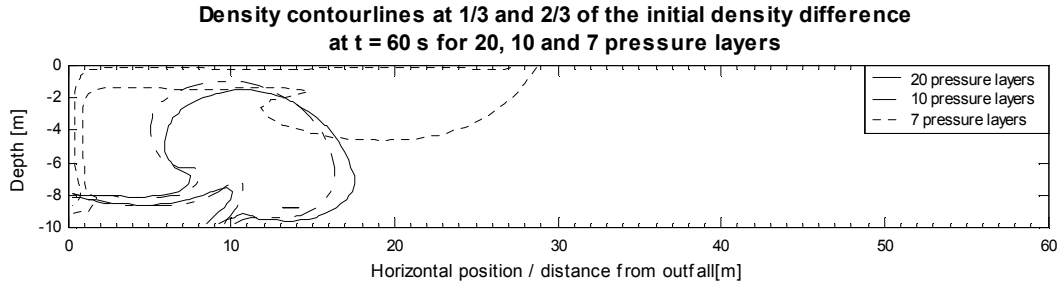


Figure 8-10: *Density contour lines at 1/3 and 2/3 of the initial density difference at $t = 60$ s for 20, 10 and 7 pressure layers. [Run2].*

Figure 8-11 and 8-12 show that the 7-layered run yields a stable steady state solution with the warmer water confined to a thin layer at the free surface. However, these results differ considerably from the results with 20 pressure layers, especially in the near-field.

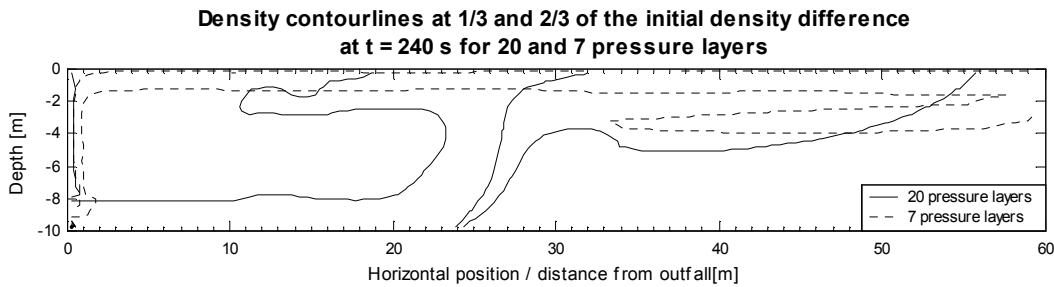


Figure 8-11: *Density contour lines at 1/3 and 2/3 of the initial density difference at $t = 240$ s for 20 and 7 pressure layers. [Run2].*

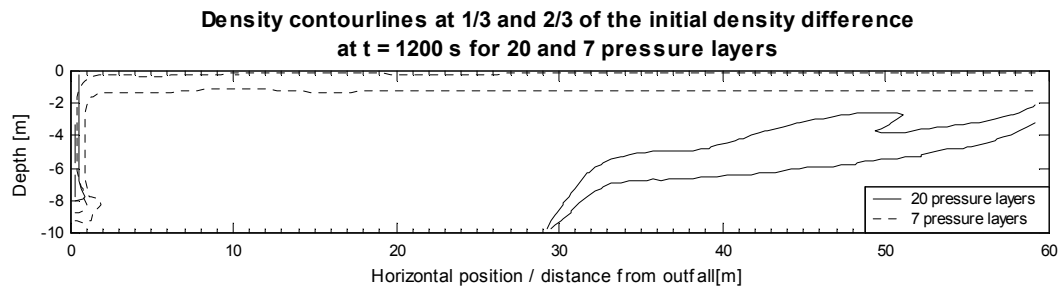


Figure 8-12: *Density contour lines at 1/3 and 2/3 of the initial density difference at $t = 1200$ s for 20 and 7 pressure layers. [Run2].*

Despite being physically incorrect in the near-field the 7-layered simulation yielded a stable result. When we compare this to the irregular hydrostatic result in chapter 7 (cf. Figure 7-28) this certainly is an improvement.

Summarised, diminishing the number of pressure layers has a considerable effect on the near-field of the jet, whereas in the surface front of the jet, the influence is hardly noticeable. The smaller relevant length scales in the near-field regions can explain this. This region is highly non-hydrostatic with pressures varying over small distances, whereas in the density front, the non-hydrostatic pressure varies more smoothly. When the thickness of the pressure layer becomes higher than the vertical dimension of the outfall, the results will change considerably.

Compared to similar computations with hydrostatic models, the use of even a few pressure layers improves the results in the sense that a stable solution is reached where the warm water is confined to the upper layer. This was not the case in the hydrostatic model.

8.6 Conclusions

The first part of this chapter deals with the extension of a sigma layered non-hydrostatic 2DV program capable of uncoupling the number of pressure and velocity layers. This program is extended with a transport equation for heat and a baroclinic pressure term, which makes it possible to accommodate a variable density and thus buoyancy induced flow. Verification of the model is done by simulating of the lock exchange case and the intrusive gravity current case also used in chapter 7. Comparison to theory yielded good results, in particular with respect to the front (intrusion) speed. The angle of the density front in the lock exchange case was slightly lower than expected on theoretical grounds.

Another test involved the simulation of the same buoyant jets as used in chapter 7. Generally two differences can be seen:

- The jets attach more to the boundaries (bottom or vertical boundary). In case of attachment to the bottom, this is only in the first meters after the outfall. Within 10 m the warmer water goes to the surface as expected.
- The jets develop slower after the start of the discharge.

- Relatively large oscillations in the dynamic pressure exist.

Much more primitive turbulence handling than in the model used in chapter 7 might play role in the first two differences. These oscillations in the dynamic pressure might be explained by explicit free surface handling.

Buoyant jets with ambient (co-) flow cause the same instabilities as in the program used in chapter 7.

The second part of this chapter dealt with the use of pressure interpolation and the effect of this on the stability of the results. With respect to the simulation of purely density driven flow it can be concluded that diminishing the number of pressure layers yields accurate results. Even the use of only three pressure layers (compared to 25 velocity layers) provides a great improvement compared to the hydrostatic results in chapter 7.

In case of submerged buoyant jets (a combination of density driven flow and forced convection) the number of pressure layers can be decreased with virtually no negative influence if the pressure layer thickness remains smaller or equal to the vertical dimension of the outlet.

Further lowering the number of pressure layers causes the near-field to become physically incorrect with the jet water immediately going to the free surface. However, unlike in the hydrostatic computations, this does not result in an irregular velocity vector field. The far-field remains stable. This means that using only a few pressure layers is certainly an important improvement compared to hydrostatic modelling.

In all the pressure-interpolated runs, further diminishing the number of pressure layers caused the runs to ‘explode’ due to oscillations in the water level. This suggest that implementing the water level implicitly in the model (now: explicit) will make it possible to further decrease the number of pressure layers.

9 Conclusions and recommendations

9.1 Conclusions

A jet simulation (Point Beach Power Station case) in chapter 5 showed that the far-field results of a non-hydrostatic model can be similar to those of a non-hydrostatic model. Especially the jet trajectory, the centreline dilution and the layer thickness of the plume using the hydrostatic program Delft3D-FLOW where in good agreement with non-hydrostatic results from CFX. This confirms the idea that a stable (but not physically correct) near-field with the buoyant jet water coming to the surface is enough to provide accurate results in the far-field.

Two-dimensional (2DV) simulations in Delft3D and CFX (chapter 6) show that the near field of a hydrostatic simulated jet is indeed physically incorrect. This is shown by unrealistically high vertical velocities located where the jets enter the ambient water body and an overestimated angle of the density fronts. The jet trajectory should start horizontally, gradually moving to the surface, because the initial momentum flux is in the horizontal direction. Unlike in the non-hydrostatic CFX results, this is not the case in the hydrostatic Delft3D results.

It is also shown that hydrostatic simulation can lead to results where the (positively) buoyant jet water does not come to the surface.

To further investigate the effect of the hydrostatic pressure assumption, more simulations are made using a program that can switch between a hydrostatic and a non-hydrostatic mode by switching the pressure correction off and on. Simulations of two purely buoyancy driven cases (the lock exchange problem and an intrusive gravity current) showed the improvements resulting from non-hydrostatic simulation (compared to hydrostatic simulation):

- More realistic (lower) vertical velocities in the density fronts. Also a much smoother spatial change in velocity.
- A better agreement between the theoretically predicted and simulated density fronts.

The main differences were found at locations with high horizontal pressure gradients (density fronts). This was also where the highest dynamic pressures were located. It should also be noted that the horizontal front speeds were almost unaffected by the pressure correction.

Simulation of various buoyant jets showed that non-hydrostatic simulation is necessary to prevent physically incorrect results. Again, only non-hydrostatic simulations yielded correct jet trajectories, starting horizontal near the outfall (due to a horizontal momentum flux) and gradually moving to the surface. Furthermore, the same improvements as in the purely buoyancy driven cases were noticeable here: more realistic (lower) vertical velocities with a smoother spatial distribution and a more realistic (less steep) angle of the density front.

Attachment of the jet to the bottom in the hydrostatic runs was always accompanied by a high densimetric Froude number at the outfall ($F_0 > 5$) or a strong ambient (co-) flow. In the case of the strong ambient current, non-hydrostatic simulation did not improve the situation. In the case of a high densimetric Froude number, non-hydrostatic simulation made the warmer water to come

to the surface and form a stable layer. This is an important improvement compared to non-hydrostatic simulation.

The last part of the study (chapter 8) was to investigate if by diminishing the number of pressure layers the desired stable non-hydrostatic results can still be obtained. Therefore a sigma layered non-hydrostatic 2DV program capable of uncoupling the number of pressure and velocity layers was extended with a transport equation for heat and a baroclinic pressure term. This makes it possible to accommodate a variable density and thus buoyancy induced flow.

After verification of the model, simulations showed that, with respect to the simulation of purely density driven flow, it can be concluded that diminishing the number of pressure layers yields accurate results. Even the use of only three pressure layers (compared to 25 velocity layers) provides a great improvement compared to hydrostatic results.

In case of submerged buoyant jets (a combination of density driven flow and forced convection) the number of pressure layers can be decreased with virtually no negative influence if the pressure layer thickness remains smaller or equal to the vertical dimension of the outlet.

Further lowering the number of pressure layers causes the near-field to become physically incorrect with the jet water immediately going to the free surface. However, unlike in the hydrostatic computations, this does not result in an irregular velocity vector field. The far-field remains stable. This means that using only a few pressure layers is certainly an important improvement compared to hydrostatic modelling.

Summarising, the conclusions of this study concerning the effect of non-hydrostatic modelling compared to hydrostatic modelling, are:

- Non-hydrostatic modelling of buoyant jets yields physically more realistic results in the near-field, i.e., more realistic (lower) vertical velocities, a smoother spatial distribution of the velocity and more realistic angles of a density fronts.
- The results of the simulation of buoyant jets in the near-field improve in that the warmer jet water comes to the surface even if the densimetric Froude number at the outfall is high ($F_0 > 5$; low initial density difference, high initial jet velocity). This was not always the case in hydrostatic results.

The conclusions of this study concerning the effect of diminishing the number of pressure layers are:

- In case of submerged buoyant jets, the number of pressure layers can be decreased with virtually no negative influence if the pressure layer thickness remains smaller or equal to the vertical dimension of the outfall.
- A pressure layer thickness larger than the vertical dimension of the outfall causes the near-field to become physically incorrect with the jet water immediately flowing to the free surface (no wake, no horizontal start of the trajectory). However, unlike in the hydrostatic computations, this does not result in an irregular velocity vector field or attachment to the bottom. This means that using only a few pressure layers is certainly an important improvement compared to hydrostatic modelling.

9.2 Recommendations

The following recommendation, mainly focusing on possible extensions of the sigma-layered model used in this research, are made:

- Extend the models to accommodate three-dimensional problems. As most density currents are three-dimensional, better testing on realistic problems will be possible.
- Investigate the effects of using non-hydrostatic simulation only confined to certain regions (e.g. the near-field).
- In all pressure-interpolated runs, further diminishing the number of pressure layers caused the runs to ‘explode’ due to oscillations in the water level. This suggest that implementing an implicit free surface treatment (now: explicit) in the sigma layer model might improve stability and makes it possible to further decrease the number of pressure layers.
- Making the advection and the baroclinic pressure term implicit as well, will remove the strict time step limitations.
- Apply a faster (iterative) solver.
- Implement the possibility of using a weakly reflective alpha boundary.
- Implement a more advanced turbulence model then is used now, as this is of great importance for more physically correct modelling of jets.

References

- AEA Technology (1999). “*Manual CFX-4.4.*”
- Batchelor, G.K. (1967). “*An introduction to fluid dynamics.*” Cambridge University Press, London.
- Bijvelds, M.D.J.P. (1995). “*Towards numerical modelling of cooling water discharges and related phenomena.*” Masters thesis, Delft University of Technology, Dept. of Civil Engineering, Delft.
- Bijvelds, M.D.J.P. (2001). “*Numerical modelling of estuarine flow over steep topography.*” Ph.D. thesis, Delft University of Technology, Delft.
- Bird, R.B., Steward, W.E. & Lightfoot, E.N. (1960). “*Transport phenomena.*” Wiley International Edition, New York.
- Dunsbergen, D.W. (1994). “*Particle models for transport in three-dimensional shallow water flow.*” Report No. 94-8, Delft University of Technology, Dept. of Civil Engineering, Delft.
- Eckart, C. (1958). “*Properties of water, Part II. The equation of State of water and sea water at low temperatures and pressures.*” American Journal of Science, 256, 225-240.
- Gross, E. S., Bonaventura, L. & Rosatti, G. (2002). “*Consistency with continuity in conservative advection schemes for free-surface models.*” International Journal for Numerical Methods in Fluids, John Wiley & Sons, Vol. 38, 307-327.
- Jirka, G.H., Doneker, R.L. & Hinton, S.W. (1996). “*User’s manual for CORMIX: A hydrodynamic mixing zone model and decision support system for pollutant discharges into surface waters.*” DeFrees Hydraulic Laboratory, Cornell University, New York.
- Lamb, H. (1974). “*Hydrodynamics.*” Cambridge University Press, London.
- Miller, D.S. & Brighthouse, B.A. (1984). “*Thermal discharges; a guide to power and process plant cooling water discharges into rivers, lakes and seas.*” BHRA, Cranfield.
- Perry, R.H. & Chilton, C.H. (1973). “*Chemical engineers’ handbook.*” McGraw-Hill Kogakusha, Ltd., Tokyo.

- Reeuwijk, M. van (2002). “*Efficient simulation of non-hydrostatic free-surface flow; Wave simulation with interpolation of the vertical pressure profile.*” Masters thesis, Delft University of Technology, Dept. of Civil Engineering, Delft.
- Serrin, J. (1959). “*Mathematical principles of classical fluid mechanics.*” in: Handbuch der Physik, Vol. 8, Springer-Verlag, Berlin.
- Turner, J. S. (1973). “*Buoyancy effects in fluids.*” Cambridge University Press, Cambridge.
- WL | delft hydraulics. (1999) “*Delft3D-FLOW user manual*” Version 3.05.
- Zijlema, M. (1998). “*TRIWAQ - Three dimensional incompressible shallow flow model; Technical documentation, version 1.0.*” Rijkswaterstaat / RIKZ.

List of main symbols

Roman symbols

b	width.....	[m]
C	Chézy bottom friction coefficient	$[\text{m}^{1/2} \text{s}^{-1}]$
C_f	Courant number.....	[-]
C_p	specific heat at constant pressure	$[\text{J kg}^{-1} \text{K}^{-1}]$
d	water depth	[m]
e	2.71828...	[-]
f	Coriolis parameter	$[\text{rad s}^{-1}]$
F	densimetric Froude number.....	[-]
$\mathbf{g}$	gravitational acceleration vector	$[\text{m s}^{-2}]$
g	magnitude of gravitational acceleration ($9.81 \dots \text{m s}^{-2}$)	$[\text{m s}^{-2}]$
g'	acceleration due to buoyancy	$[\text{m s}^{-2}]$
$\mathbf{G}$	body force per unit mass	$[\text{m s}^{-2}]$
h	static enthalpy.....	$[\text{J kg}^{-1}]$
h	height.....	[m]
h_i	intruding layer thickness	[m]
H	water depth	[m]
J_o	kinematic buoyancy flux	$[\text{m}^2 \text{s}^{-3}]$
k	turbulent kinetic energy.....	$[\text{m}^2 \text{s}^{-2}]$
L	length.....	[m]
M_o	kinematic momentum flux	$[\text{m}^4 \text{s}^{-2}]$
$\mathbf{n}$	unit vector normal to water table.....	[-]
p	fluid pressure.....	$[\text{Pa} = \text{N m}^{-2}]$
p_{atm}	atmospheric pressure	[Pa]
p_h	hydrostatic part of fluid pressure.....	[Pa]
q	hydrodynamic part of fluid pressure	[Pa]
Q	discharge	$[\text{m}^3 \text{s}^{-1}]$
Re	Reynolds number	[-]
Ri	Richardson number	[-]
$\mathbf{r}$	position vector.....	[m]
s	salinity	[ppt]
S	entropy.....	$[\text{J kg}^{-1} \text{K}^{-1}]$
t	time.....	[s]
T	temperature.....	[K] or $^{\circ}\text{C}$
$\mathbf{u}$	velocity vector.....	$[\text{m s}^{-1}]$

u	velocity in x -direction.....	[m s ⁻¹]
u_f	front speed	[m s ⁻¹]
u_i	intrusion speed.....	[m s ⁻¹]
v	velocity in y -direction.....	[m s ⁻¹]
w	velocity in z -direction.....	[m s ⁻¹]
x	Cartesian horizontal co-ordinate.....	[m]
y	Cartesian horizontal co-ordinate.....	[m]
z	Cartesian vertical co-ordinate.....	[m]

Greek symbols

α	thermal diffusivity	[m ² s ⁻¹]
α	reflection coefficient - water level boundary.....	[s ²]
α	reflection coefficient - velocity boundary	[s]
α_0, α_1	profile coefficients.....	[-]
β	coefficient of thermal expansion	[K ⁻¹]
δ	density interface layer thickness.....	[m]
Δp	deviation from static pressure.....	[Pa]
Δt	time step	[s]
$\Delta \rho$	deviation from reference density	[kg m ⁻³]
$\Delta x, \Delta z$	grid size in respectively x - and z -direction	[m]
ε	turbulent energy dissipation rate	[m ² s ⁻³]
Φ	viscous dissipation.....	[J m ⁻³ s ⁻¹]
λ	thermal conductivity.....	[W m ⁻¹ K ⁻¹]
μ	dynamic viscosity (or briefly: viscosity)	[Pa s]
ν	kinematic viscosity	[m ² s ⁻¹]
ρ	density	[kg m ⁻³]
σ	vertical co-ordinate in sigma co-ordinate system.....	[-]
σ_t	turbulent Prandtl number	[-]
$\bar{\tau}^{(t)}$	Reynolds stress tensor	[kg m ⁻¹ s ⁻²]
ω	transformed vertical velocity	[m s ⁻¹]
Ω	angular velocity vector	[rad s ⁻¹]
Ω	magnitude of angular velocity	[rad s ⁻¹]
ζ	water level elevation.....	[m]

Operators

div	divergence operator	[m ⁻¹]
D/Dt	substantial or material derivative	[s ⁻¹]
grad	gradient operator	[m ⁻¹]
∇^2	Laplacian operator	[m ⁻²]

Subscripts

0	in equilibrium or initial state or reference value
<i>a</i>	ambient
<i>k</i>	vertical index of velocity layer
<i>m</i>	horizontal index of grid point
<i>o</i>	at outfall

Superscripts

'	transformed or deviation from time-smoothed value
–	ensemble averaged / time-smoothed
(<i>eff</i>)	effective (molecular plus turbulent)
<i>n</i>	index of time step
(<i>t</i>)	turbulent

Appendices

Appendix A: Simplification of Continuity Equation

By making some reasonable assumptions concerning the transport of dissolved mass and heat, it can be shown that the continuity equation simplifies to

$$\text{div } \mathbf{u} = 0 \quad (\text{A.1})$$

The following assumptions are used:

- By assuming the caloric equation $dh = C_p dT - \rho^{-1} dp$, and excluding phase transitions, heat conduction and viscous dissipation in the enthalpy equation, the enthalpy equation is simplified to (cf. Appendix B):

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \text{grad } T = 0 \quad (\text{A.2})$$

- In a more or less similar way we may state that, due to neglect of molecular diffusion, the transport equation for dissolved mass is simplified to

$$\frac{\partial c}{\partial t} + \mathbf{u} \cdot \text{grad } c = 0 \quad (\text{A.3})$$

- The fluid is incompressible ($\partial \rho / \partial p = 0$). Therefore the equation of state is only dependent on the concentration of dissolved matter and the temperature:

$$\rho = \rho(c, T) \quad (\text{A.4})$$

We start with the continuity equation in its most general form (cf. Equation 2.2):

$$\frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \text{grad } \rho + \rho \text{div } \mathbf{u} = 0 \quad (\text{A.5})$$

Using the chain rule this can be written as:

$$\left(\frac{\partial c}{\partial t} + \mathbf{u} \cdot \text{grad } c \right) \frac{\partial \rho}{\partial c} + \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \text{grad } T \right) \frac{\partial \rho}{\partial T} + \rho \text{div } \mathbf{u} = 0 \quad (\text{A.6})$$

Substituting Equations (A.2) and (A.3) in (A.6) yields:

$$\rho \text{div } \mathbf{u} = 0 \Rightarrow \text{div } \mathbf{u} = 0 \quad (\text{A.7})$$

NOTE: These assumptions are not necessarily used in the other equations described.

Appendix B: Energy equation

For non-isothermal flow, an equation governing the transport of heat is needed. In this appendix a suitable advection-diffusion equation for the temperature will be derived.

Let us start with the entropy equation [Serrin 1954, p. 239]:

$$\rho T \frac{DS}{Dt} = \Phi + \text{div}(\lambda \text{grad } T) \quad (\text{B.1})$$

where S is the entropy [$\text{J kg}^{-1} \text{K}^{-1}$], Φ is the viscous dissipation [$\text{J m}^{-3} \text{s}^{-1}$], λ is the thermal conductivity [$\text{W m}^{-1} \text{K}^{-1}$] and T is the temperature [K].

The thermodynamic equation is represented by [Perry & Chilton 1973, p. 4-45]:

$$dh = TdS + \rho^{-1}dp \quad (\text{B.2})$$

with the static enthalpy h [J kg^{-1}].

Substitution of Equation (B.2) into Equation (B.1) yields:

$$\rho \frac{Dh}{Dt} = \frac{Dp}{Dt} + \Phi + \text{div}(\lambda \text{grad } T) \quad (\text{B.3})$$

The caloric equation is [Perry & Chilton 1973, 4-46]:

$$dh = C_p dT + \left[\rho^{-1} - T \left(\frac{\partial \rho^{-1}}{\partial T} \right)_p \right] dp \approx C_p dT + \rho^{-1} dp \quad (\text{B.4})$$

where C_p is the specific heat at constant pressure [$\text{J kg}^{-1} \text{K}^{-1}$].

The justification of the approximation in the above equation (B.4) lies in the following. The equation of state for an incompressible liquid ($\partial \rho / \partial p = 0$) is [AEA Technology 1999]:

$$\rho(T) = \rho_0 (1 - \beta(T - T_0)) \quad (\text{B.5})$$

which is equivalent with

$$\rho^{-1} - \rho_0^{-1} = \rho^{-1} \beta (T - T_0) \quad (\text{B.6})$$

where β is the coefficient of thermal expansion [K^{-1}]. Since $\beta(T - T_0)$ is small compared to 1, we may approximate Equation (B.6) to:

$$\rho^{-1} - \rho_0^{-1} = \rho_0^{-1} \beta (T - T_0) \quad (\text{B.7})$$

Hence, $\partial \rho^{-1} / \partial T = \rho_0^{-1} \beta$. Since for liquids $\rho_0^{-1} \beta T \ll \rho^{-1}$, we may use the approximation in Equation (B.4).

Substitution of the approximation of Equation (B.4) into Equation (B.3) yields:

$$\rho C_p \frac{DT}{Dt} = \text{div}(\lambda \text{grad} T) + \Phi \quad (\text{B.8})$$

Neglect of the viscous dissipation Φ and substitution of ρ_0 for ρ (Boussinesq approximation) yields the advection-diffusion equation for the temperature:

$$\frac{DT}{Dt} = \alpha \nabla^2 T \quad (\text{B.9})$$

where $\alpha = \lambda / \rho_0 C_p$ is the thermal diffusivity [$\text{m}^2 \text{s}^{-1}$].

Appendix C: Derivation of kinematic boundary conditions

Let us consider the water table as a time varying plane:

$$F(\mathbf{x}, t) = 0 \quad (\text{C.1})$$

The unit vector normal to this plane is:

$$\mathbf{n} = \text{grad } F / |\text{grad } F| \quad (\text{C.2})$$

In a Cartesian co-ordinate system with x and y as horizontal co-ordinates and z as vertical co-ordinate (positive in upward direction), the water table can be expressed as:

$$z = \zeta(x, y, t), \quad (\text{C.3})$$

hence,

$$F(\mathbf{x}, t) = \zeta(x, y, t) - z = 0 \quad (\text{C.4})$$

The normal unit vector can now be expressed as:

$$(n_x, n_y, n_z) = \frac{\left(\frac{\partial \zeta}{\partial x}, \frac{\partial \zeta}{\partial y}, -1 \right)}{\sqrt{\left(\frac{\partial \zeta}{\partial x} \right)^2 + \left(\frac{\partial \zeta}{\partial y} \right)^2 + 1}} \quad (\text{C.5})$$

The flow velocity of the water perpendicular to the water table is:

$$\mathbf{u} \cdot \mathbf{n} = \frac{u \frac{\partial \zeta}{\partial x} + v \frac{\partial \zeta}{\partial y} - w}{\sqrt{\left(\frac{\partial \zeta}{\partial x} \right)^2 + \left(\frac{\partial \zeta}{\partial y} \right)^2 + 1}} \quad (\text{flow velocity} \perp \text{water surface}) \quad (\text{C.6})$$

Here $\hat{\mathbf{x}}$, with Cartesian co-ordinates $\hat{x}, \hat{y}, \hat{z}$ are points on the water table. At time t , the water table velocity, $\hat{\mathbf{u}}$, (not necessarily being equal to the flow velocity $\mathbf{u}$) equals:

$$(\hat{u}, \hat{v}, \hat{w}) = \left(\frac{\partial \hat{x}}{\partial t}, \frac{\partial \hat{y}}{\partial t}, \frac{\partial \hat{z}}{\partial t} \right) = \left(0, 0, \frac{\partial \zeta}{\partial t} \right) \quad (\text{water table velocity}) \quad (\text{C.7})$$

Based on this, the following expression holds:

$$\hat{\mathbf{u}} \cdot \mathbf{n} = \frac{\left(0, 0, -\frac{\partial \zeta}{\partial t} \right)}{\sqrt{\left(\frac{\partial \zeta}{\partial x} \right)^2 + \left(\frac{\partial \zeta}{\partial y} \right)^2 + 1}} \quad (\text{water surface velocity} \perp \text{water surface}) \quad (\text{C.8})$$

If replenishment (precipitation minus evaporation) is neglected, the water velocity perpendicular to the water surface (C.6) is equal to the water table velocity perpendicular to the water table (C.8). Equating Eq. (C.6) to Eq. (C.8) gives:

$$w(x, y, \zeta, t) = \frac{\partial \zeta}{\partial t} + u \frac{\partial \zeta}{\partial x} + v \frac{\partial \zeta}{\partial y} = \frac{D\zeta}{Dt} \quad (\text{C.9})$$

Hence, the vertical water velocity equals the substantial derivative of the water level height. This is the kinematic boundary condition at the surface (i.e., the water table).

The dynamic boundary condition is the condition that, on the water surface, the pressure is equal to the atmospheric pressure (neglecting the surface tension).

For the vertical velocity at the bottom $z = -d(x, y, t)$, the same can be done. This gives:

$$w(x, y, -d, t) = -\frac{\partial d}{\partial t} - u \frac{\partial d}{\partial x} - v \frac{\partial d}{\partial y} = -\frac{Dd}{Dt} \quad (\text{C.10})$$

Assuming that the bottom position does not change in time, this reduces further to:

$$w(x, y, -d) = -u \frac{\partial d}{\partial x} - v \frac{\partial d}{\partial y} \quad (\text{C.11})$$

Appendix D: Depth integrated Continuity Equation

We can write the Continuity Equation $\text{div } \mathbf{u} = 0$ in 3D Cartesian co-ordinates:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (\text{D.1})$$

Integration from $z = -d$ to $z = \zeta$ gives:

$$\int_{-d}^{\zeta} \frac{\partial u}{\partial x} dz + \int_{-d}^{\zeta} \frac{\partial v}{\partial y} dz + w(x, y, \zeta, t) - w(x, y, -d, t) = 0 \quad (\text{D.2})$$

In general Leibniz's rule holds:

$$\frac{d}{dc} \int_{a(c)}^{b(c)} f(c, z) dz = \int_{a(c)}^{b(c)} \frac{\partial f(c, z)}{\partial c} dz + f(c, b) \frac{db(c)}{dc} - f(c, a) \frac{da(c)}{dc} \quad (\text{D.3})$$

Applying this to the continuity equation (D.1) gives:

$$\begin{aligned} \frac{\partial}{\partial x} \int_{-d}^{\zeta} u dz - u \frac{\partial \zeta}{\partial x} + u \frac{\partial(-d)}{\partial x} + \frac{\partial}{\partial y} \int_{-d}^{\zeta} v dz - v \frac{\partial \zeta}{\partial y} + v \frac{\partial(-d)}{\partial y} + \\ w(x, y, \zeta, t) - w(x, y, -d, t) = 0 \end{aligned} \quad (\text{D.4})$$

The kinematic boundary conditions are (cf. Equations (C.9) and (C.10)):

$$w(x, y, \zeta, t) = \frac{\partial \zeta}{\partial t} + u \frac{\partial \zeta}{\partial x} + v \frac{\partial \zeta}{\partial y} \quad (\text{water surface}) \quad (\text{D.5})$$

$$w(x, y, -d, t) = -\frac{\partial d}{\partial t} - u \frac{\partial d}{\partial x} - v \frac{\partial d}{\partial y} \quad (\text{bottom}) \quad (\text{D.6})$$

See Appendix C for the derivation of the kinematic boundary conditions.

Substitution of Equation (D.5) and (D.6) in Equation (D.4) yields:

$$\frac{\partial \zeta}{\partial t} + \frac{\partial d}{\partial t} + \frac{\partial}{\partial x} \int_{-d}^{\zeta} u dz + \frac{\partial}{\partial y} \int_{-d}^{\zeta} v dz = 0 \quad (\text{D.7})$$

Assuming that the position of the bottom does not change in time, this simplifies to:

$$\frac{\partial \zeta}{\partial t} + \frac{\partial}{\partial x} \int_{-d}^{\zeta} u dz + \frac{\partial}{\partial y} \int_{-d}^{\zeta} v dz = 0 \quad (\text{D.8})$$

Appendix E: Sigma transformation

The sigma transformation starting from a Cartesian frame of reference with one horizontal dimension x [m] and one vertical dimension z [m], is expressed by:

$$\begin{aligned} t &= t' \\ x &= x' \\ z &= \sigma H(x, t) + \zeta(x, t) \Rightarrow \sigma = \frac{z - \zeta(x, t)}{H(x, t)} \end{aligned} \quad (E.1)$$

with:

$$\begin{aligned} H(x, t) &= \zeta(x, t) + d(x, t) \\ -d(x) &\leq z \leq \zeta(x, t) \Rightarrow -1 \leq \sigma \leq 0 \end{aligned}$$

in which t is the time [s], σ is the transformed vertical co-ordinate [-], H is the total water depth [m], ζ is the water level elevation [m] and d is the bottom depth [m]. Both ζ and d are relative to an arbitrary horizontal reference line.

With use of the chain rule, the first order time and spatial derivatives in Cartesian co-ordinates can be expressed in terms of sigma co-ordinates.

$$\begin{aligned} \frac{\partial}{\partial x} &= \frac{\partial}{\partial t'} \frac{\partial t'}{\partial x} + \frac{\partial}{\partial x'} \frac{\partial x'}{\partial x} + \frac{\partial}{\partial \sigma} \frac{\partial \sigma}{\partial x} = \frac{\partial}{\partial x'} + \frac{\partial \sigma}{\partial x} \frac{\partial}{\partial \sigma} \\ \frac{\partial}{\partial z} &= \frac{\partial}{\partial t'} \frac{\partial t'}{\partial z} + \frac{\partial}{\partial x'} \frac{\partial x'}{\partial z} + \frac{\partial}{\partial \sigma} \frac{\partial \sigma}{\partial z} = \frac{\partial \sigma}{\partial z} \frac{\partial}{\partial \sigma} \\ \frac{\partial}{\partial t} &= \frac{\partial}{\partial t'} \frac{\partial t'}{\partial t} + \frac{\partial}{\partial x'} \frac{\partial x'}{\partial t} + \frac{\partial}{\partial \sigma} \frac{\partial \sigma}{\partial t} = \frac{\partial}{\partial t'} + \frac{\partial \sigma}{\partial t} \frac{\partial}{\partial \sigma} \end{aligned} \quad (E.2)$$

With reference to the Cartesian co-ordinate system, a sigma-plane can be described by the function z_σ :

$$z = z_\sigma(x, t) = \sigma \cdot H(x, t) + \zeta(x, t), \quad (E.3)$$

where σ has a fixed value.

The function z_σ represents a sigma iso-line; a line in the Cartesian co-ordinate system with a constant sigma. The line has a slope related to a horizontal line s_x described by

$$s_x = \left(\frac{\partial z}{\partial x} \right)_\sigma = \frac{\partial z_\sigma}{\partial x} = \frac{\partial \zeta}{\partial x} + \sigma \frac{\partial H}{\partial x} \quad (E.4)$$

The first order derivative of z_σ with respect to σ is known:

$$\frac{\partial z_\sigma}{\partial \sigma} = \frac{dz_\sigma}{d\sigma} = H \quad (E.5)$$

Also:

$$\frac{\partial}{\partial \sigma} = \frac{\partial}{\partial z_\sigma} \frac{\partial z_\sigma}{\partial \sigma} = H \frac{\partial}{\partial z_\sigma} \quad (\text{E.6})$$

Furthermore the Cartesian partial derivatives of σ can be written as:

$$\begin{aligned} \frac{\partial \sigma}{\partial x} &= -\frac{1}{H} \left(\frac{\partial \zeta}{\partial x} + \sigma \frac{\partial H}{\partial x} \right) = -\frac{1}{H} \frac{\partial z_\sigma}{\partial x} \\ \frac{\partial \sigma}{\partial z} &= \frac{1}{H} \\ \frac{\partial \sigma}{\partial t} &= -\frac{1}{H} \left(\frac{\partial \zeta}{\partial t} + \sigma \frac{\partial H}{\partial t} \right) = -\frac{1}{H} \frac{\partial z_\sigma}{\partial t} \end{aligned} \quad (\text{E.7})$$

Substituting the expressions (E.6) and (E.7) for the Cartesian derivatives of σ into the expressions (E.2) yields:

$$\begin{aligned} \frac{\partial}{\partial x} &= \frac{\partial}{\partial x'} - \frac{1}{H} \frac{\partial z_\sigma}{\partial x} \frac{\partial}{\partial \sigma} = \frac{\partial}{\partial x'} - \frac{\partial z_\sigma}{\partial x} \frac{\partial}{\partial z_\sigma} \\ \frac{\partial}{\partial z} &= \frac{1}{H} \frac{\partial z_\sigma}{\partial z} \frac{\partial}{\partial \sigma} = \frac{\partial z_\sigma}{\partial z} \frac{\partial}{\partial z_\sigma} = \frac{\partial}{\partial z_\sigma} \\ \frac{\partial}{\partial t} &= \frac{\partial}{\partial t'} - \frac{1}{H} \frac{\partial z_\sigma}{\partial t} \frac{\partial}{\partial \sigma} = \frac{\partial}{\partial t'} - \frac{\partial z_\sigma}{\partial t} \frac{\partial}{\partial z_\sigma} \end{aligned} \quad (\text{E.8})$$

These are the first order Cartesian derivatives, expressed in the σ -space, with σ on its turn expressed in a Cartesian vertical co-ordinate.

Defining the transformed vertical velocity

The Cartesian vertical velocity w [m s^{-1}] is defined as $w = Dz/Dt = dz/dt$ [Bijvelds 2001, p.15].

Similarly the vertical transformed velocity ω [m s^{-1}] is defined by $\omega = HD\sigma/Dt$:

$$\begin{aligned} \omega &= H \frac{D\sigma}{Dt} = H \left[\frac{1}{H} \frac{Dz}{Dt} - \frac{1}{H} \frac{D\zeta}{Dt} - \frac{(z-\zeta)}{H^2} \frac{DH}{Dt} \right] \Leftrightarrow \\ \omega &= \frac{Dz}{Dt} - \frac{D\zeta}{Dt} - \sigma \frac{DH}{Dt} \Leftrightarrow \\ \omega &= w - \left(\frac{\partial \zeta}{\partial t} + \sigma \frac{\partial H}{\partial t} \right) - u \left(\frac{\partial \zeta}{\partial x} + \sigma \frac{\partial H}{\partial x} \right) \Leftrightarrow \\ \omega &= w - \frac{\partial z_\sigma}{\partial t} - u \frac{\partial z_\sigma}{\partial x} \end{aligned} \quad (\text{E.9})$$

In words: the vertical velocity of a particle in a Cartesian frame of reference (w) is equal to the vertical velocity relative to the z_σ -line (ω) plus the vertical velocity of the z_σ -co-ordinate itself, relative to the Cartesian frame, going along with the particle.

Transformation of the material derivative

Substituting the expressions for the spatial and time derivatives (E.8) and the expression for the transformed velocity ω (E.9) yields:

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + w \frac{\partial}{\partial z} = \frac{\partial}{\partial t'} - \frac{\partial z_\sigma}{\partial t} \frac{\partial}{\partial z_\sigma} + u \frac{\partial}{\partial x'} - u \frac{\partial z_\sigma}{\partial x} \frac{\partial}{\partial z_\sigma} + w \frac{\partial}{\partial z_\sigma} \Leftrightarrow$$

$$\frac{D}{Dt} = \frac{\partial}{\partial t'} + u \frac{\partial}{\partial x'} + \left[w - \frac{\partial z_\sigma}{\partial t} - u \frac{\partial z_\sigma}{\partial x} \right] \frac{\partial}{\partial z_\sigma} \Leftrightarrow$$

$$\frac{D}{Dt} = \frac{\partial}{\partial t'} + u' \frac{\partial}{\partial x'} + \omega \frac{\partial}{\partial z_\sigma} = \frac{D}{Dt'} \quad (E.10)$$

Appendix F: Derivation of the 2DV hydrodynamic sigma model

F.1 Introduction

The purpose of this appendix is to mathematically derive a 2DV hydrostatic sigma layered model for flow and transport. This means that there will be one horizontal and one vertical dimension. The momentum equation will be solved in both directions.

Part of the model existed already. For this study the program is extended with a transport equation and the baroclinic pressure terms in the horizontal momentum equation. However the derivation of the discrete equations begins with the general Cartesian equations from chapter 3. When additional terms arise compared to the original model, this will be noted in the text.

The model is equipped for interpolation of the vertical pressure profile. This means that the model uses a coarser vertical grid to solve the pressures, after which interpolation is used to get a pressure point at every velocity layer. For more information about the interpolation and solving the system of equations, the reader is referred to [Van Reeuwijk 2002].

F.2 Model definition in Cartesian co-ordinates

Consider a 2DV Cartesian co-ordinate system with horizontal co-ordinate x and vertical co-ordinate z . (The z -axis is pointing opposite to the direction of the gravitational acceleration vector.)

We will use the continuity and momentum equations (3.38) to (3.41), without using the y -direction. This gives us the flow equations in the 2DV form.

For simplicity we now ignore the viscous terms. Thereby, the Navier-Stokes equation reduces to the Euler equation [Bird 1960, p. 81].

Continuity equation:

$$\frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} = 0 \quad (\text{F.1})$$

Momentum equation in the x -direction:

$$\frac{Du}{Dt} = \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x} \quad (\text{F.2})$$

Momentum equation in the z -direction:

$$\frac{Dw}{Dt} = \frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + w \frac{\partial w}{\partial z} = -\left(\frac{\rho}{\rho_0}\right)g - \frac{1}{\rho_0} \frac{\partial p}{\partial z} \quad (\text{F.3})$$

For the free surface and bottom kinematic boundary conditions are needed. These are:

$$w(x, -d, t) = -\frac{\partial d}{\partial t} - u \frac{\partial d}{\partial x} = -\frac{Dd}{Dt} \quad z = -d(x, t) : (\text{Bottom}) \quad (\text{F.4})$$

$$w(x, \zeta, t) = \frac{\partial \zeta}{\partial t} + u \frac{\partial \zeta}{\partial x} = \frac{D\zeta}{Dt} \quad z = \zeta(x, t) : (\text{Free surface}) \quad (\text{F.5})$$

See Appendix C for a derivation of these boundary conditions.

Now we split the pressure in the momentum equations in two parts: a hydrostatic part p_h and a hydrodynamic part q .

$$p = p_h + q \quad (\text{F.6})$$

Note: the hydrostatic pressure is not the same as the static pressure. A hydrostatic pressure distribution can cause flow, whereas in case of static pressure there is a no flow situation.

At the free surface the pressure is equal to the atmospheric pressure. In addition, we define the hydrodynamic pressure to be zero at the free surface.

$$q(x, \zeta, t) = 0 \quad (\text{F.7})$$

It now follows that:

$$p_h(x, \zeta, t) = p_{atm} \quad (\text{F.8})$$

Substituting this gives the following momentum equations:

$$\frac{Du}{Dt} = -\frac{1}{\rho_0} \left[\frac{\partial p_h}{\partial x} + \frac{\partial q}{\partial x} \right] \quad (\text{F.9})$$

$$\frac{Dw}{Dt} = -\left(\frac{\rho}{\rho_0} \right) g - \frac{1}{\rho_0} \left[\frac{\partial p_h}{\partial z} + \frac{\partial q}{\partial z} \right] \quad (\text{F.10})$$

We now define the hydrostatic part of the pressure by:

$$-\left(\frac{\rho}{\rho_0} \right) g - \frac{1}{\rho_0} \frac{\partial p_h}{\partial z} = 0 \Leftrightarrow \frac{\partial p_h}{\partial z} = -\rho g \quad (\text{F.11})$$

Substitution of Equation (F.11) into Equation (F.10) yields

$$\frac{Dw}{Dt} = -\frac{1}{\rho_0} \frac{\partial q}{\partial z} \quad (\text{F.12})$$

Note: this means that implementing the hydrostatic pressure relation does not constitute to equating the hydrodynamic pressure (gradient) zero.

It means that when calculating the vertical acceleration, the vertical gradient of the hydrodynamic pressure is not taken into account, only continuity.

It is now possible to write an expression for the hydrostatic pressure. Integration over z gives:

$$\begin{aligned}\frac{\partial p_h}{\partial z} &= -\rho g \Rightarrow \int_z^\zeta \frac{\partial p_h}{\partial z'} dz' = -g \int_z^\zeta \rho dz' \Leftrightarrow \\ p_h|_z^\zeta &= -g \int_z^\zeta \rho dz' \Leftrightarrow \\ p_h &= p_{atm} + g \int_z^\zeta \rho dz'\end{aligned}\tag{F.13}$$

After having obtained expression (F.13) for the hydrostatic pressure, we differentiate to x to get the horizontal pressure gradient.

$$\frac{\partial}{\partial x} [p_h(x, z, t)] = \frac{\partial p_{atm}}{\partial x} + g \frac{\partial}{\partial x} \left[\int_z^{\zeta(x, t)} \rho(x, z', t) dz' \right]\tag{F.14}$$

We assume that $\partial p_{atm} / \partial x = 0$.

$$\frac{\partial}{\partial x} [p_h(x, z, t)] = g \frac{\partial}{\partial x} \left[\int_z^{\zeta(x, t)} \rho(x, z', t) dz' \right]\tag{F.15}$$

Leibnitz's rule is represented by:

$$\frac{d}{dc} \int_{a(c)}^{b(c)} f(c, z) dz = \int_{a(c)}^{b(c)} \frac{\partial f(c, z)}{\partial c} dz + f(c, b) \frac{db(c)}{dc} - f(c, a) \frac{da(c)}{dc}\tag{F.16}$$

Applying Leibnitz's rule on (F.15) gives:

$$\begin{aligned}\frac{\partial}{\partial x} [p_h(x, z, t)] &= g \int_z^{\zeta(x, t)} \frac{\partial}{\partial x} [\rho(x, z', t)] dz' + \rho(x, z', t) g \frac{d[\zeta(x, t)]}{dx} - \rho(x, z', t) g \frac{\partial z}{\partial x} \\ &= g \int_z^{\zeta(x, t)} \frac{\partial}{\partial x} [\rho(x, z', t)] dz' + \rho(x, z', t) g \frac{\partial [\zeta(x, t)]}{\partial x}\end{aligned}\tag{F.17}$$

$$\frac{\partial p_h}{\partial x} = g \int_z^\zeta \frac{\partial \rho}{\partial x} dz' + \rho g \frac{\partial \zeta}{\partial x}\tag{F.18}$$

Substituting the expressions for the gradients of p_h in the momentum equations (F.9) and (F.10) and using the definition of the hydrostatic part of the pressure gives:

$$\frac{Du}{Dt} + \frac{\rho g}{\rho_0} \frac{\partial \zeta}{\partial x} + \frac{1}{\rho_0} \frac{\partial q}{\partial x} + \frac{g}{\rho_0} \int_z^{\zeta(x, t)} \frac{\partial \rho}{\partial x} dz' = 0\tag{F.19}$$

$$\frac{Dw}{Dt} + \frac{1}{\rho_0} \frac{\partial q}{\partial z} = 0 \quad (\text{F.20})$$

Combined with the equation of continuity (F.1), here repeated for convenience,

$$\frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} = 0 \quad (\text{F.1})$$

we now have three equations with three unknowns.

Comparison with the equations for uniform density (as used in the original model) leads to the conclusion that extending the problem from uniform to non-uniform densities introduces the following term in the horizontal momentum equation (F.19):

$$\frac{g}{\rho_0} \int_z^{\zeta(x,t)} \frac{\partial \rho}{\partial x} dz'.$$

Also, in the water level gradient term there now appears a factor ρ/ρ_0 . This factor is introduced by the use of the Boussinesq approximation.

Obviously this factor was equal to 1 in case of uniform density. In case of non-uniform densities this factor is approaching unity when the density differences are small compared to the reference density.

Because (i) this factor is introduced by the use of an approximation and (ii) in some cases the appearance of an extra variable in a term can give rise to (numerical) instabilities, it is decided to drop this factor in the equation.

$$\left(\frac{\rho}{\rho_0} \right) g \frac{\partial \zeta}{\partial x} \rightarrow g \frac{\partial \zeta}{\partial x} \quad (\text{F.21})$$

The horizontal momentum equation now reduces to:

$$\frac{Du}{Dt} + g \frac{\partial \zeta}{\partial x} + \frac{1}{\rho_0} \frac{\partial q}{\partial x} + \frac{g}{\rho_0} \int_z^{\zeta(x,t)} \frac{\partial \rho}{\partial x} dz' = 0 \quad (\text{F.22})$$

F.3 Sigma transformation

The model that will be extended makes use of equations in the sigma co-ordinate form. Therefore, all the equations in Cartesian co-ordinates have to be transformed to sigma co-ordinates.

The equation will not be solved in the actual sigma co-ordinate form. After the transform to sigma co-ordinates, another transform will be used to express the vertical in z_σ -co-ordinates ($-d \leq z_\sigma \leq \zeta$), instead of the actual sigma co-ordinates ($-1 \leq \sigma \leq 0$). Hence, z_σ can be seen as a function expressing the Cartesian height of a line with constant σ (an σ -isoline).

An advantage of the sigma grid is that it is easier to implement from a computational point of view. This is because there is no need for difficult procedures for flooding and drying of cells at the water surface.

A drawback of this grid is that it can cause artificial mixing in stratified environments. However, this is only true in case of steep gradients in σ -isolines. Or, as Bijvelds put it:

“Significant differences between the σ -model and the z -model are only observed when modelling regions with steep topography using coarse horizontal grids. For sufficiently high horizontal resolution of the computational grid, the results of both models should converge, irrespective of the slope of the bottom.” [Bijvelds, 2001, p.111]

Because in this thesis no use is made of steep bottoms, using a σ -grid is not expected to influence the results.

In case of a horizontal bottom, the only cause of horizontal gradients in the σ -isolines is a variation in the water level elevation. This remains sufficiently small compared to the total water depth to give the σ -grid approximate Cartesian properties.

The first order Cartesian derivatives, expressed in the σ -space, with σ on its turn expressed in a Cartesian vertical co-ordinate (z_σ) are:

$$\begin{aligned}\frac{\partial}{\partial x} &= \frac{\partial}{\partial x'} - \frac{\partial z_\sigma}{\partial x} \frac{\partial}{\partial z_\sigma} \\ \frac{\partial}{\partial z} &= \frac{\partial}{\partial z_\sigma} \\ \frac{\partial}{\partial t} &= \frac{\partial}{\partial t'} - \frac{\partial z_\sigma}{\partial t} \frac{\partial}{\partial z_\sigma}\end{aligned}\tag{F.23}$$

The vertical transformed velocity ω is represented by:

$$\omega = w - \frac{\partial z_\sigma}{\partial t} - u \frac{\partial z_\sigma}{\partial x}\tag{F.24}$$

The transformed material derivative can be written as:

$$\frac{D}{Dt} = \frac{\partial}{\partial t'} + u' \frac{\partial}{\partial x'} + \omega \frac{\partial}{\partial z_\sigma} = \frac{D}{Dt'}\tag{F.25}$$

More information concerning the σ -transform and the derivation of Equations (F.23) to (F.25) can be found in Appendix E.

F.4 Transforming the equations

After having expressed the first order Cartesian derivatives in z_σ -co-ordinates, it is easy to transform the equations by substituting Equations (F.23) to (F.25) in the relevant equations.

Continuity equation

$$\frac{\partial u}{\partial x} + \frac{\partial w}{\partial z} = 0 \xrightarrow{T_\sigma} \frac{\partial u}{\partial x'} + \frac{\partial w}{\partial z_\sigma} - \frac{\partial z_\sigma}{\partial x} \frac{\partial u}{\partial z_\sigma} = 0 \quad (\text{F.26})$$

Depth integrated continuity equation

In some cases use is made of the depth integrated continuity equation. For a derivation of this equation see Appendix D.

$$\frac{\partial \zeta}{\partial t} + \frac{\partial}{\partial x} \int_{-d}^{\zeta} u dz = 0 \xrightarrow{T_\sigma} \frac{\partial \zeta}{\partial t'} + \frac{\partial}{\partial x'} \int_{-d}^{\zeta} u' dz_\sigma = 0 \quad (\text{F.27})$$

Horizontal momentum equation

$$\begin{aligned} \frac{Du}{Dt} + g \frac{\partial \zeta}{\partial x} + \frac{1}{\rho_0} \frac{\partial q}{\partial x} + \frac{g}{\rho_0} \int_z^{\zeta(x,t)} \frac{\partial \rho}{\partial x} dz' &= 0 \xrightarrow{T_\sigma} \\ \frac{Du}{Dt'} + g \left[\frac{\partial \zeta}{\partial x'} - \frac{\partial z_\sigma}{\partial x} \frac{\partial \zeta}{\partial z_\sigma} \right] + \frac{1}{\rho_0} \left[\frac{\partial q}{\partial x'} - \frac{\partial z_\sigma}{\partial x} \frac{\partial q}{\partial z_\sigma} \right] + \frac{g}{\rho_0} \int_{z_\sigma}^{\zeta(x,t)} \frac{\partial \rho}{\partial x'} dz'_\sigma - \frac{g}{\rho_0} \int_{z_\sigma}^{\zeta(x,t)} \frac{\partial z_\sigma}{\partial x} \frac{\partial \rho}{\partial z_\sigma} dz'_\sigma &= 0 \\ \frac{Du}{Dt'} + g \frac{\partial \zeta}{\partial x'} + \frac{1}{\rho_0} \frac{\partial q}{\partial x'} - \frac{1}{\rho_0} \frac{\partial z_\sigma}{\partial x} \frac{\partial q}{\partial z_\sigma} + \frac{g}{\rho_0} \int_{z_\sigma}^{\zeta(x,t)} \frac{\partial \rho}{\partial x'} dz'_\sigma - \frac{g}{\rho_0} \int_{z_\sigma}^{\zeta(x,t)} \frac{\partial z_\sigma}{\partial x} \frac{\partial \rho}{\partial z_\sigma} dz'_\sigma &= 0 \end{aligned} \quad (\text{F.28})$$

Vertical momentum equation

$$\frac{Dw}{Dt} + \frac{1}{\rho_0} \frac{\partial q}{\partial z} = 0 \xrightarrow{T_\sigma} \frac{Dw}{Dt'} + \frac{1}{\rho_0} \frac{\partial q}{\partial z_\sigma} = 0 \quad (\text{F.29})$$

Transport equation

$$\frac{D\rho}{Dt} = \frac{\partial \rho}{\partial t} + u \frac{\partial \rho}{\partial x} + w \frac{\partial \rho}{\partial z} = 0 \xrightarrow{T_\sigma} \frac{D\rho}{Dt'} = \frac{\partial \rho}{\partial t'} + u' \frac{\partial \rho}{\partial x'} + \omega \frac{\partial \rho}{\partial z_\sigma} = 0 \quad (\text{F.30})$$

As can be seen above, the equations are expressed in terms of $(x, x', z_\sigma, t, t', u, u', w, \omega)$.

Because $x = x', t = t', u = u'$, we can and will from now on drop the indices. We now have the equations in terms of (x, z, t, u, w, ω) . It is possible to substitute an expression for ω in the

equations, but for the ease of the computation we leave them here in the equations. Note that ω can be found in the expression for the transformed material derivative (cf. Equation (F.25)).

F.5 Discretisation of the equations

After having reformulated the equations in z_σ -co-ordinates, they will be discretised, thus forming a discrete analogue of the original continuum equations. To do so, a staggered grid with a fixed number of layers is introduced.

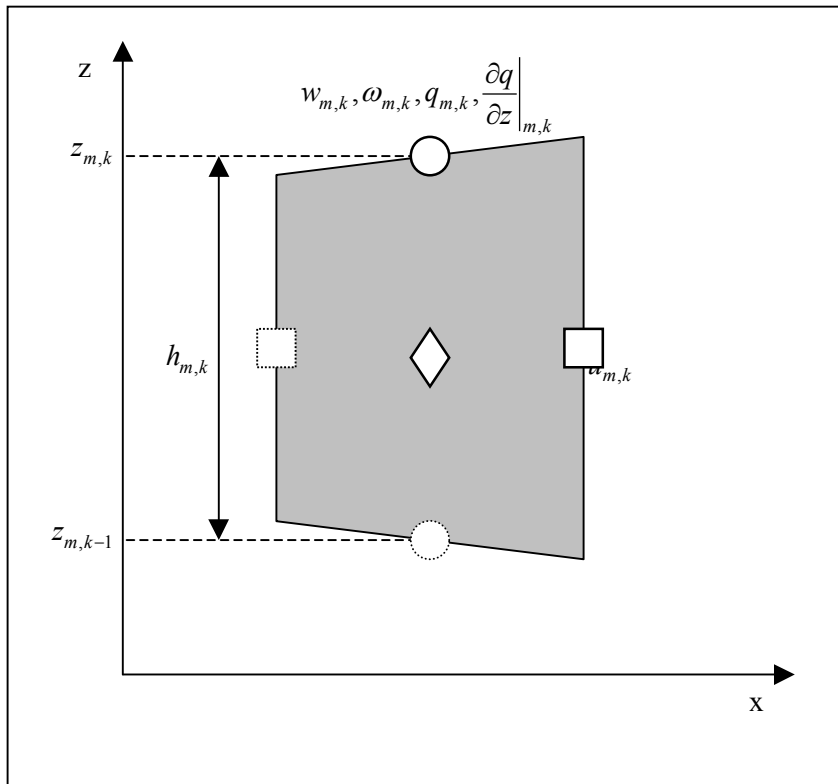


Figure F-1: *Cell definition of the staggered grid.*

The layers are defined from layer 1 (bottom layer) to layer $kmax$ (top layer). Note that the number of layer interfaces is $kmax + 1$.

The layer thickness and z -co-ordinate is defined at pressure points. However, sometimes the layer thickness or water level at a velocity point is needed. Therefore an upwind approach is applied. This means that the layer thickness or water level in a velocity point is dependent on the direction of the depth-averaged flow. This method guarantees a positive total water depth at pressure points.

To implement this, an upwind operator μ is defined as:

$$\mu(m) = \begin{cases} m & \text{if } U_m > 0 \\ m+1 & \text{if } U_m < 0 \\ m & \text{if } U_m = 0 \quad \wedge \quad \zeta_m > \zeta_{m+1} \\ m+1 & \text{if } U_m = 0 \quad \wedge \quad \zeta_m < \zeta_{m+1} \end{cases} \quad (\text{F.31})$$

F.5.1 Local continuity equation

$$\frac{\partial u}{\partial x'} + \frac{\partial w}{\partial z_\sigma} - \frac{\partial z_\sigma}{\partial x} \frac{\partial u}{\partial z_\sigma} = 0 \quad (\text{8.22})$$

Before discretising this equation we integrate over layer k :

$$\int_{z_{k-1}}^{z_k} \left(\frac{\partial u}{\partial x'} + \frac{\partial w}{\partial z_\sigma} - \frac{\partial z_\sigma}{\partial x} \frac{\partial u}{\partial z_\sigma} \right) dz_\sigma = 0 \quad (\text{F.32})$$

$$\frac{\partial (h_k \bar{u})}{\partial x'} + w(z_k) - w(z_{k-1}) - \frac{\partial z_\sigma}{\partial x} u \Big|_{z_{k-1}}^{z_k} = 0 \quad (\text{F.33})$$

with:

$$h_k = z_k - z_{k-1}$$

$$\bar{u}_k = \frac{1}{h_k} \int_{z_{k-1}}^{z_k} u dz'_\sigma$$

Discretisation:

$$\frac{h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1}}{\Delta x} + w_{m,k}^{n+1} - w_{m,k-1}^{n+1} +$$

$$- \left(\tilde{u}_{m,k}^{n+1} \frac{z_{m+1,k}^n - z_{m-1,k}^n}{2\Delta x} - \tilde{u}_{m,k-1}^{n+1} \frac{z_{m+1,k-1}^n - z_{m-1,k-1}^n}{2\Delta x} \right) = 0 \quad (\text{F.34})$$

with:

$$\tilde{u}_{m,k}^{n+1} = \frac{h_{\mu(m-1),k}^n u_{m-1,k}^{n+1} + h_{\mu(m-1),k+1}^n u_{m-1,k+1}^{n+1} + h_{\mu(m),k}^n u_{m,k}^{n+1} + h_{\mu(m),k}^n u_{m,k+1}^{n+1}}{2h_{m,k}^n + 2h_{m,k+1}^n} = 0 \quad (\text{F.35})$$

$\tilde{u}$ is the horizontal velocity at a ω - point. Because the horizontal velocity is not defined there, it is (upwind) interpolated, as shown in (F.35).

F.5.2 Kinematic boundary conditions

Discretising the expression for the vertical sigma velocity (IV.11) gives:

$$\omega_{m,k}^{n+1} = w_{m,k}^{n+1} - \frac{z_{m,k}^{n+1} - z_{m,k}^n}{\Delta t} - \tilde{u}_{m,k}^{n+1} \frac{z_{m+1,k}^n + z_{m-1,k}^n}{2\Delta x} \quad (\text{F.36})$$

thereby also using expression (F.35) for the interpolated velocities at ω - points.

The discretised kinematic boundary conditions, using Equation (F.4) and (F.5) become:

$$w_{m,kmax}^{n+1} = \frac{\zeta_m^{n+1} - \zeta_m^n}{\Delta t} + \frac{h_{\mu(m-1),kmax}^n u_{m-1,kmax}^{n+1} + h_{\mu(m),kmax}^n u_{m,kmax}^{n+1}}{2h_{m,kmax}^n} \frac{\zeta_{m+1}^n - \zeta_{m-1}^n}{2\Delta x} \quad (\text{F.37})$$

$$w_{m,0}^{n+1} = -\frac{h_{\mu(m-1),1}^n u_{m-1,1}^{n+1} + h_{\mu(m),1}^n u_{m,1}^{n+1}}{2h_{m,1}^n} \frac{d_{m+1} - d_{m-1}}{2\Delta x} \quad (\text{F.38})$$

F.5.3 Depth integrated continuity equation

Discretising the depth integrated continuity equation (cf. Equation (F.27)) yields

$$\frac{\zeta_m^{n+1} - \zeta_m^n}{\Delta t} + \frac{1}{\Delta x} \sum_{k=1}^{k=\max} \left[h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1} \right] = 0 \quad (\text{F.39})$$

F.5.4 Horizontal momentum equation

The primary unknown to be solved will be the vertical non-hydrostatic pressure gradient. Therefore, all the non-hydrostatic pressures in the momentum equations have to be replaced by the non-hydrostatic vertical pressure gradient.

This is done by substituting the following relation:

$$\frac{q_{m,k} - q_{m,k-1}}{h_{m,k}} = \frac{1}{2} \frac{\partial q}{\partial z} \Big|_{m,k} + \frac{1}{2} \frac{\partial q}{\partial z} \Big|_{m,k-1} \quad (\text{F.40})$$

Since $q=0$ at the water surface, $q_{m,k}$ can be expressed as a sum of the vertical pressure gradients:

$$q_{m,k} = -\frac{1}{2} \sum_{l=k+1}^{kmax} h_{m,l} \left[\frac{\partial q}{\partial z} \Big|_{m,l}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m,l-1}^{n+1} \right] \quad (\text{F.41})$$

With this, the horizontal momentum equation (F.28) is discretised as:

$$\begin{aligned} & \frac{u_{m,k}^{n+1} - u_{m,k}^n}{\Delta t} + ADV(u)_{m,k}^n + g \frac{\zeta_{m+1}^n - \zeta_m^n}{\Delta x} - \frac{1}{\rho_0} \frac{(q_{m+1,k}^{n+1} + q_{m+1,k-1}^{n+1}) - (q_{m,k}^{n+1} + q_{m,k-1}^{n+1})}{2\Delta x} + \\ & - \frac{1}{4\rho_0} \left[\frac{\partial q}{\partial z} \Big|_{m,k}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m,k-1}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m+1,k}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m+1,k-1}^{n+1} \right] \frac{z_{m+1,k}^n - z_{m,k}^n + z_{m+1,k-1}^n - z_{m,k-1}^n}{2\Delta x} + \quad (F.42) \\ & + \text{Baroclinic Pressure Term}_{m,k}^n = 0 \end{aligned}$$

For discretising the baroclinic pressure term, we choose to use a different method. Summarised, this is done by integrating the Cartesian pressure term over a layer with constant sigma. The actual discretisation of the baroclinic pressure term is done in Appendix G and yields as result:

$$\frac{g}{\rho_0} \left\{ \frac{1}{2} \tilde{h}_{m,k}^n \frac{\rho_{m+1,k}^n - \rho_{m,k}^n}{\Delta x} + \sum_{j=k+1}^{kmax} \left[\tilde{h}_{m,j}^n \frac{\rho_{m+1,j}^n - \rho_{m,j}^n}{\Delta x} + (\tilde{\rho}_{m,j}^n - \tilde{\rho}_{m,k}^n) \frac{h_{m+1,j}^n - h_{m,j}^n}{\Delta x} \right] \right\} \quad (F.43)$$

with:

$$\tilde{h}_{m,k} = h_{\mu(m),k}$$

$$\tilde{h}_{m,k} = h_{\mu(m),k}$$

As can be seen, $\tilde{h}_{m,k}$ is discretised using an upstream approach, the same as in the continuity equation and the transport equation. As a check, it can be easily seen that in a uniform density field the baroclinic pressure term is zero.

Now, the discretised horizontal momentum equation is:

$$\begin{aligned} & \frac{u_{m,k}^{n+1} - u_{m,k}^n}{\Delta t} + ADV(u)_{m,k}^n + g \frac{\zeta_{m+1}^n - \zeta_m^n}{\Delta x} - \frac{1}{\rho_0} \frac{(q_{m+1,k}^{n+1} + q_{m+1,k-1}^{n+1}) - (q_{m,k}^{n+1} + q_{m,k-1}^{n+1})}{\Delta 2x} + \\ & - \frac{1}{4\rho_0} \left[\frac{\partial q}{\partial z} \Big|_{m,k}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m,k-1}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m+1,k}^{n+1} + \frac{\partial q}{\partial z} \Big|_{m+1,k-1}^{n+1} \right] \frac{z_{m+1,k}^n - z_{m,k}^n + z_{m+1,k-1}^n - z_{m,k-1}^n}{2\Delta x} + \quad (F.44) \\ & + \frac{g}{\rho_0} \left\{ \frac{1}{2} \tilde{h}_{m,k}^n \frac{\rho_{m+1,k}^n - \rho_{m,k}^n}{\Delta x} + \sum_{j=k+1}^{kmax} \left[\tilde{h}_{m,j}^n \frac{\rho_{m+1,j}^n - \rho_{m,j}^n}{\Delta x} + (\tilde{\rho}_{m,j}^n - \tilde{\rho}_{m,k}^n) \frac{h_{m+1,j}^n - h_{m,j}^n}{\Delta x} \right] \right\} = 0 \end{aligned}$$

F.5.5 Vertical momentum equation

$$\frac{D\omega}{Dt'} + \frac{1}{\rho_0} \frac{\partial q}{\partial z_\sigma} = 0$$

$$\frac{w_{m,k}^{n+1} - w_{m,k}^n}{\Delta t} + \text{ADV}(w)_{m,k}^n + \frac{1}{\rho_0} \frac{\partial q}{\partial z} \Big|_{m,k}^{n+1} = 0$$

F.5.6 Transport equation

To discretise the transport equation (cf. Equation (F.30)),

$$\frac{D\rho}{Dt'} = \frac{\partial \rho}{\partial t'} + u' \frac{\partial \rho}{\partial x'} + \omega \frac{\partial \rho}{\partial z_\sigma} = 0 \quad (\text{F.45})$$

in a conservative way, we start by adding the continuity equation (Eq. (F.26)).

$$\frac{D\rho}{Dt'} = 0 \Leftrightarrow \frac{D\rho}{Dt'} + \rho \left[\frac{\partial u}{\partial x'} + \frac{\partial w}{\partial z_\sigma} - \frac{\partial z_\sigma}{\partial x} \frac{\partial u}{\partial z_\sigma} \right] = 0 \quad (\text{F.46})$$

Substituting an expression for w (cf. Equation (E.9)) and writing out the material derivative gives for the transport (advection) equation:

$$\frac{D\rho}{Dt'} + \rho \left[\frac{\partial u}{\partial x'} + \frac{\partial \omega}{\partial z_\sigma} \right] = 0 \Leftrightarrow \frac{\partial \rho}{\partial t'} + u \frac{\partial \rho}{\partial x'} + \rho \frac{\partial u}{\partial x'} + \omega \frac{\partial \rho}{\partial z_\sigma} + \rho \frac{\partial \omega}{\partial z_\sigma} = 0 \quad (\text{F.47})$$

This can also be written as:

$$\frac{\partial \rho}{\partial t'} + \frac{\partial(u\rho)}{\partial x'} + \frac{\partial(\omega\rho)}{\partial z_\sigma} = 0 \quad (\text{F.48})$$

Now we integrate over a layer k :

$$\begin{aligned} \int_{z_{k-1}}^{z_k} \left(\frac{\partial \rho}{\partial t'} + \frac{\partial(u\rho)}{\partial x'} + \frac{\partial(\omega\rho)}{\partial z_\sigma} \right) dz'_\sigma &= 0 \quad \Leftrightarrow \\ \int_{z_{k-1}}^{z_k} \frac{\partial \rho}{\partial t'} dz'_\sigma + \int_{z_{k-1}}^{z_k} \frac{\partial(u\rho)}{\partial x'} dz'_\sigma + \int_{z_{k-1}}^{z_k} \frac{\partial(\omega\rho)}{\partial z_\sigma} dz'_\sigma &= 0 \quad \Leftrightarrow \\ \frac{\partial(h_k \rho_k)}{\partial t'} + \frac{\partial(h_k u_k \rho_k)}{\partial x'} + \omega_k \rho(z_k) - \omega_{k-1} \rho(z_{k-1}) & \end{aligned} \quad (\text{F.49})$$

Using:

$$\int_{z_{k-1}}^{z_k} \rho dz = h_k \rho_k$$

The transport equation (F.49) will be discretised explicitly using control volumes with a density point in the centre of a control volume.

Because the density distribution of time step $n+1$ will be computed after solving the system of momentum and continuity equations, the new geometry and velocity field is already known.

Therefore these variables can be used explicitly, while being denoted in the transport equation with a time step superscript $n + 1$.

The discretisation for the transport equation is:

$$\frac{(h_{m,k}^{n+1}\rho_{m,k}^{n+1} - h_{m,k}^n\rho_{m,k}^n)}{\Delta t} + \frac{(\tilde{h}_{m,k}^{n+1}u_{m,k}^{n+1}\rho_E^n - \tilde{h}_{m-1,k}^{n+1}u_{m-1,k}^{n+1}\rho_W^n)}{\Delta x} + \omega_{m,k}^{n+1}\rho_N^n - \omega_{m,k-1}^{n+1}\rho_S^n = 0 \quad (\text{F.50})$$

$$h_{m,k}^{n+1}\rho_{m,k}^{n+1} = h_{m,k}^n\rho_{m,k}^n - \frac{\Delta t}{\Delta x}(\tilde{h}_{m,k}^{n+1}u_{m,k}^{n+1}\rho_E^n - \tilde{h}_{m-1,k}^{n+1}u_{m-1,k}^{n+1}\rho_W^n) + \Delta t(\omega_{m,k}^{n+1}\rho_N^n - \omega_{m,k-1}^{n+1}\rho_S^n) \quad (\text{F.51})$$

The following discretisation is used for the transported densities through the cell faces.

$$\begin{aligned} \rho_E &= \begin{cases} \rho_{m,k} & , u_{m,k} > 0 \\ \rho_{m+1,k} & , u_{m,k} < 0 \end{cases} & \rho_N &= \begin{cases} \rho_{m,k} & , \omega_{m,k} > 0 \\ \rho_{m,k+1} & , \omega_{m,k} < 0 \end{cases} \\ \rho_W &= \begin{cases} \rho_{m-1,k} & , u_{m-1,k} > 0 \\ \rho_{m,k} & , u_{m-1,k} < 0 \end{cases} & \rho_S &= \begin{cases} \rho_{m,k-1} & , \omega_{m-1,k} > 0 \\ \rho_{m,k} & , \omega_{m-1,k} < 0 \end{cases} \end{aligned} \quad (\text{F.52})$$

Also important is the discretisation of the cell face heights (at horizontal velocity points). This discretisation should be chosen such that the discretised transport equation is consistent with the discretised continuity equation. Therefore the upstream method is used, as this has also been used in discretising the continuity equation.

$$\tilde{h}_{m,k} = h_{\mu(m),k} \quad (\text{F.53})$$

F.6 Consistency with continuity

A minimal monotonicity requirement is that an initially uniform scalar field remains uniform in the absence of sources and sinks. This property is granted if the discretisation of the advection equation is consistent with the discretisation of the continuity equation. This is called the concept of *consistency with continuity* (CWC).

A definition is given by [Gross et. al. 2002]:

A discretisation of the advection equation is consistent with continuity if, given a spatially uniform scalar field as initial datum, and a general flow field, the discretized scalar advection equation reduces to the discretised continuity equation.

In order to prove consistency with continuity, we start with rewriting the discretised local continuity equation (F.34) by substituting the expression for the vertical velocity (F.36)

$$\frac{h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1}}{\Delta x} + \frac{z_{m,k}^{n+1} - z_{m,k}^n}{\Delta t} - \frac{z_{m,k-1}^{n+1} - z_{m,k-1}^n}{\Delta t} + \omega_{m,k}^{n+1} - \omega_{m,k-1}^{n+1} = 0 \quad (\text{F.54})$$

Substituting $z_{m,k} - z_{m,k-1} = h_{m,k}$ gives:

$$\frac{h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1}}{\Delta x} + \frac{h_{m,k}^{n+1} - h_{m,k}^n}{\Delta t} + \omega_{m,k}^{n+1} - \omega_{m,k-1}^{n+1} = 0 \quad (\text{F.55})$$

We can write this as:

$$\omega_{m,k}^{n+1} = \omega_{m,k-1}^{n+1} - \frac{h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1}}{\Delta x} - \frac{h_{m,k}^{n+1} - h_{m,k}^n}{\Delta t} \quad (\text{F.56})$$

The transport equation:

$$\begin{aligned} h_{m,k}^{n+1} \rho_{m,k}^{n+1} &= h_{m,k}^n \rho_{m,k}^n - \frac{\Delta t}{\Delta x} \left(\tilde{h}_{m,k} u_{m,k} \hat{\rho}_{m,k} - \tilde{h}_{m-1,k} u_{m-1,k} \hat{\rho}_{m-1,k} \right) + \\ &+ \Delta t \left(\omega_{m,k-1} \hat{\rho}_{m,k-1} - \omega_{m,k} \hat{\rho}_{m,k} \right) \end{aligned} \quad (\text{F.57})$$

CWC will be illustrated for an arbitrary cell. Assuming $\rho_{m,k}^n = \rho^0$ (i.e., uniform density), Equation (F.57) can be written

$$h_{m,k}^{n+1} \rho_{m,k}^{n+1} = \rho^0 \left[h_{m,k}^n - \frac{\Delta t}{\Delta x} \left(\tilde{h}_{m,k} u_{m,k} - \tilde{h}_{m-1,k} u_{m-1,k} \right) + \Delta t \left(\omega_{m,k-1} - \omega_{m,k} \right) \right] \quad (\text{F.58})$$

By substitution of Equation (F.56), Equation (F.58) can be simplified to

$$h_{m,k}^{n+1} \rho_{m,k}^{n+1} = \rho^0 \left[h_{m,k}^n - \frac{\Delta t}{\Delta x} \left(\tilde{h}_{m,k} u_{m,k} - \tilde{h}_{m-1,k} u_{m-1,k} \right) + \Delta t \left(\frac{h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1}}{\Delta x} + \frac{h_{m,k}^{n+1} - h_{m,k}^n}{\Delta t} \right) \right] \quad (\text{F.59})$$

$$h_{m,k}^{n+1} \rho_{m,k}^{n+1} = \rho^0 \left[h_{m,k}^n - \frac{\Delta t}{\Delta x} \left(\tilde{h}_{m,k} u_{m,k} - \tilde{h}_{m-1,k} u_{m-1,k} \right) + \frac{\Delta t}{\Delta x} \left(h_{\mu(m),k}^n u_{m,k}^{n+1} - h_{\mu(m-1),k}^n u_{m-1,k}^{n+1} \right) + \left(h_{m,k}^{n+1} - h_{m,k}^n \right) \right] \quad (\text{F.60})$$

$$h_{m,k}^{n+1} \rho_{m,k}^{n+1} = \rho^0 \left[h_{m,k}^n + \left(h_{m,k}^{n+1} - h_{m,k}^n \right) \right] \quad (\text{F.61})$$

$$h_{m,k}^{n+1} \rho_{m,k}^{n+1} = h_{m,k}^{n+1} \rho^0 \quad (\text{F.62})$$

Therefore it holds that $\rho_{m,k}^{n+1} = \rho^0$ and Equation (F.58) is equivalent to the discretised continuity Equation (F.34).

Appendix G: Transformation and discretisation of the baroclinic pressure term

In this appendix, the baroclinic pressure term will first be transformed and vertically discretised analogous to the transformation used in Zijlema [1998]. Second, horizontal and temporal discretisation will be carried out.

The vertical discretisation and σ -transformation is done by integrating the Cartesian pressure term over a layer between lines of constant sigma. The layers are numbered $k = 1$ to $k = kmax$ from the bottom to the surface.

The hydrostatic pressure is given by (cf. Equation (F.13)):

$$p_h(x, z, t) = p_{atm} + g \int_z^{\zeta(x, t)} \rho(x, z', t) dz' \quad (G.1)$$

By dividing the vertical in a fixed number of layers (with arbitrary thickness) and averaging the density in a layer, Equation (G.1) can be rewritten as:

$$p = p_{atm} + g \int_z^{z_k} \rho dz' + g \sum_{j=k+1}^{kmax} \int_{z_{j-1}}^{z_j} \rho dz' = p_{atm} + g \int_z^{z_k} \rho dz' + g \sum_{j=k+1}^{kmax} h_j \bar{\rho}_j \quad (G.2)$$

with:

$$\bar{\rho}_k = \frac{1}{h_k} \int_{z_{k-1}}^{z_k} \rho dz'$$

$$h_k = z_k - z_{k-1}$$

Now we differentiate in the x -direction and integrate over layer k . (Thereby assuming no gradients in the atmospheric pressure.)

$$\int_{z_{k-1}}^{z_k} \frac{\partial p}{\partial x} dz'' = g \int_{z_{k-1}}^{z_k} \frac{\partial}{\partial x} \int_z^{z_k} \rho dz' dz'' + g h_k \sum_{j=k+1}^{kmax} \left[h_j \frac{\partial \bar{\rho}_j}{\partial x} + \bar{\rho}_j \frac{\partial h_j}{\partial x} \right] \quad (G.3)$$

We take the first term on the right hand side of (G.3) and rewrite it with use of Leibniz's rule (cf. Equation (F.16)).

$$\begin{aligned} g \int_{z_{k-1}}^{z_k} \frac{\partial}{\partial x} \int_z^{z_k} \rho dz' dz'' &= g \int_{z_{k-1}}^{z_k} \rho \frac{\partial z_k}{\partial x} dz'' + g \int_{z_{k-1}}^{z_k} \int_z^{z_k} \frac{\partial \rho}{\partial x} dz' dz'' \\ &= g h_k \bar{\rho}_k \frac{\partial z_k}{\partial x} + g \int_{z_{k-1}}^{z_k} (z_k - z'') \frac{\partial \bar{\rho}_k}{\partial x} dz'' \\ &= g h_k \bar{\rho}_k \frac{\partial \zeta}{\partial x} - g h_k \bar{\rho}_k \sum_{j=k+1}^{kmax} \frac{\partial h_j}{\partial x} + \frac{1}{2} g h_k^2 \frac{\partial \bar{\rho}_k}{\partial x} \end{aligned} \quad (G.4)$$

We have used:

$$z_k = \zeta - \sum_{j=k+1}^{kmax} h_j$$

$$\int_{z_{k-1}}^{z_k} (z_k - z'') dz'' = \int_{z_k}^{z_{k-1}} (z_k - z'') d(z_k - z'') = \frac{1}{2} (z_k - z'')^2 \Big|_{z_k}^{z_{k-1}} = \frac{1}{2} (z_k - z_{k-1})^2 = \frac{1}{2} h_k^2$$

Substituting Equation (G.4) and dividing by h_k gives the final expression for the layer averaged (horizontal) pressure gradient:

$$\frac{1}{h_k} \int_{z_{k-1}}^{z_k} \frac{\partial p}{\partial x} dz'' = g \left\{ \bar{\rho}_k \frac{\partial \zeta}{\partial x} + \frac{1}{2} h_k \frac{\partial \bar{\rho}_k}{\partial x} + \sum_{j=k+1}^{kmax} \left[h_j \frac{\partial \bar{\rho}_j}{\partial x} + (\bar{\rho}_j - \bar{\rho}_k) \frac{\partial h_j}{\partial x} \right] \right\} \quad (G.5)$$

The hydrostatic horizontal pressure gradient term can be split up in two parts, the barotropic pressure term,

$$g \bar{\rho}_k \frac{\partial \zeta}{\partial x}, \quad (G.6)$$

which represents the gradient in hydrostatic pressure due to variations in the water level, and the baroclinic pressure term

$$g \frac{1}{2} h_k \frac{\partial \bar{\rho}_k}{\partial x} + g \sum_{j=k+1}^{kmax} \left[h_j \frac{\partial \bar{\rho}_j}{\partial x} + (\bar{\rho}_j - \bar{\rho}_k) \frac{\partial h_j}{\partial x} \right], \quad (G.7)$$

which represents the gradient in hydrostatic pressure due to variations in the density.

Dividing by Equation (G.7) by ρ_0 and discretising the baroclinic pressure gradient in the horizontal direction, gives the discrete baroclinic pressure term:

$$\frac{g}{\rho_0} \left\{ \frac{1}{2} \tilde{h}_{m,k}^n \frac{\rho_{m+1,k}^n - \rho_{m,k}^n}{\Delta x} + \sum_{j=k+1}^{kmax} \left[\tilde{h}_{m,j}^n \frac{\rho_{m+1,j}^n - \rho_{m,j}^n}{\Delta x} + (\tilde{\rho}_{m,j}^n - \tilde{\rho}_{m,k}^n) \frac{h_{m+1,j}^n - h_{m,j}^n}{\Delta x} \right] \right\} \quad (G.8)$$

with:

$$\tilde{h}_{m,k} = h_{\mu(m),k}$$

$$\tilde{h}_{m,k} = h_{\mu(m),k}$$

As can be seen, $\tilde{h}_{m,k}$ is discretised using an upstream approach.

Figures

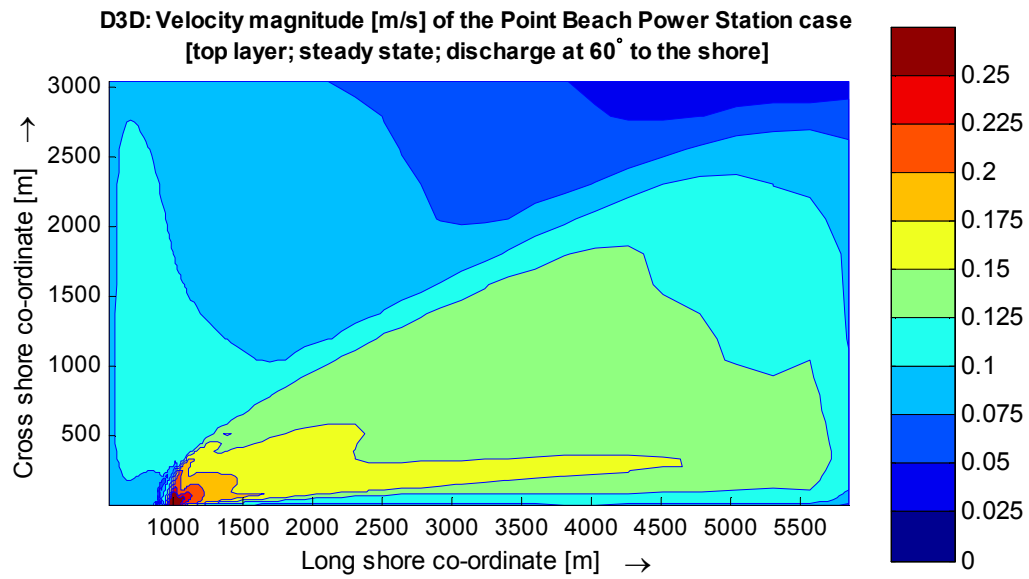


Figure 5-18: *Velocity magnitude [$m s^{-1}$] at the surface of the Point Beach Power Station case. Discharge at 60° to the shore, Delft3D simulation.*

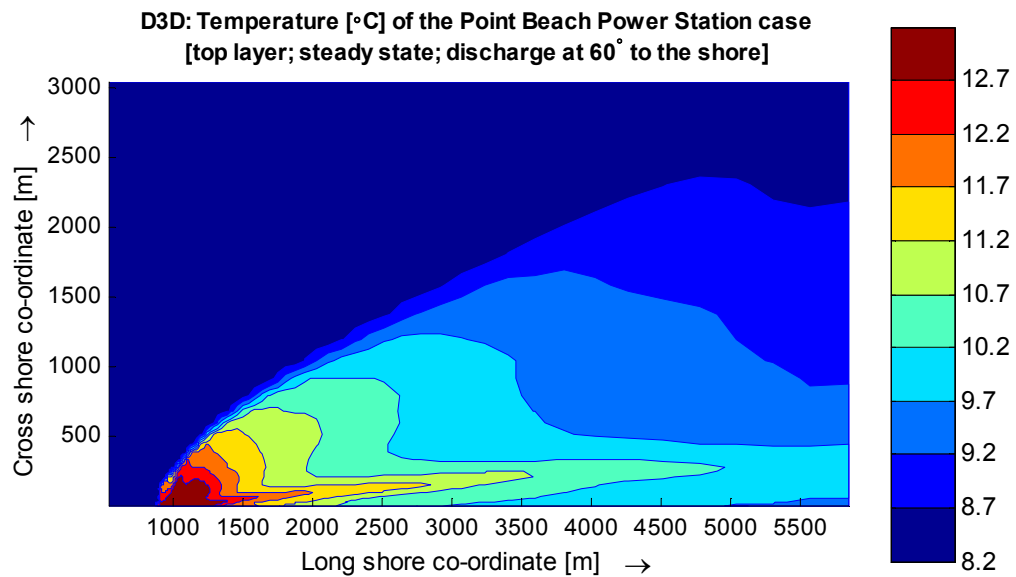


Figure 5-19: *Temperature [°C] at the surface of the Point Beach Power Station case. Discharge at 60° to the shore, Delft3D simulation.*

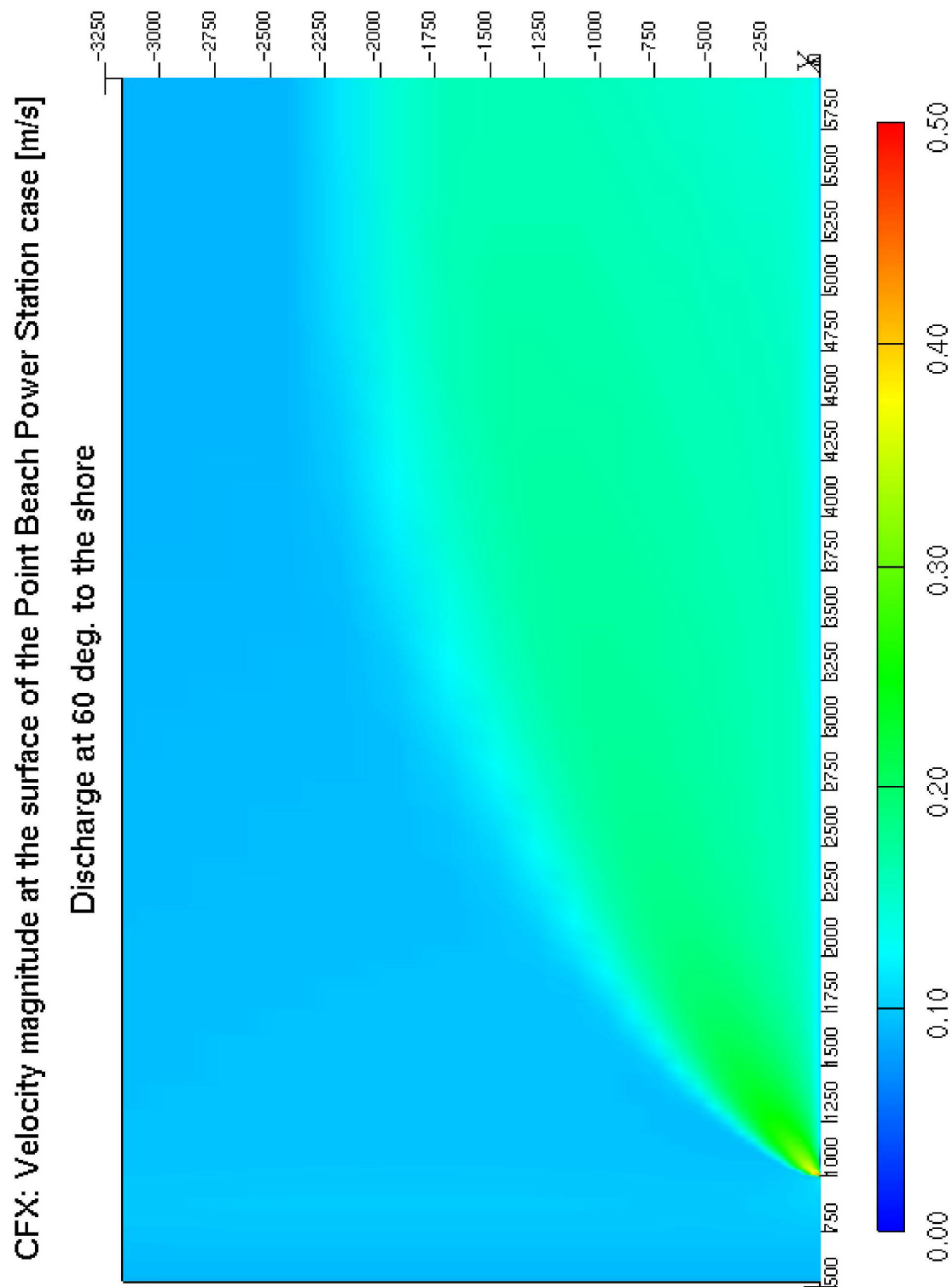


Figure 5-20: *Velocity magnitude [m s⁻¹] at the surface of the Point Beach Power Station case. Discharge at 60° to the shore, CFX simulation.*

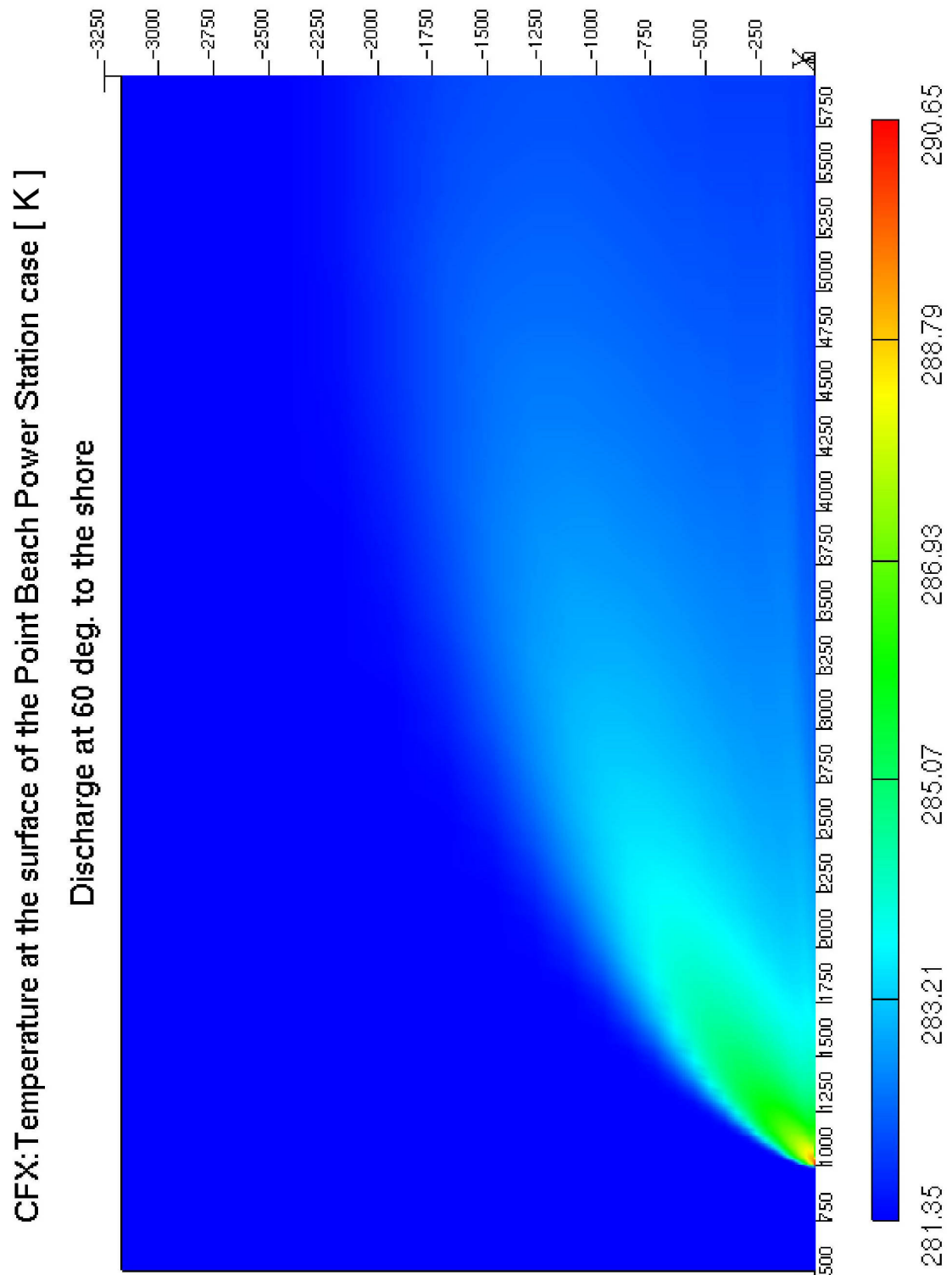


Figure 5-21: *Temperature [K] at the surface of the Point Beach Power Station case. Discharge at 60° to the shore, CFX simulation.*

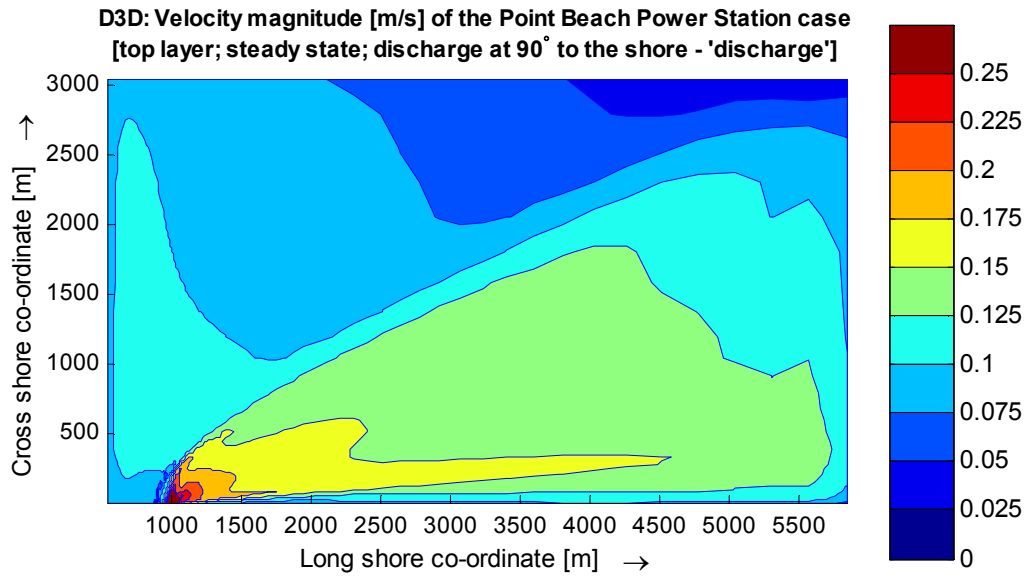


Figure 5-22: *Velocity magnitude [$m s^{-1}$] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore using a source term to simulate the discharge, Delft3D simulation.*

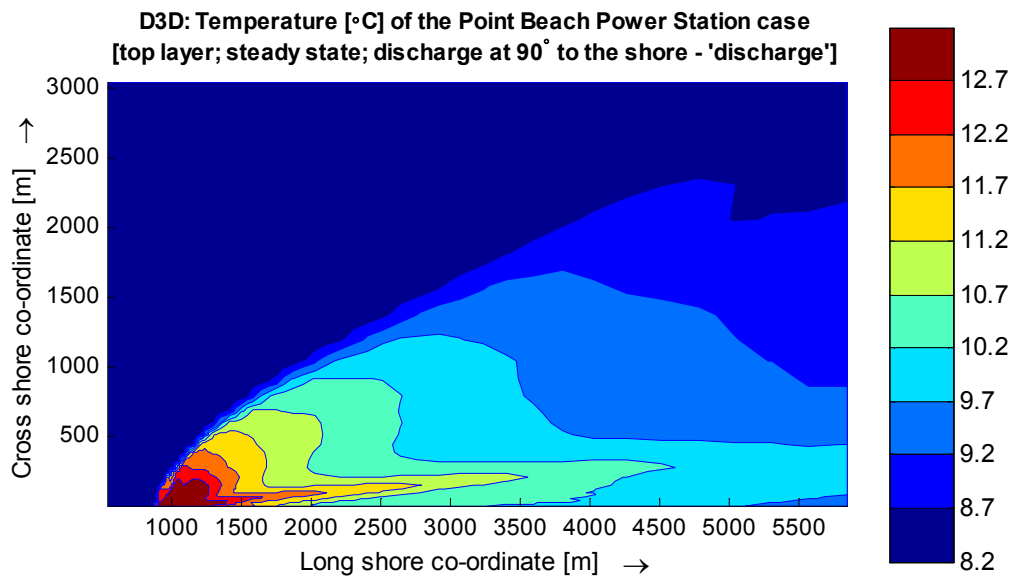


Figure 5-23: *Temperature [°C] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore using a source term to simulate the discharge, Delft3D simulation.*

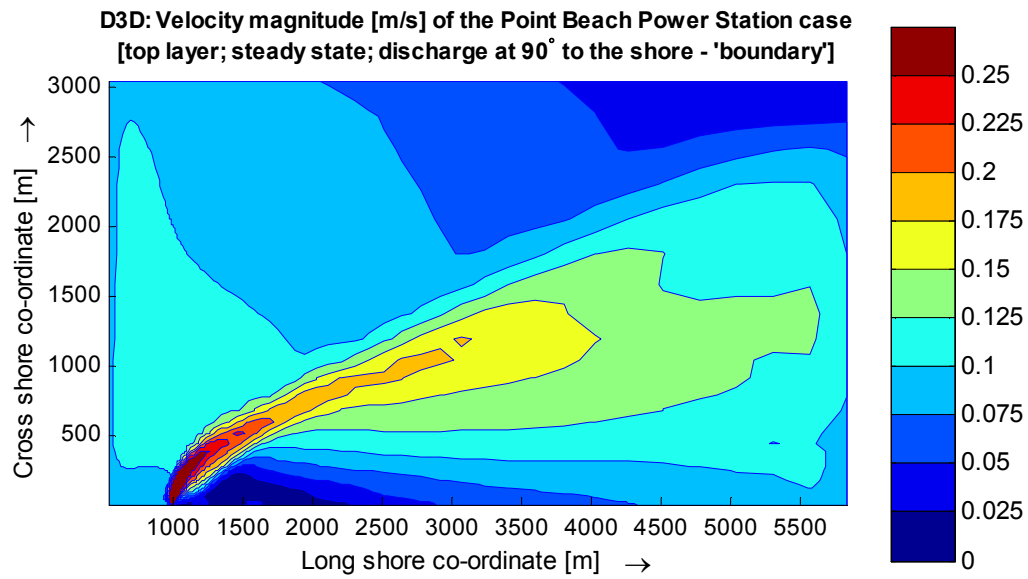


Figure 5-24: *Velocity magnitude [m s^{-1}] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore using a boundary condition to simulate the discharge, Delft3D simulation.*

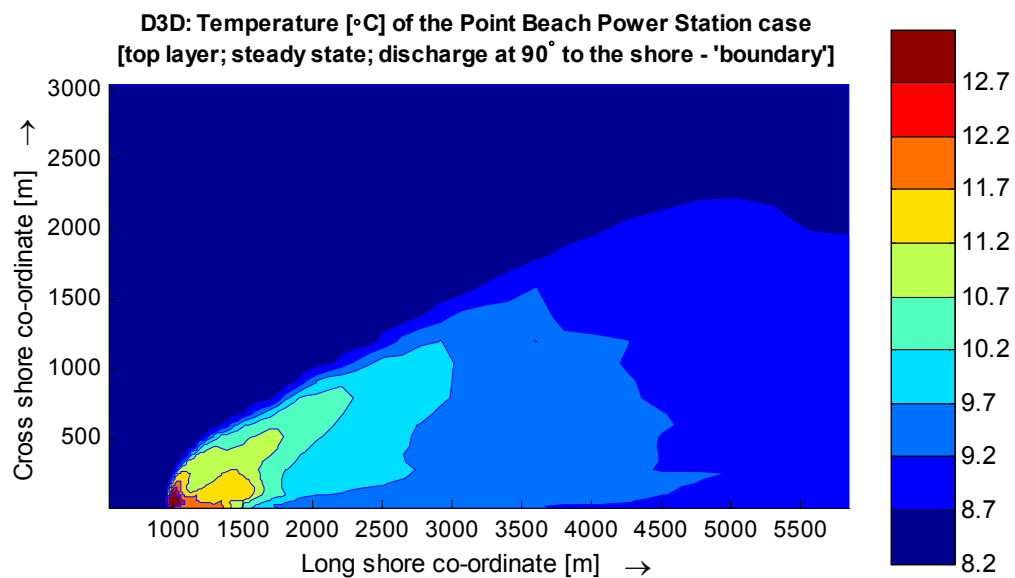


Figure 5-25: *Temperature [°C] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore using a boundary condition to simulate the discharge, Delft3D simulation.*

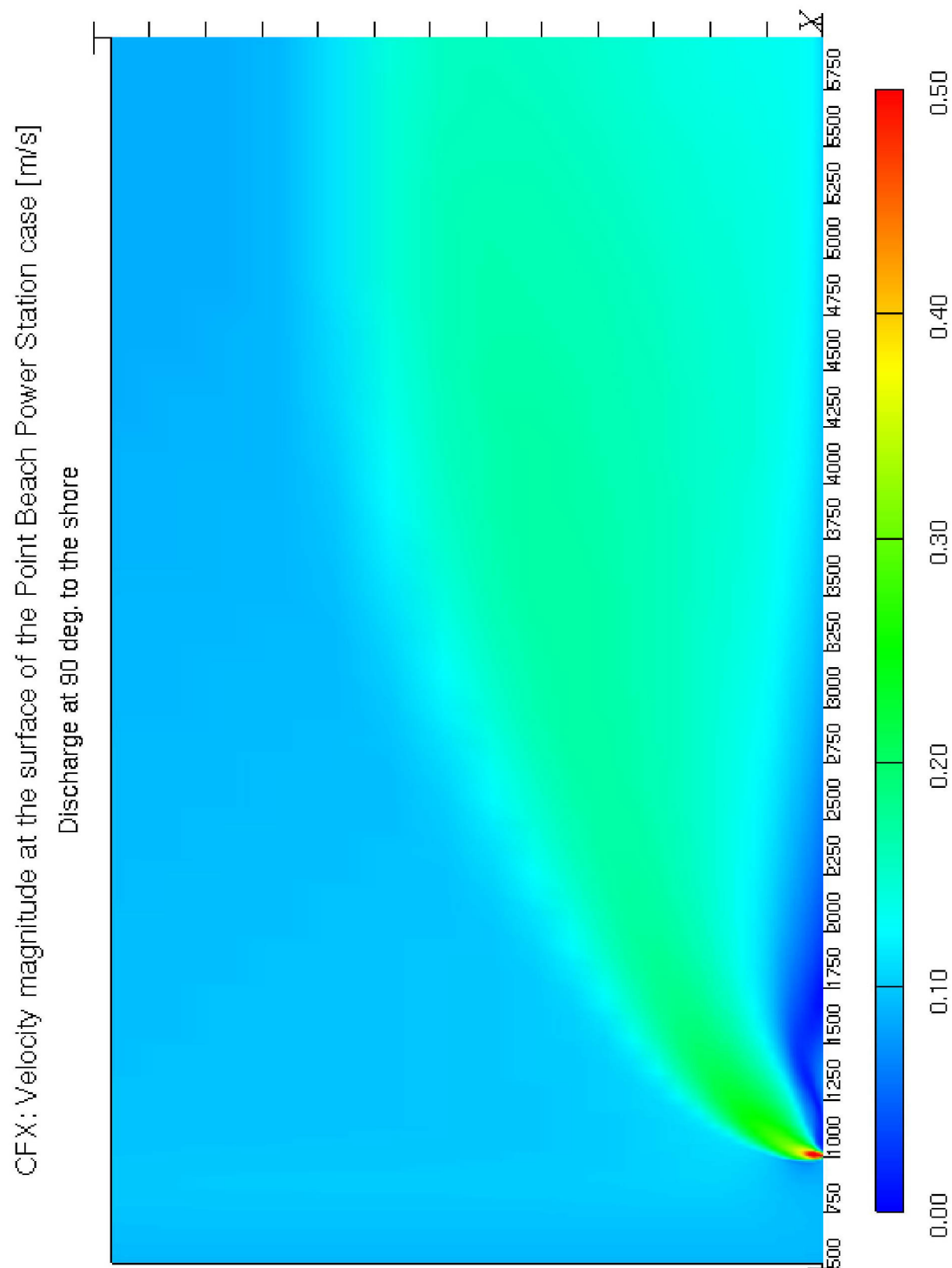


Figure 5-26: *Velocity magnitude [$m s^{-1}$] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore, CFX simulation.*

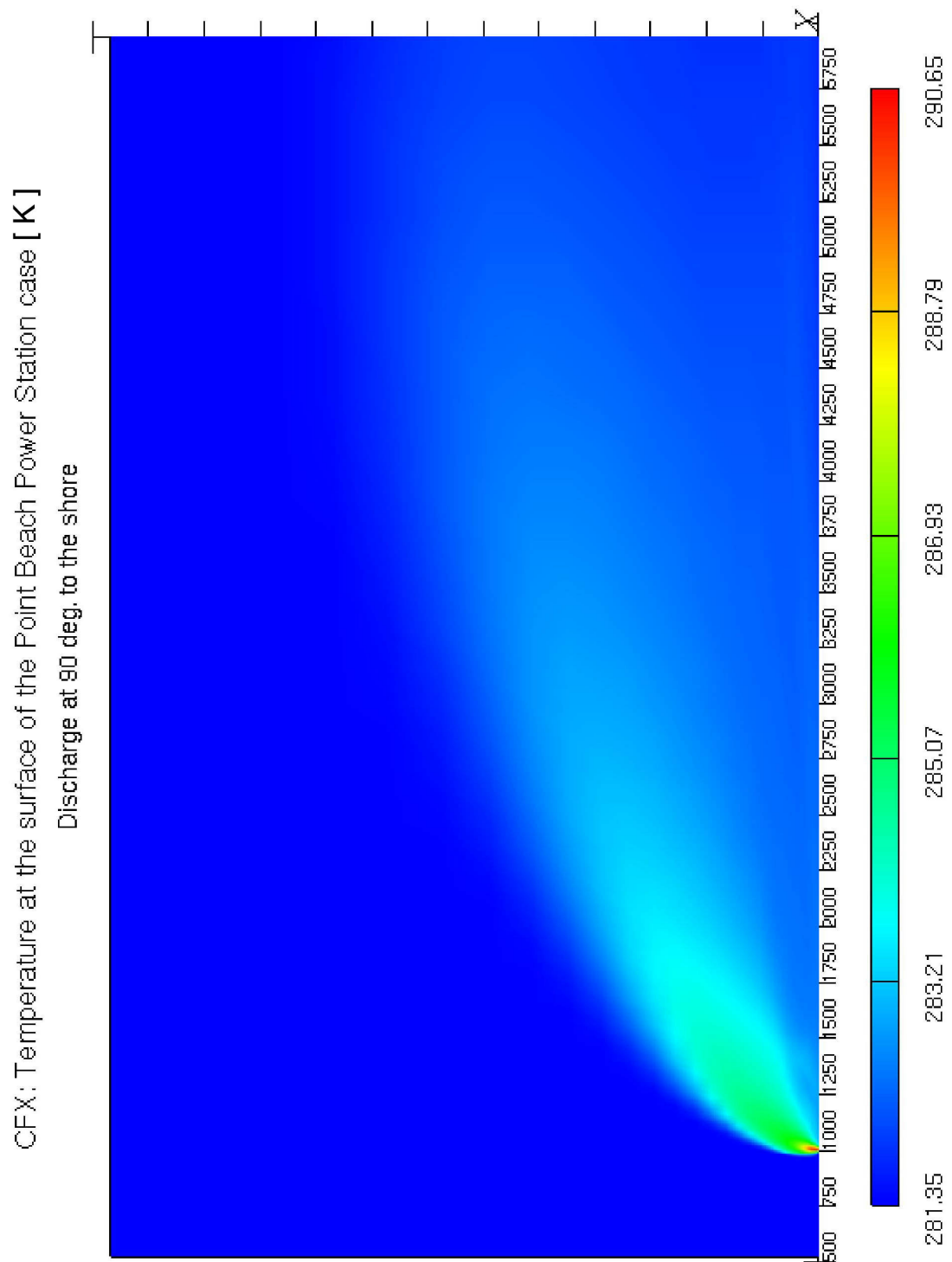


Figure 5-27: *Temperature [K] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore, CFX simulation.*

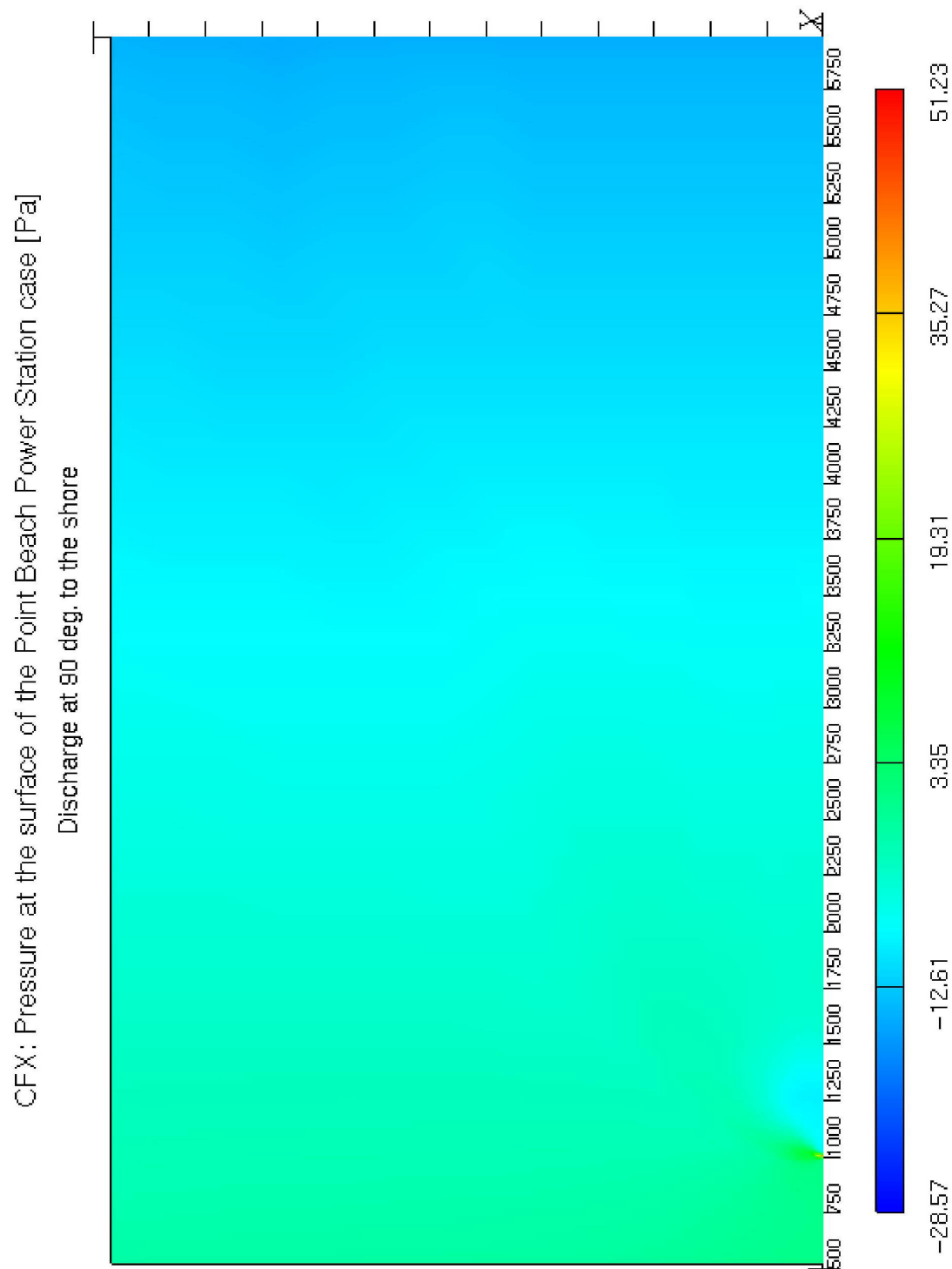


Figure 5-28: *Pressure [Pa] at the surface of the Point Beach Power Station case. Discharge at 90° to the shore, CFX simulation.*

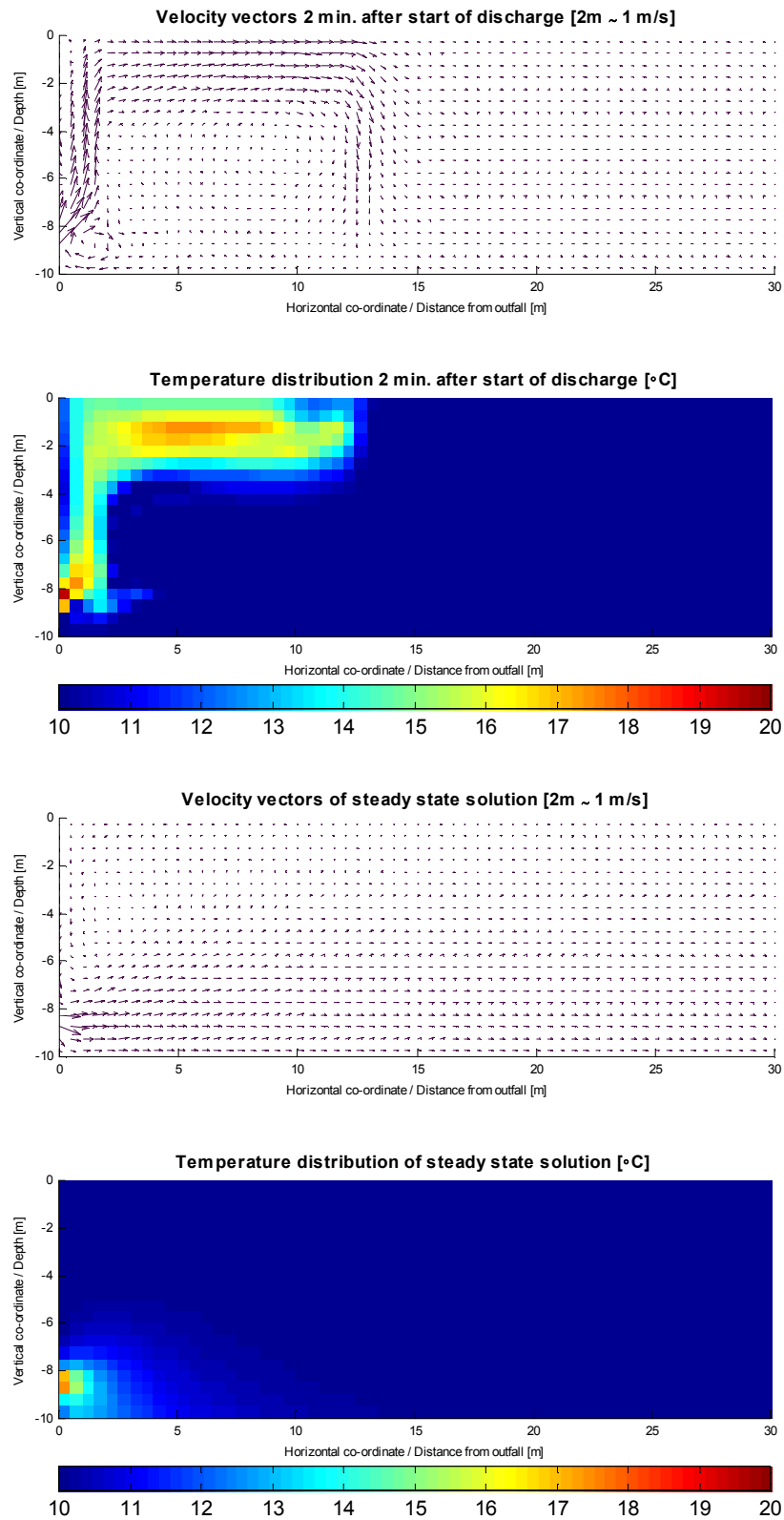


Figure 6-5: *Results of 2DV buoyant jet simulation in Delft3D (Run1; reference model).*

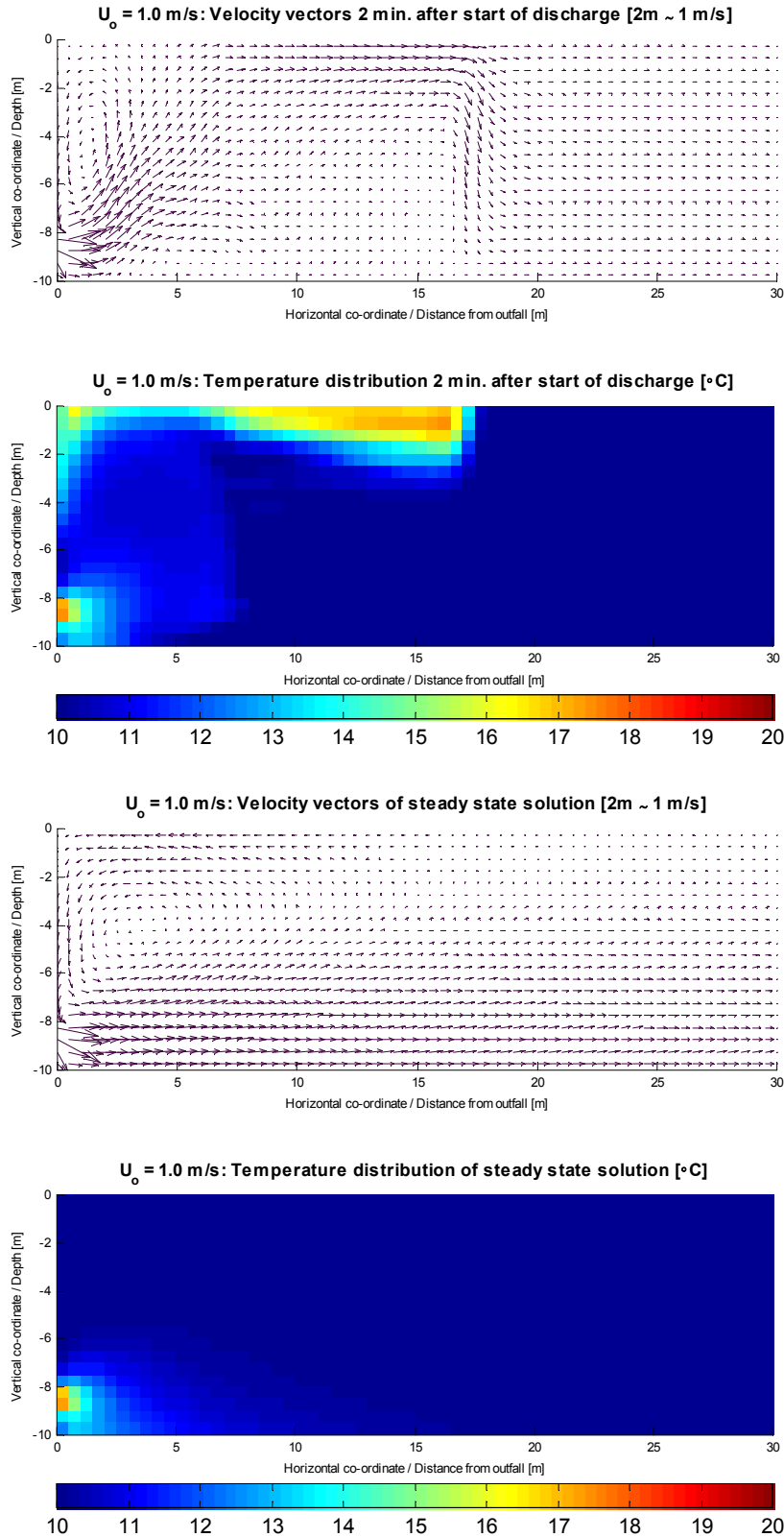


Figure 6-6: *Results of 2DV buoyant jet simulation in Delft3D (Run2; increased discharge velocity).*

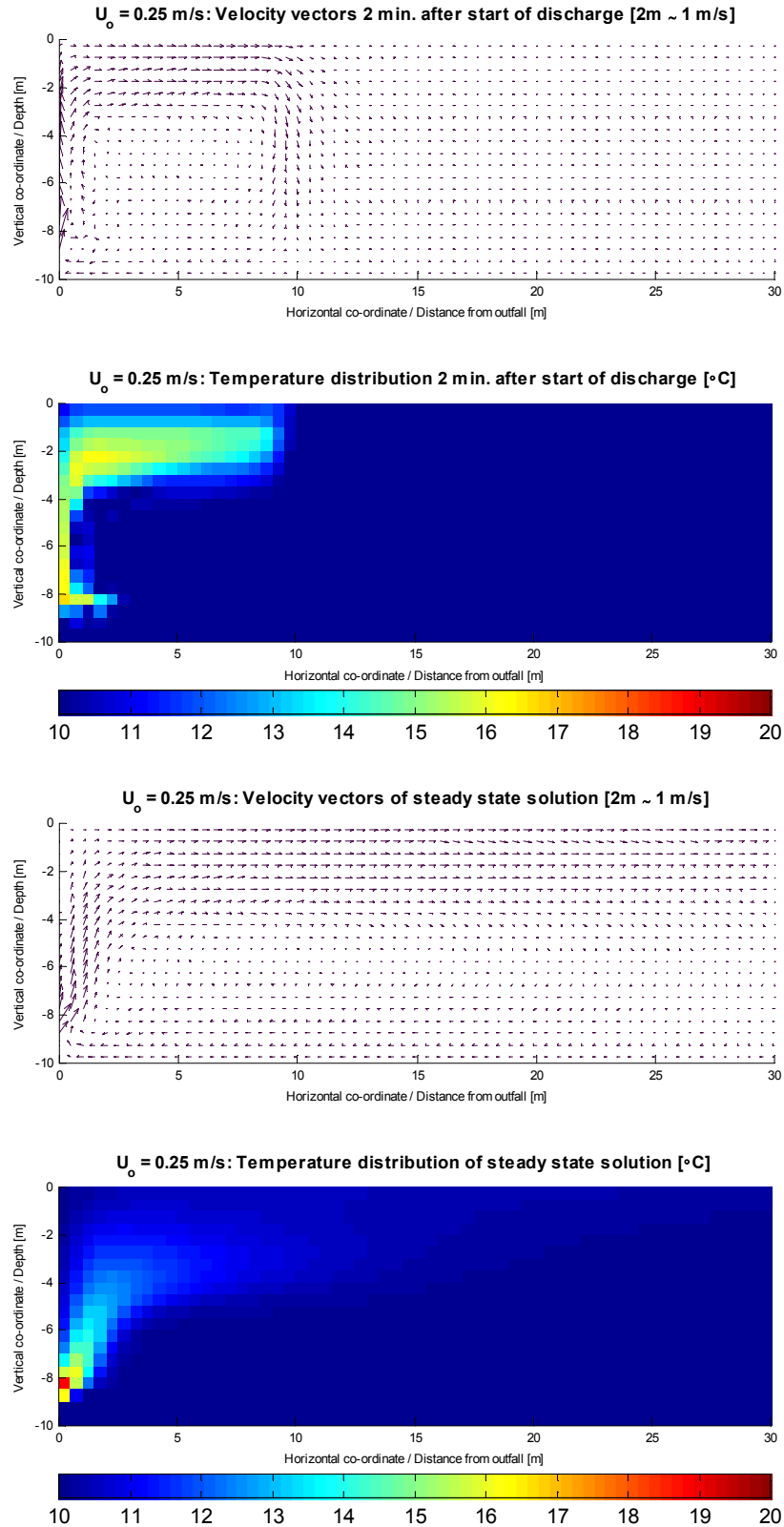


Figure 6-7: *Results of 2DV buoyant jet simulation in Delft3D (Run3; decreased discharge velocity).*

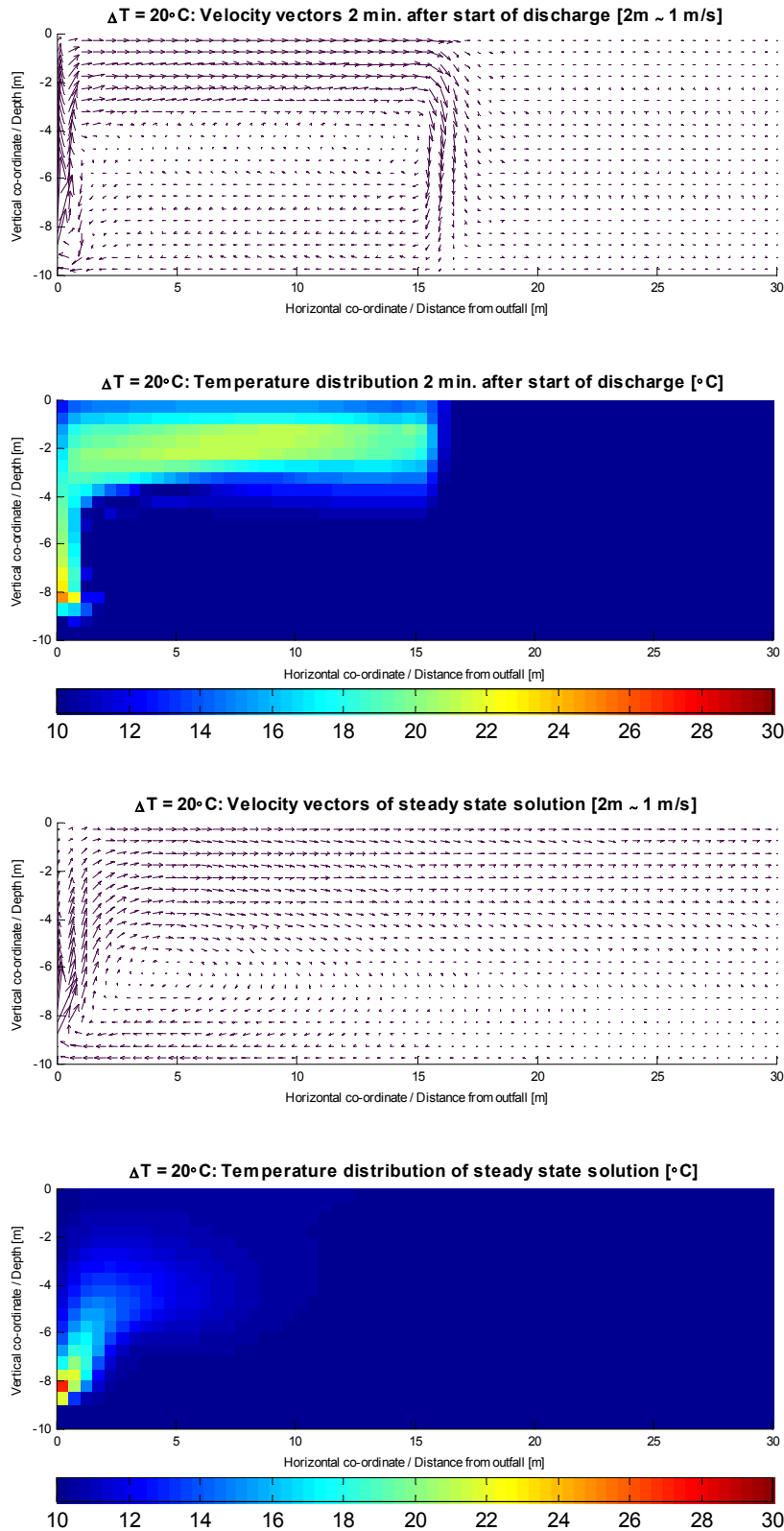


Figure 6-8: *Results of 2DV buoyant jet simulation in Delft3D (Run4; increased temperature difference).*

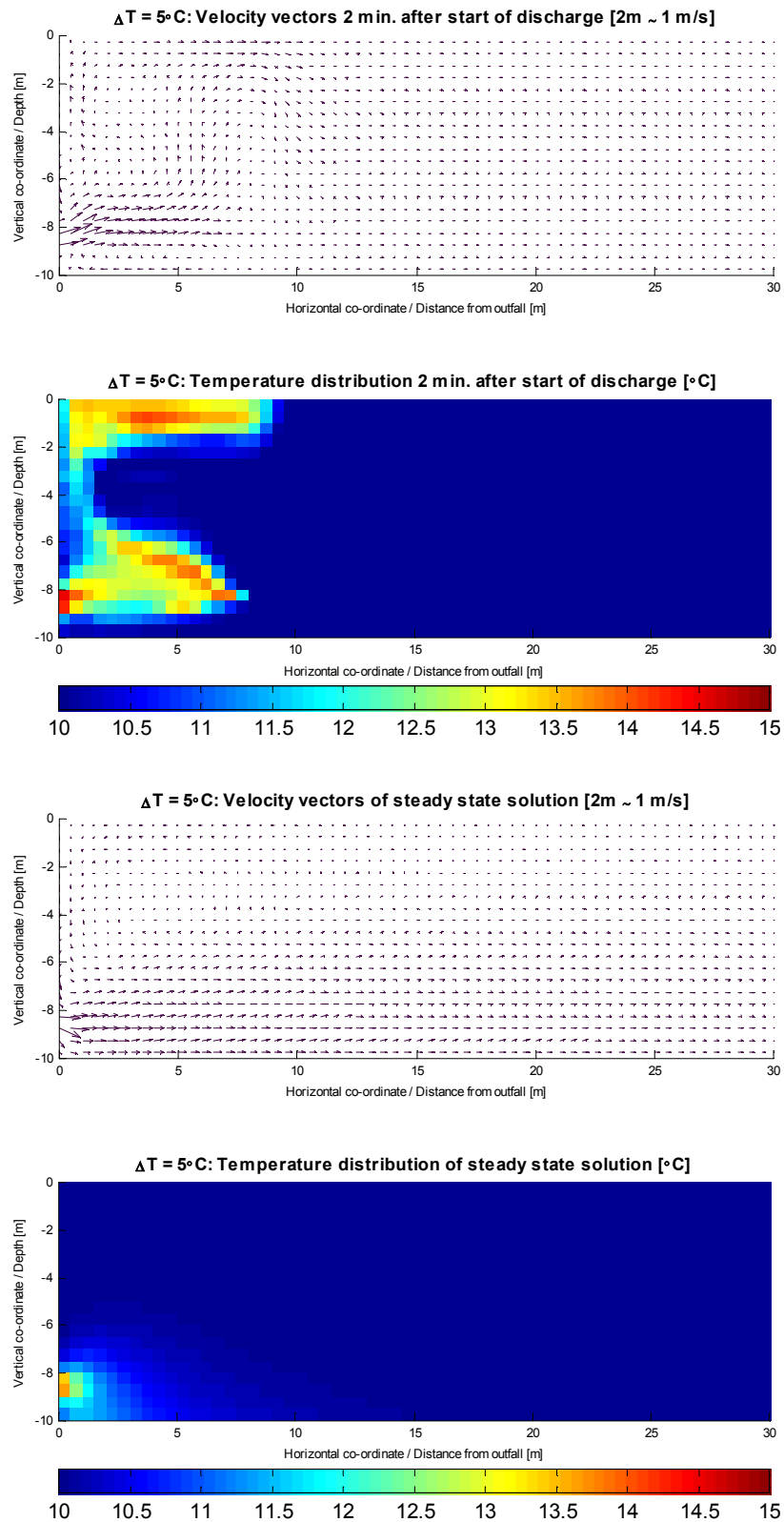


Figure 6-9: *Results of 2DV buoyant jet simulation in Delft3D (Run5; decreased temperature difference).*

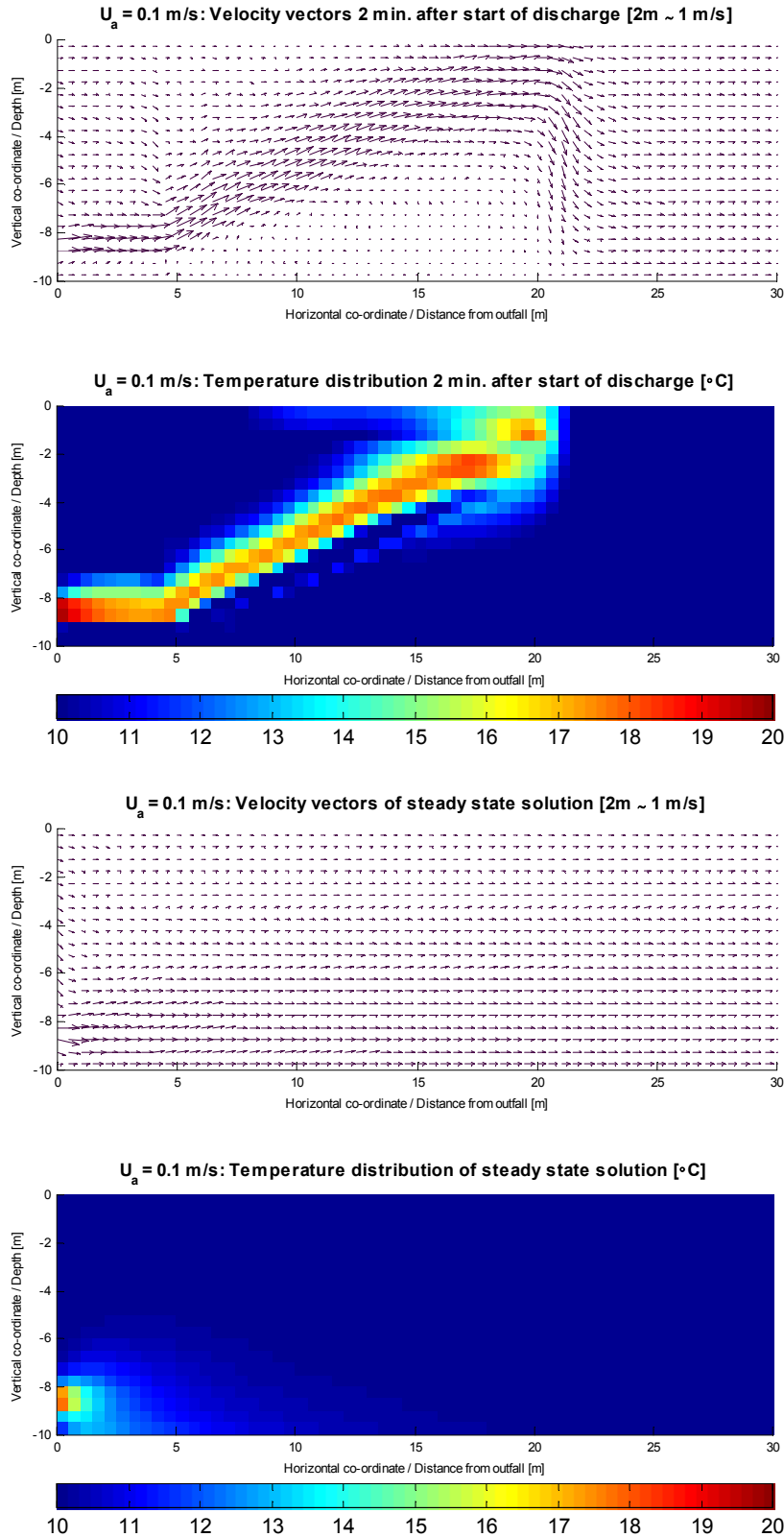


Figure 6-10: *Results of 2DV buoyant jet simulation in Delft3D (Run6; increased ambient velocity).*

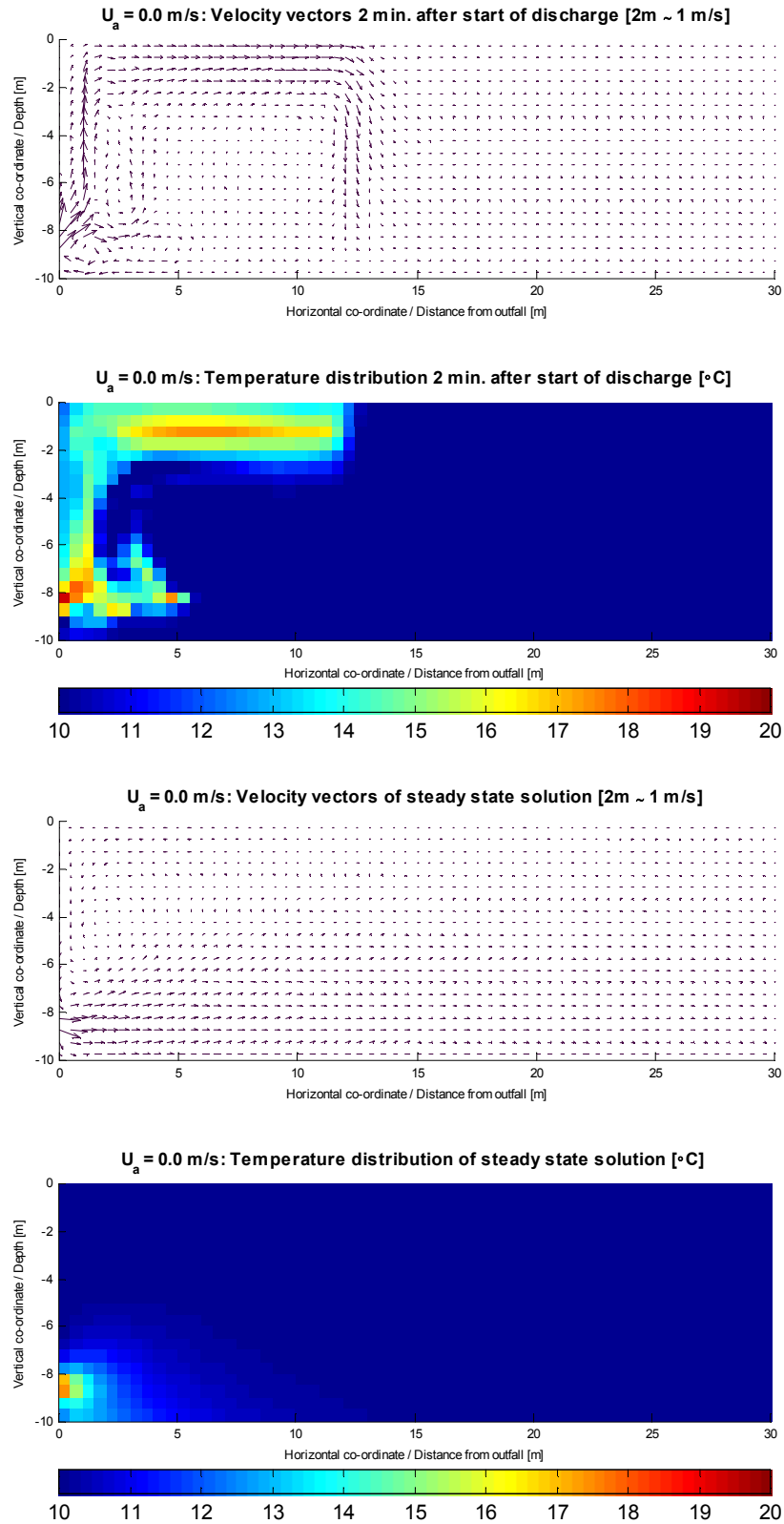


Figure 6-11: *Results of 2DV buoyant jet simulation in Delft3D (Run7; decreased ambient velocity).*

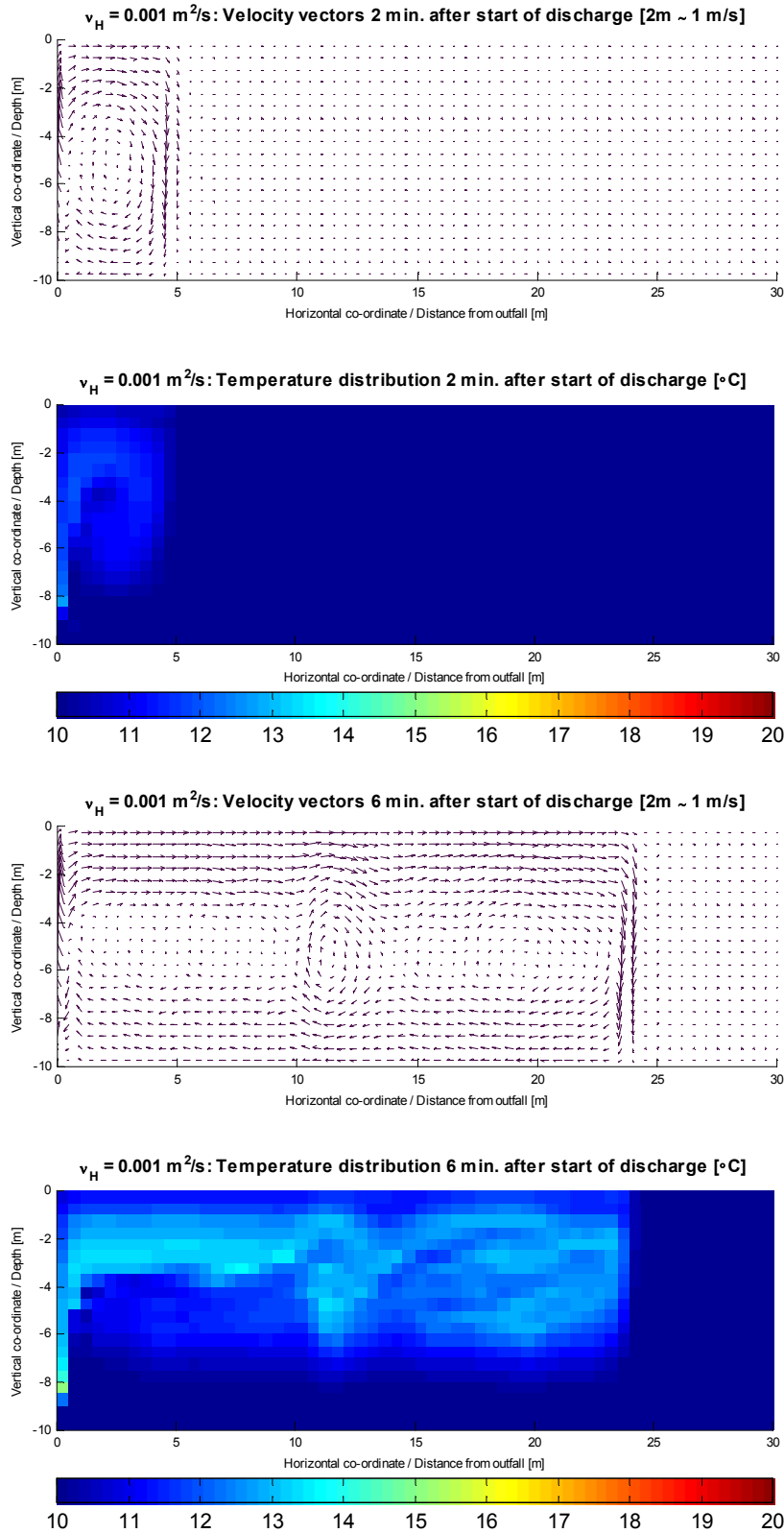


Figure 6-12a: *Results of 2DV buoyant jet simulation in Delft3D (Run8; decreased horizontal eddy viscosity).*

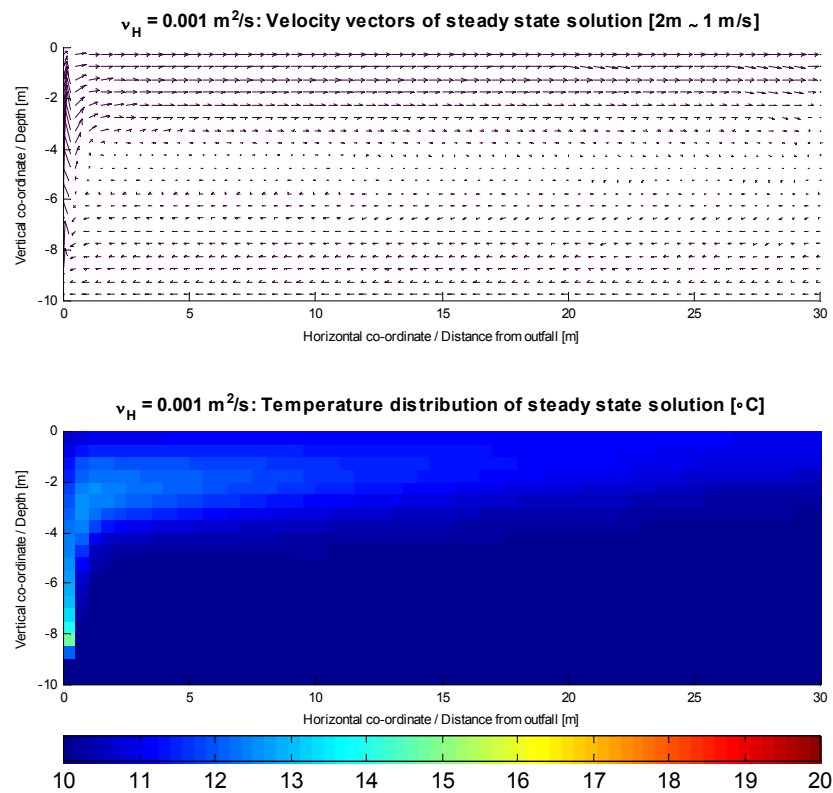


Figure 6-12b: *Results of 2DV buoyant jet simulation in Delft3D (Run8; decreased horizontal eddy viscosity).*

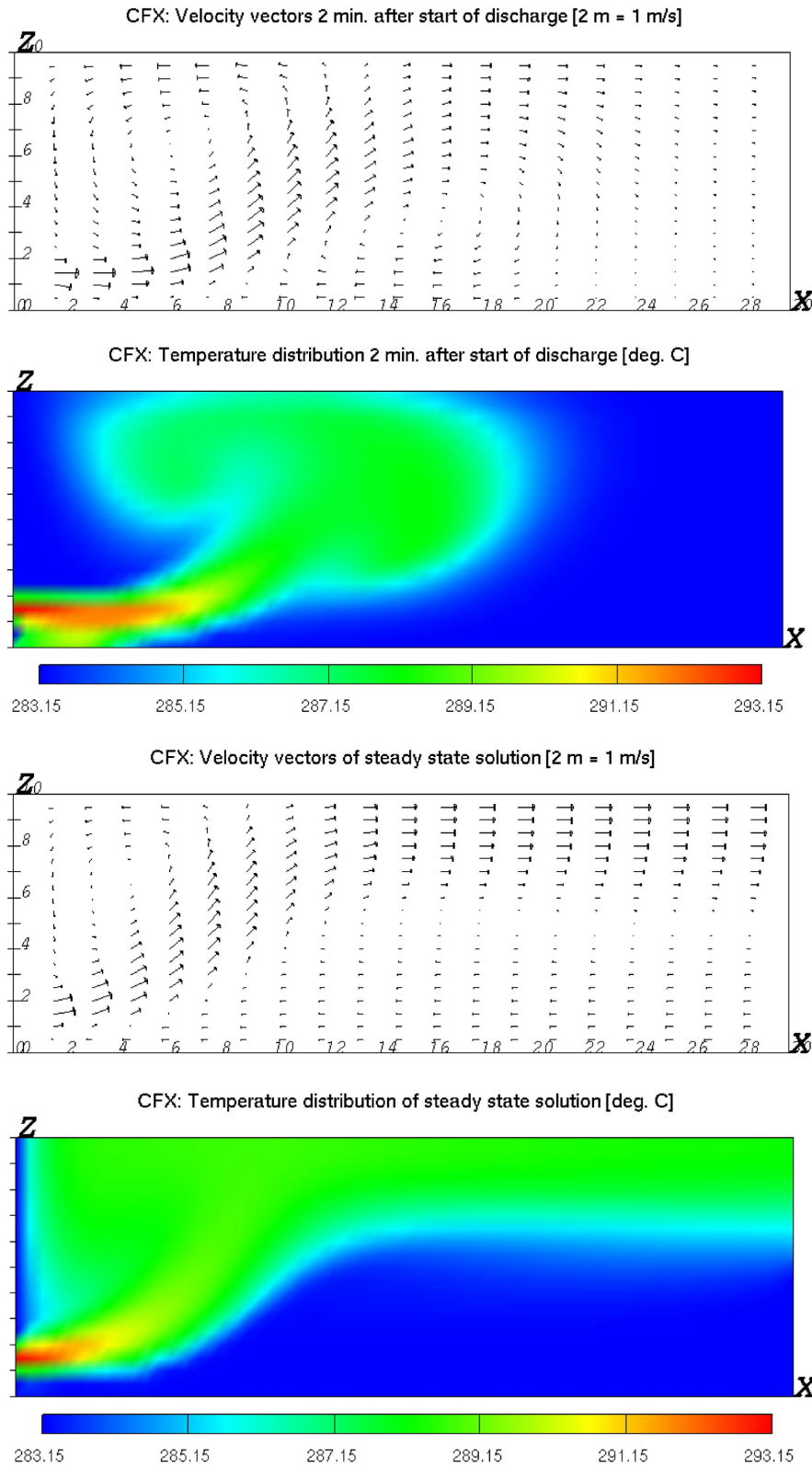


Figure 6-13a: Results of 2DV buoyant jet simulation in CFX (Run1; reference model).

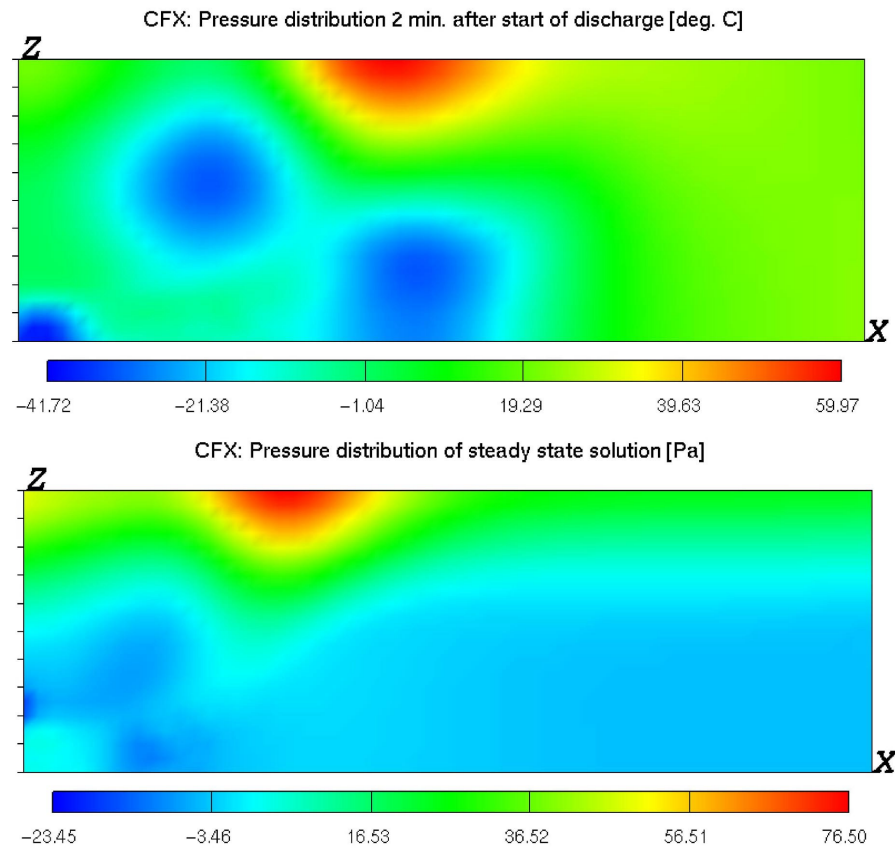


Figure 6-13b: *Results of 2DV buoyant jet simulation in CFX (Run1; reference model).*

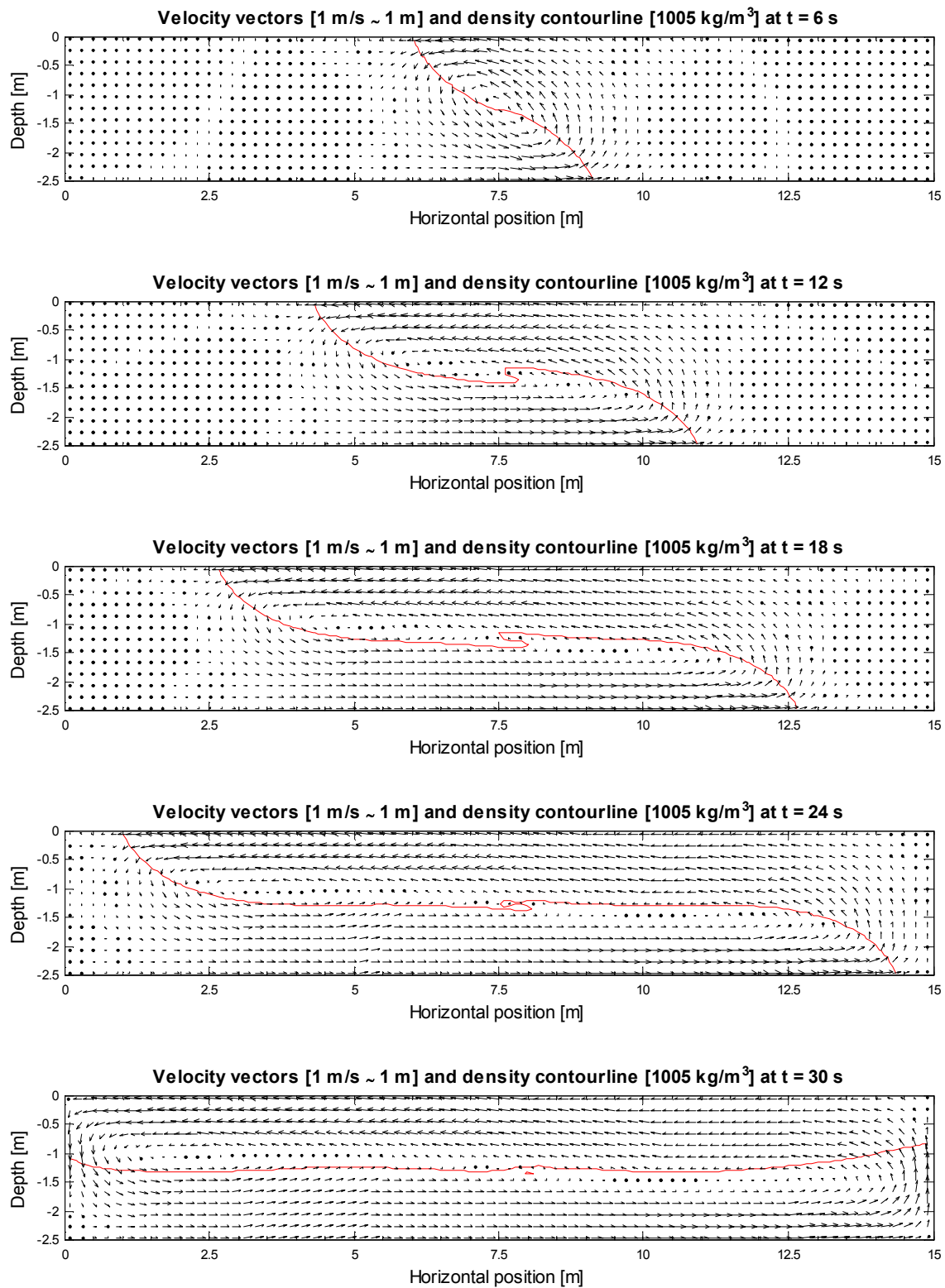


Figure 7-6: Velocity vectors and a density contourline for the Lock Exchange case at $t = 6; 12; 18; 24; 30$ s. **Non-hydrostatic** simulation.

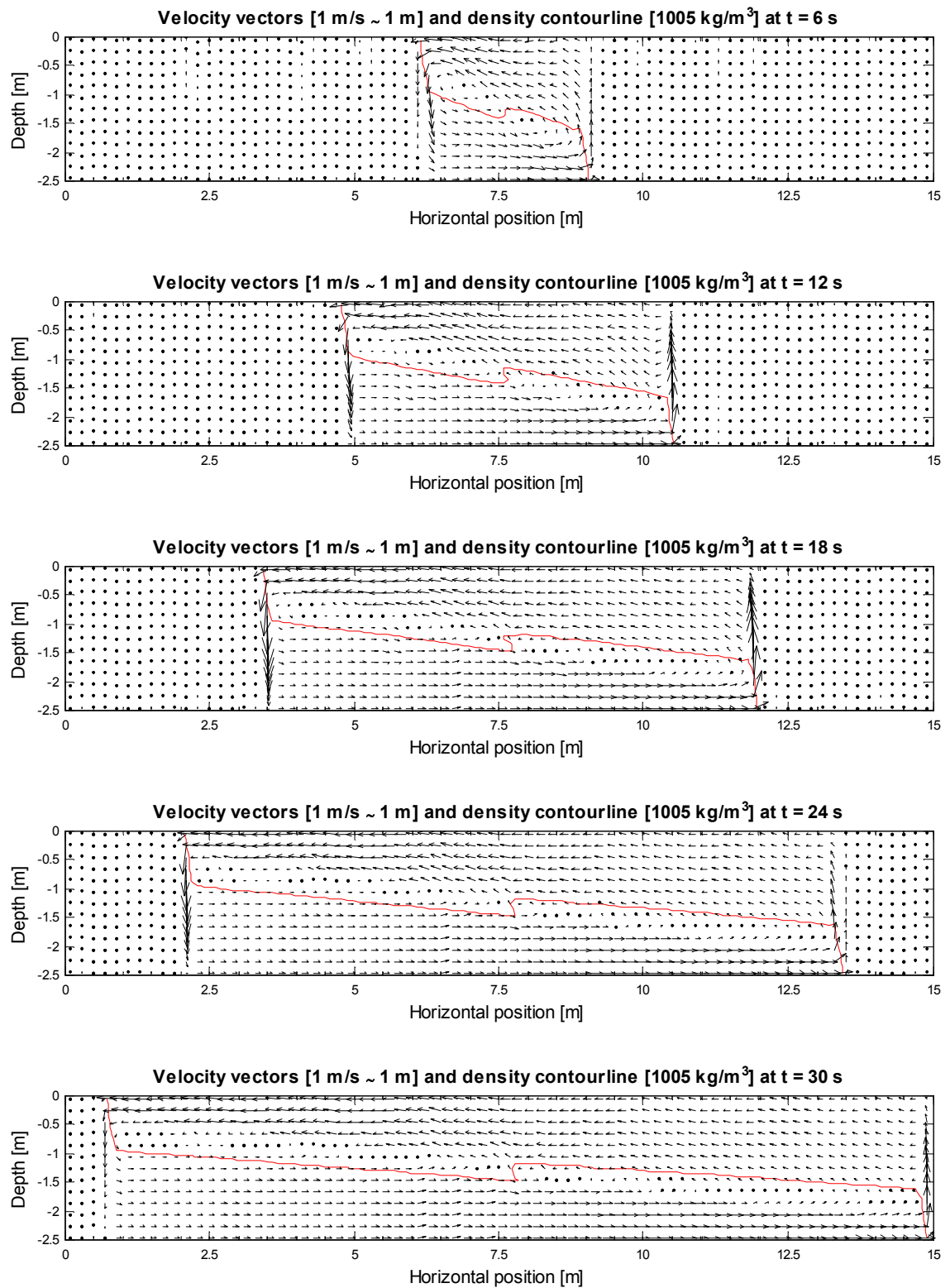


Figure 7-7: Velocity vectors and a density contourline for the Lock Exchange case at $t = 6; 12; 18; 24; 30$ s. **Hydrostatic** simulation.

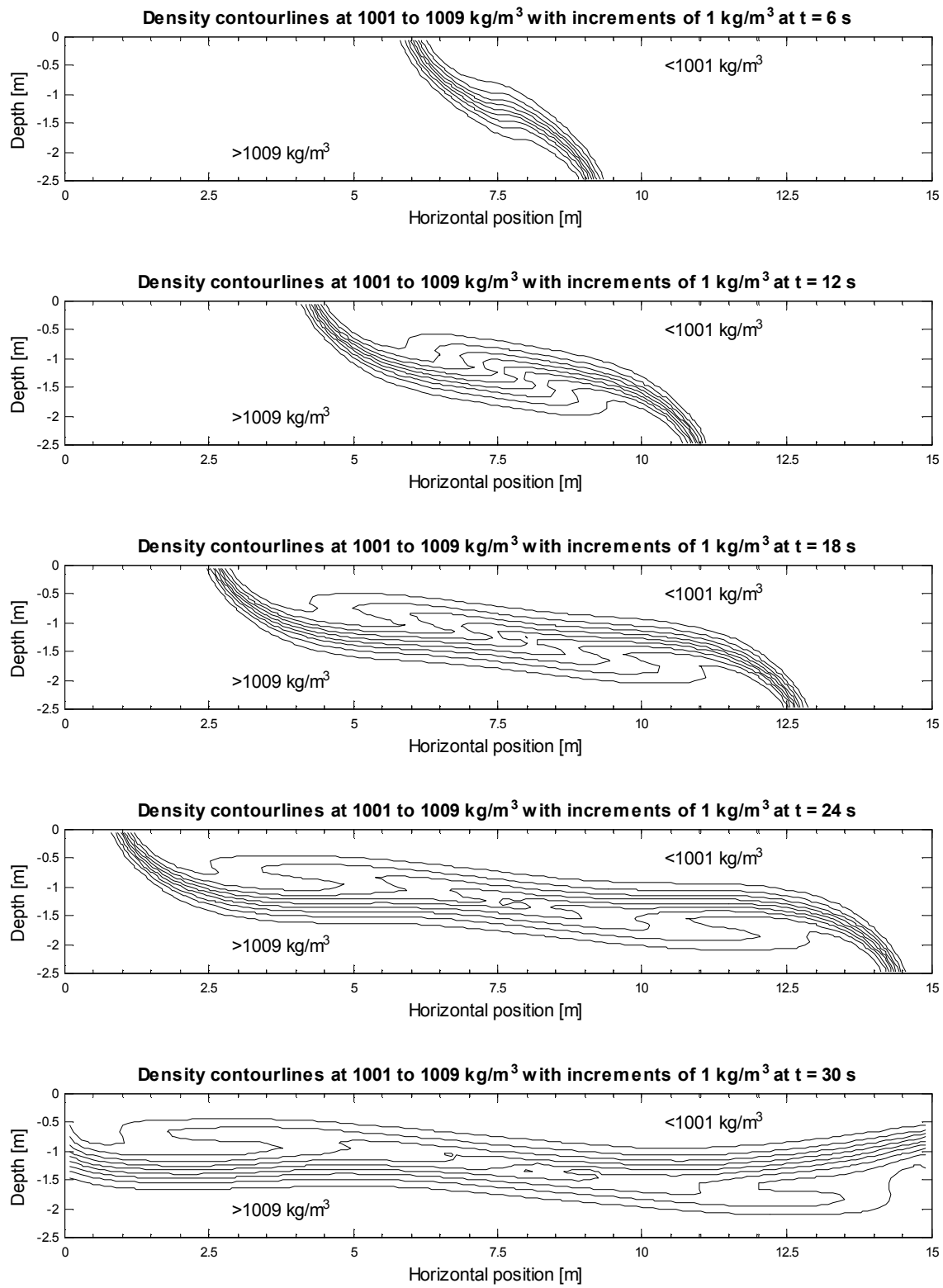


Figure 7-8: *Density contourlines for the Lock Exchange case at t = 6; 12; 18; 24; 30 s. Non-hydrostatic simulation.*

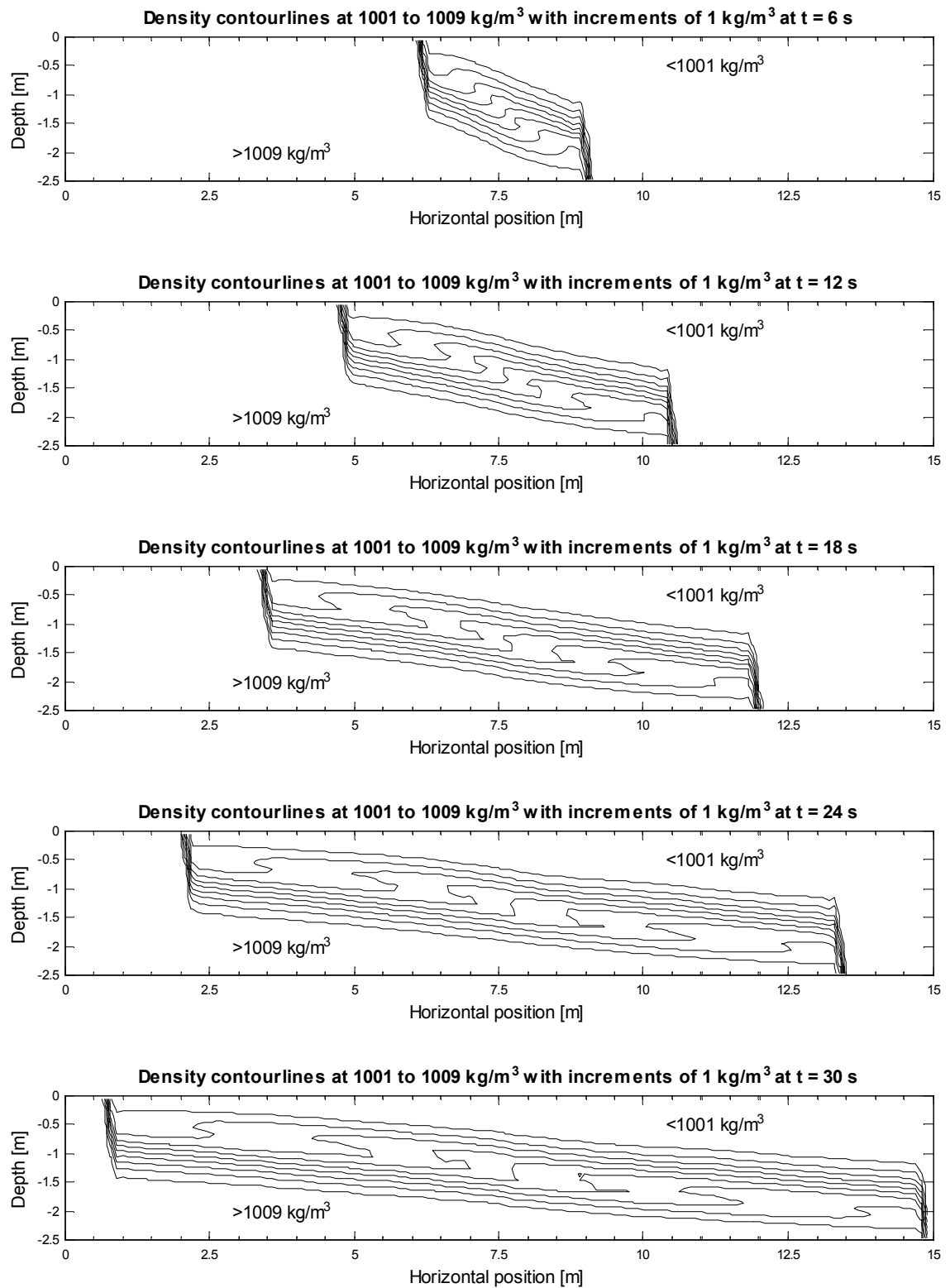


Figure 7-9: *Density contourlines for the Lock Exchange case at t = 6; 12; 18; 24; 30 s. Hydrostatic simulation.*

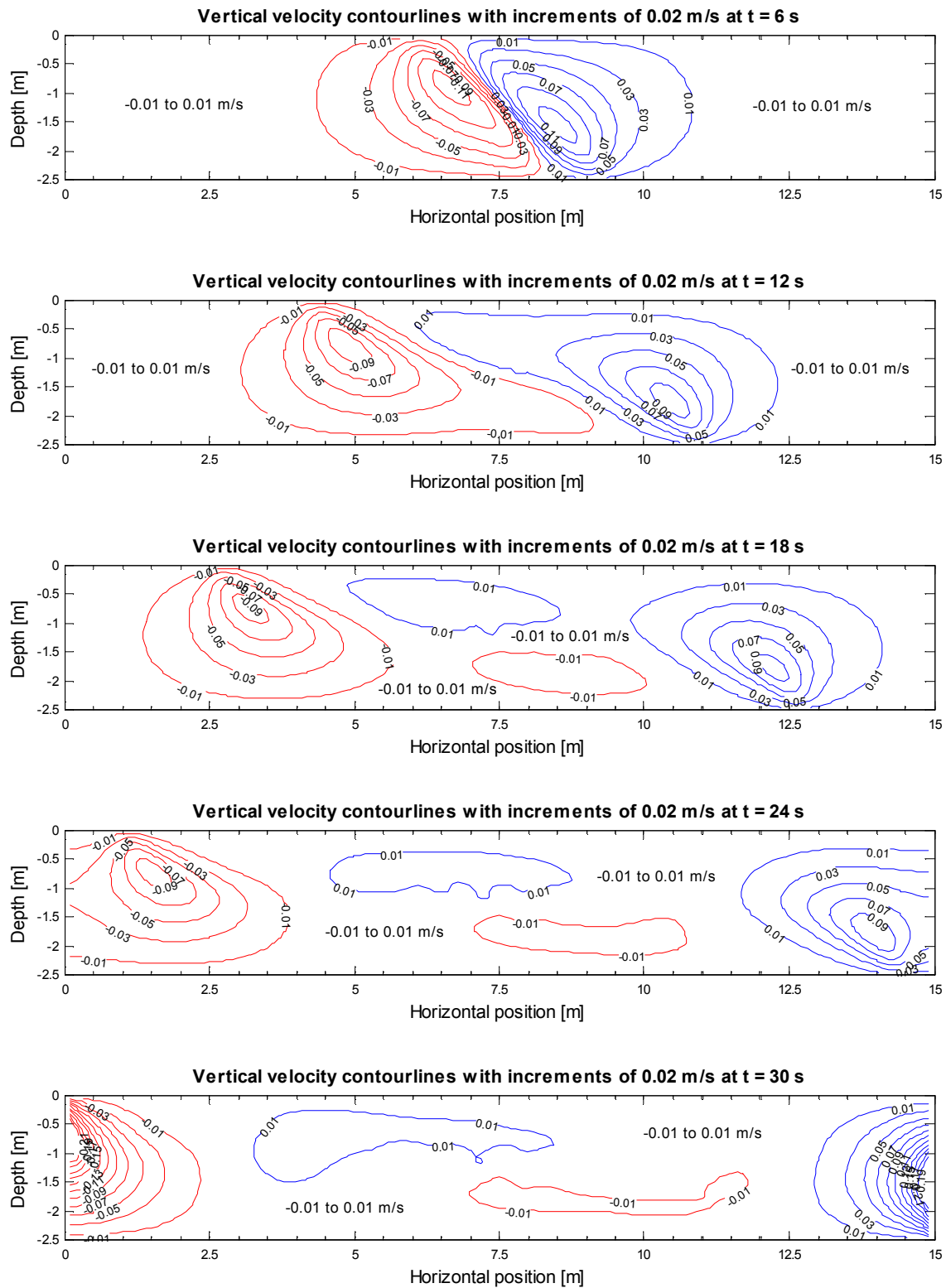


Figure 7-10: Vertical velocity contourlines for the Lock Exchange case at $t = 6$; 12; 18; 24; 30 s. The red lines mark areas with negative velocities; the blue lines mark areas with positive velocities. **Non-hydrostatic** simulation.

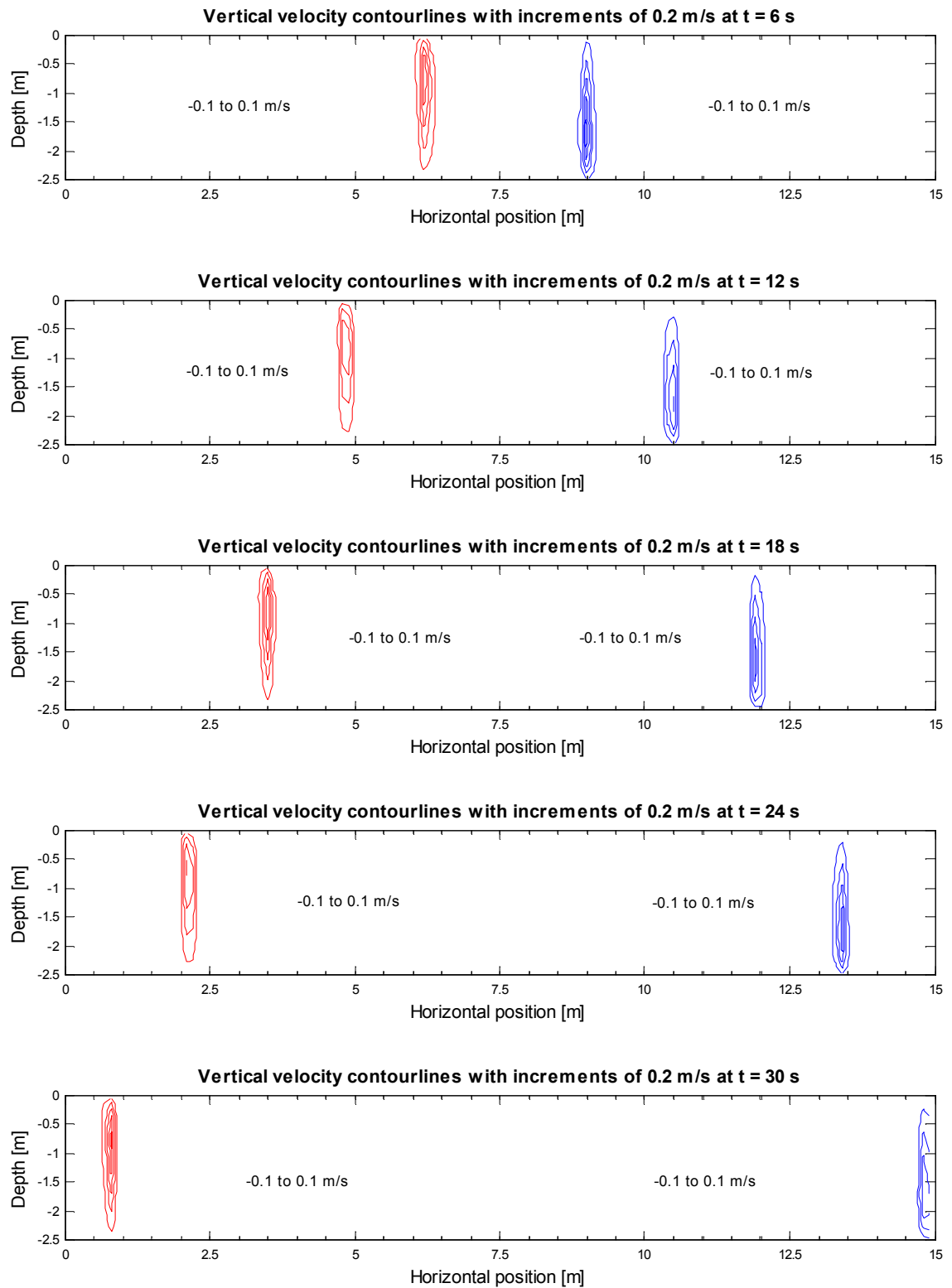


Figure 7-11: *Vertical velocity contourlines for the Lock Exchange case at t = 6; 12; 18; 24; 30 s. The red lines mark areas with negative velocities; the blue lines mark areas with positive velocities. **Hydrostatic** simulation.*

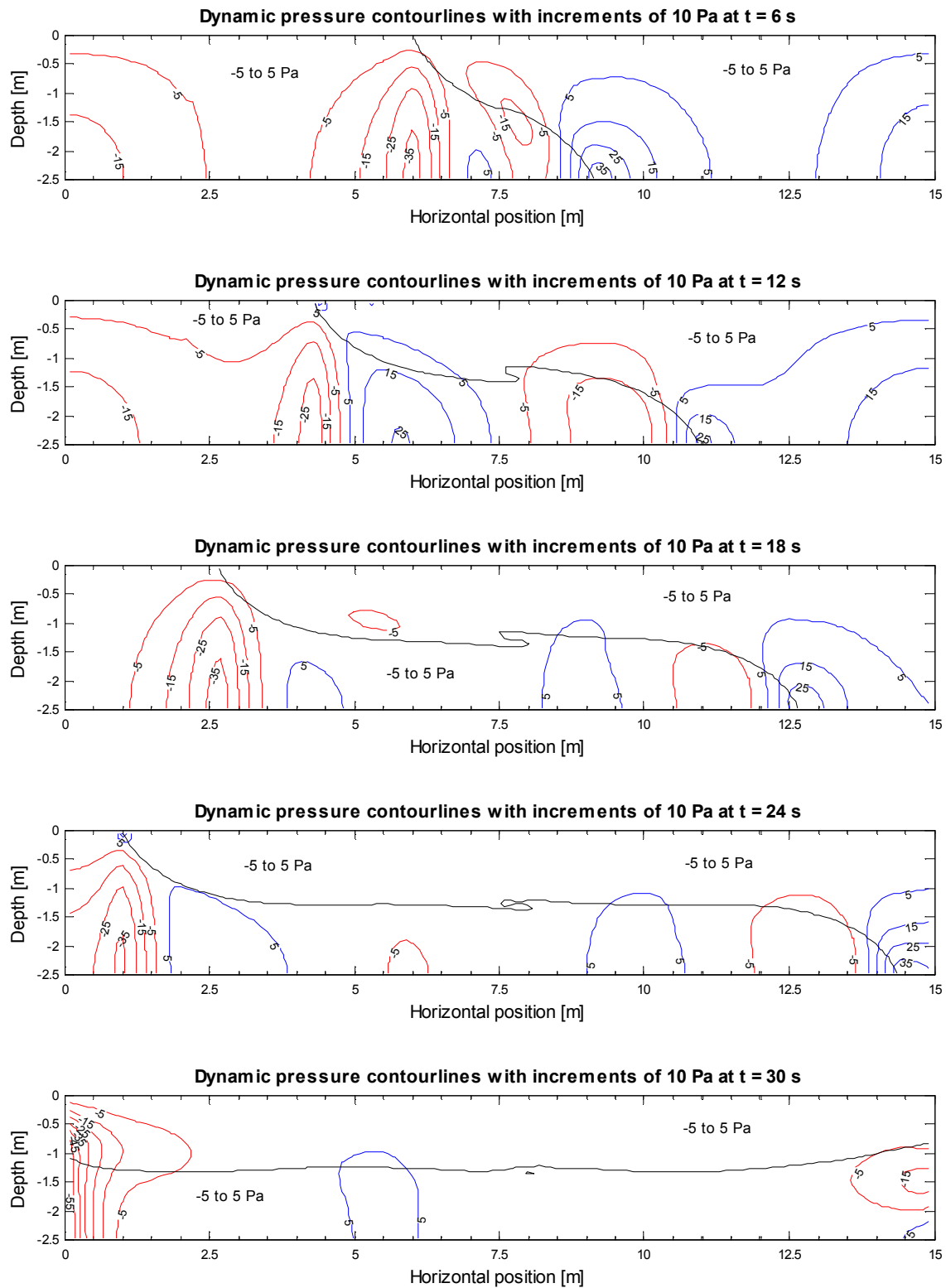


Figure 7-12: *Dynamic pressure contourlines for the Lock Exchange case at $t = 6$; 12; 18; 24; 30 s. The red lines mark areas with under-pressure; the blue lines mark areas with over-pressures. **Non-hydrostatic** simulation.*

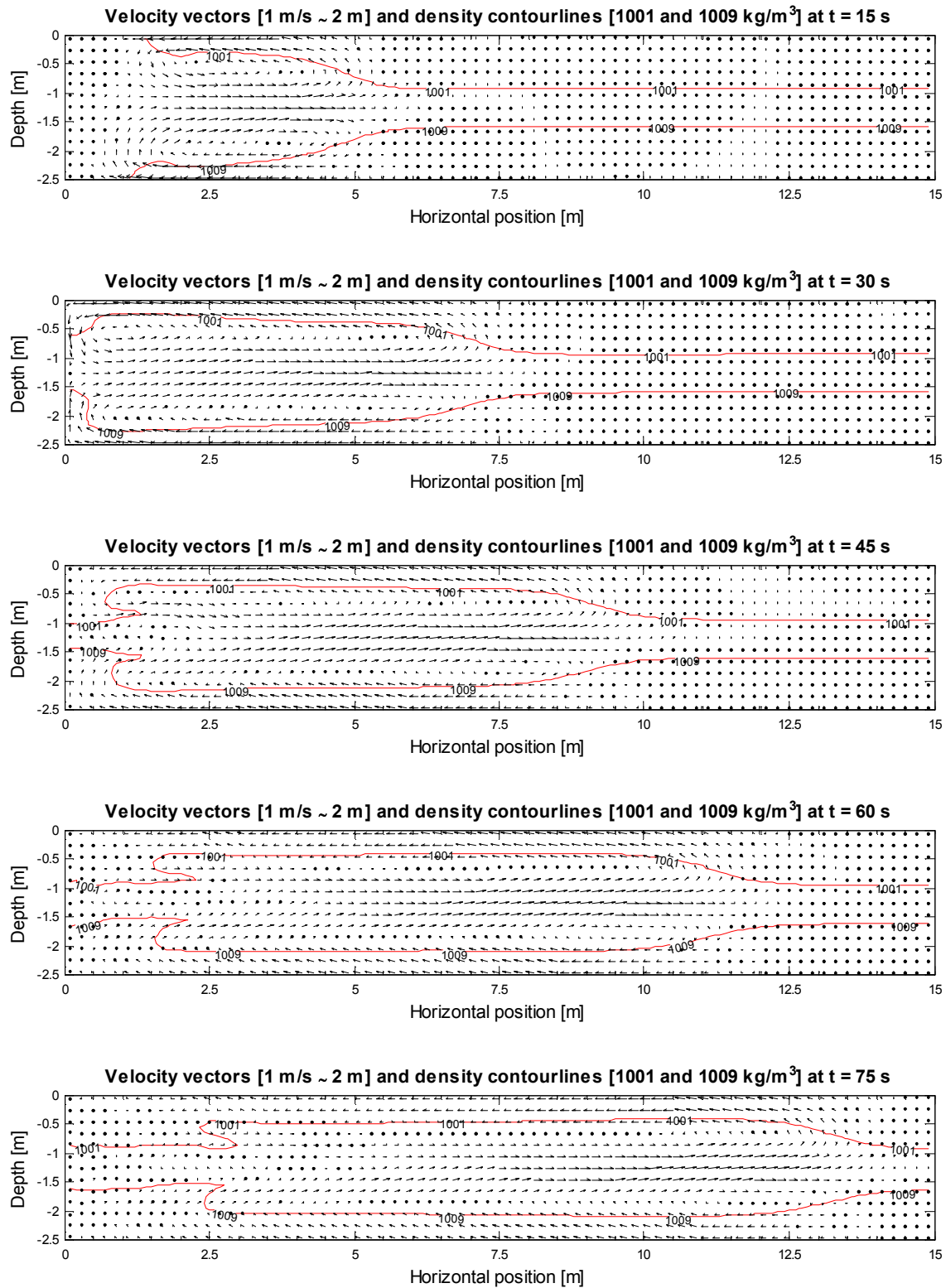


Figure 7-13: Velocity vectors and two density contourlines for the intrusive gravity current case at $t = 15; 30; 45; 60; 75$ s. **Non-hydrostatic** simulation.

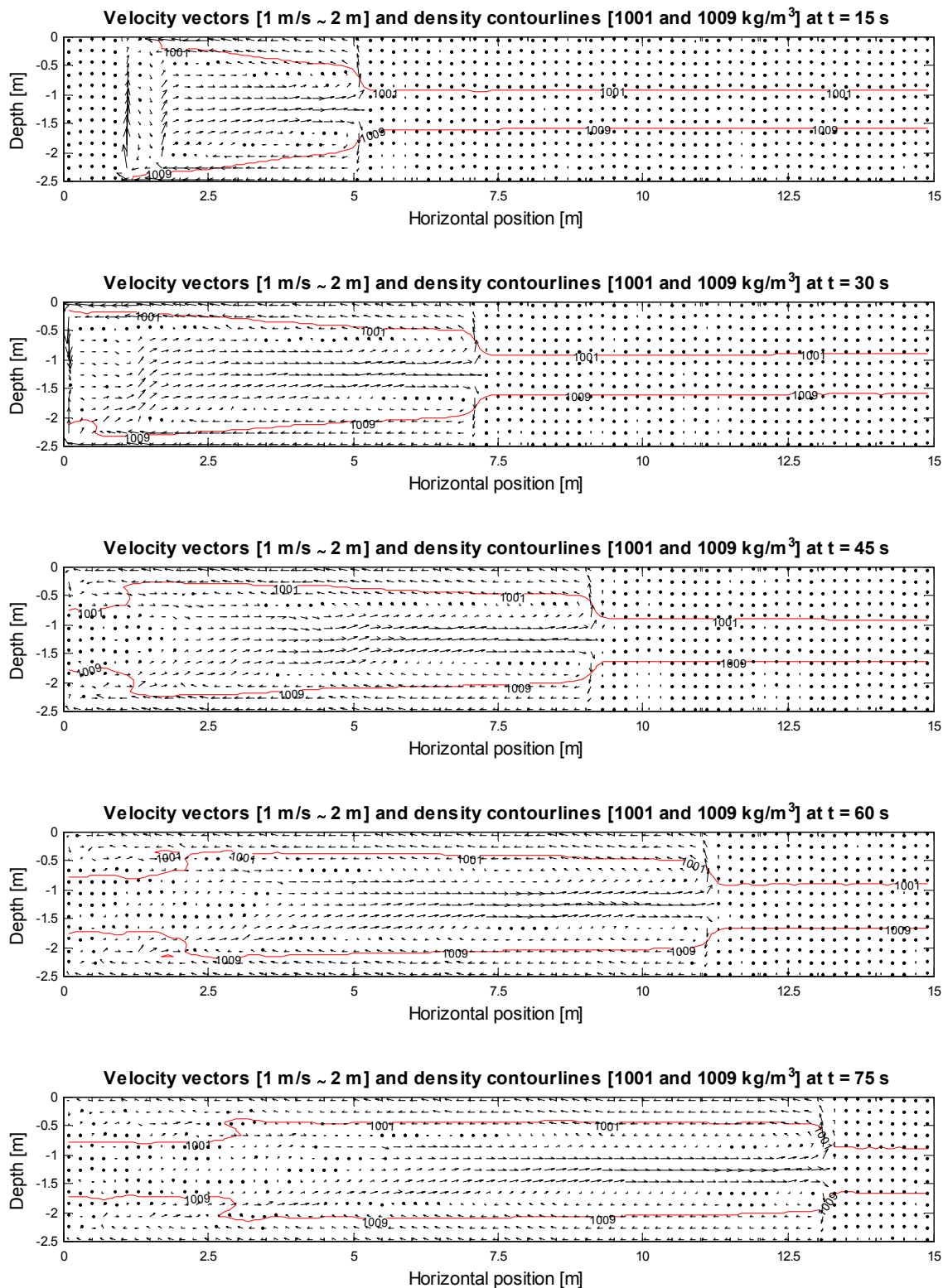


Figure 7-14: Velocity vectors and two density contourlines for the intrusive gravity current case at $t = 15; 30; 45; 60; 75$ s. **Hydrostatic** simulation.

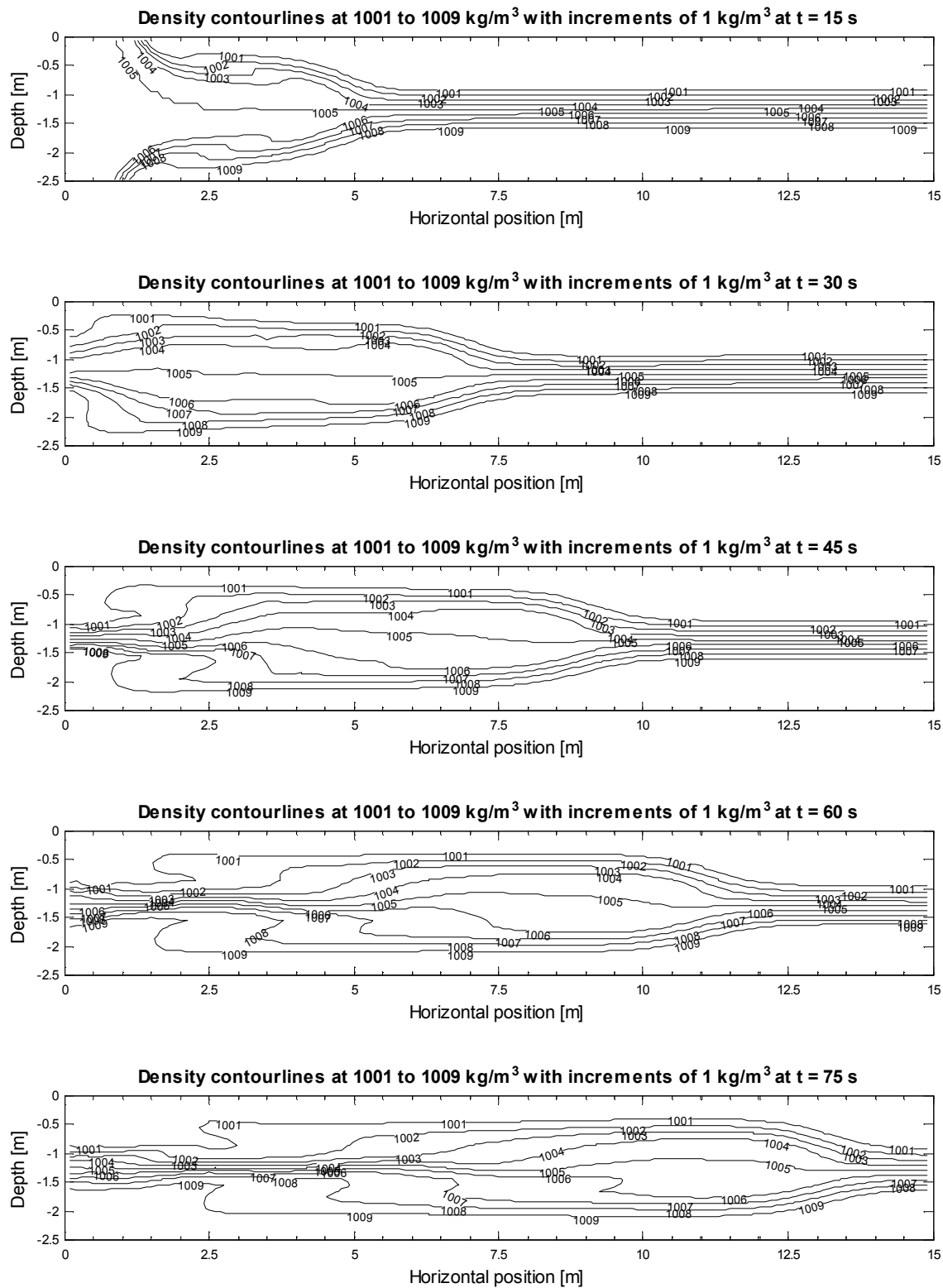


Figure 7-15: *Density contourlines for the intrusive gravity current case at $t = 15; 30; 45; 60; 75$ s. Non-hydrostatic simulation.*

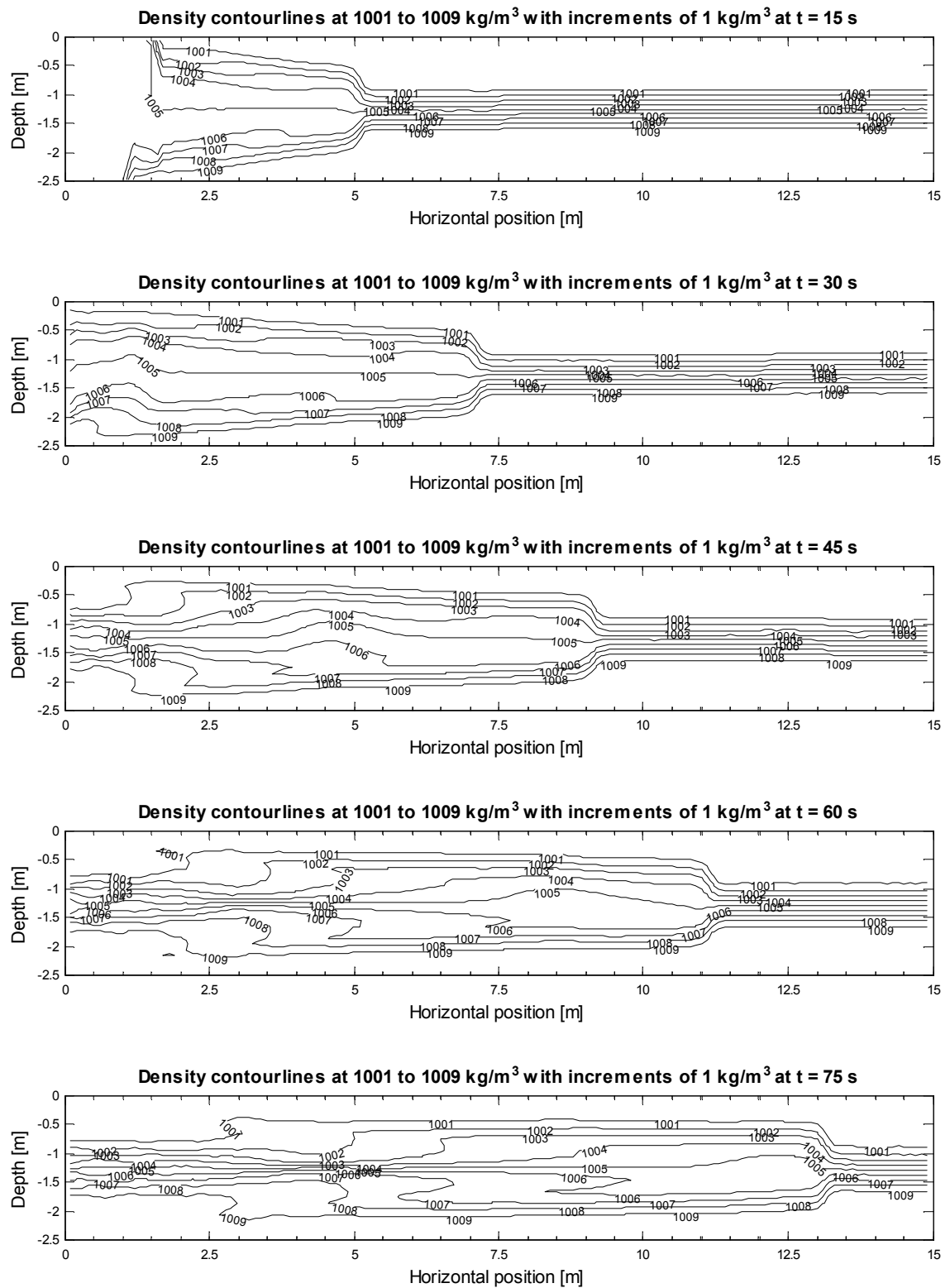


Figure 7-16: *Density contourlines for the intrusive gravity current case at t = 15; 30; 45; 60; 75 s. Hydrostatic simulation.*

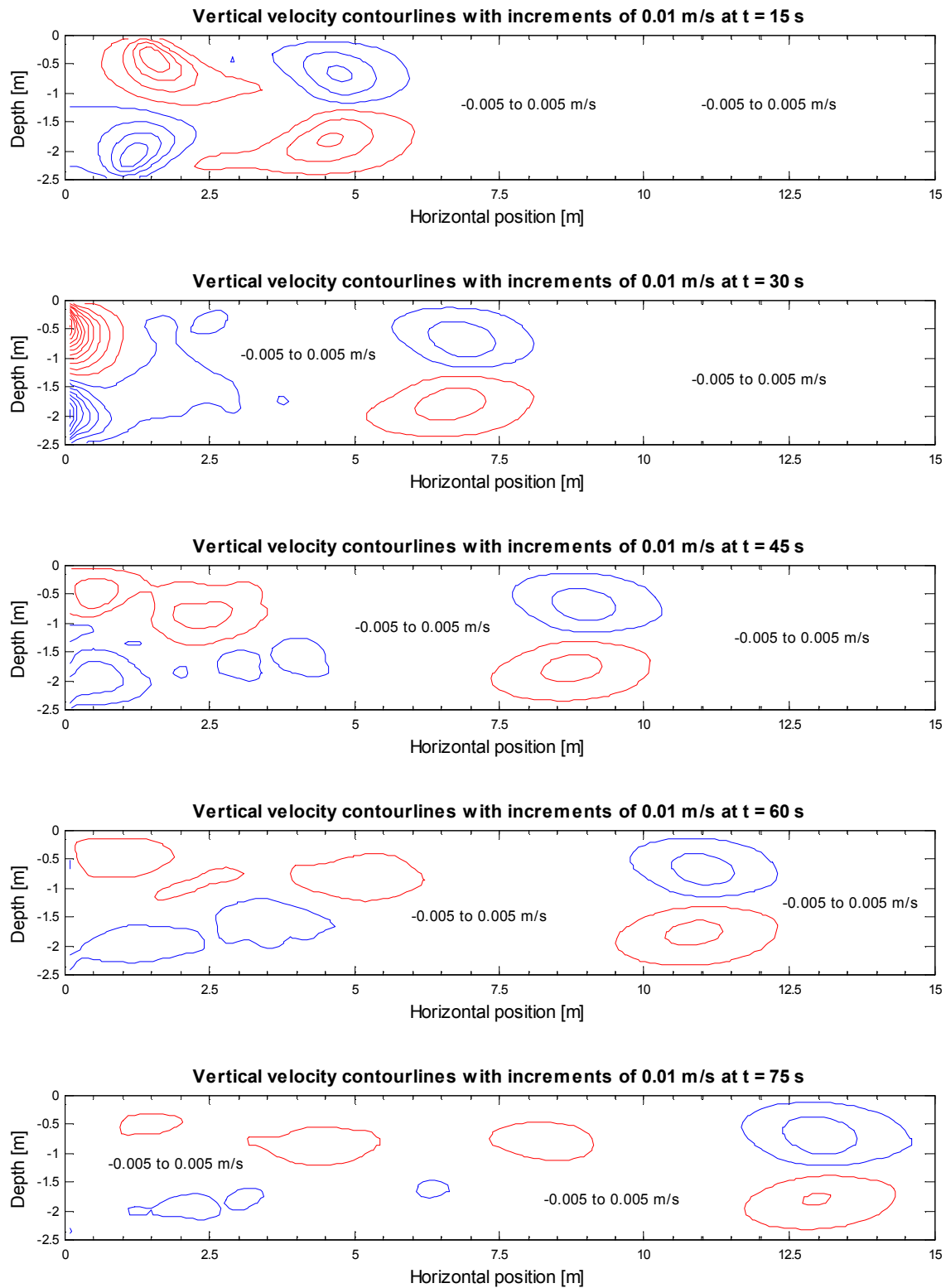


Figure 7-17: Vertical velocity contourlines for the intrusive gravity current at $t = 15$; 30; 45; 60; 75 s. The red and blue lines mark areas with respectively negative and positive velocities. **Non-hydrostatic** simulation.

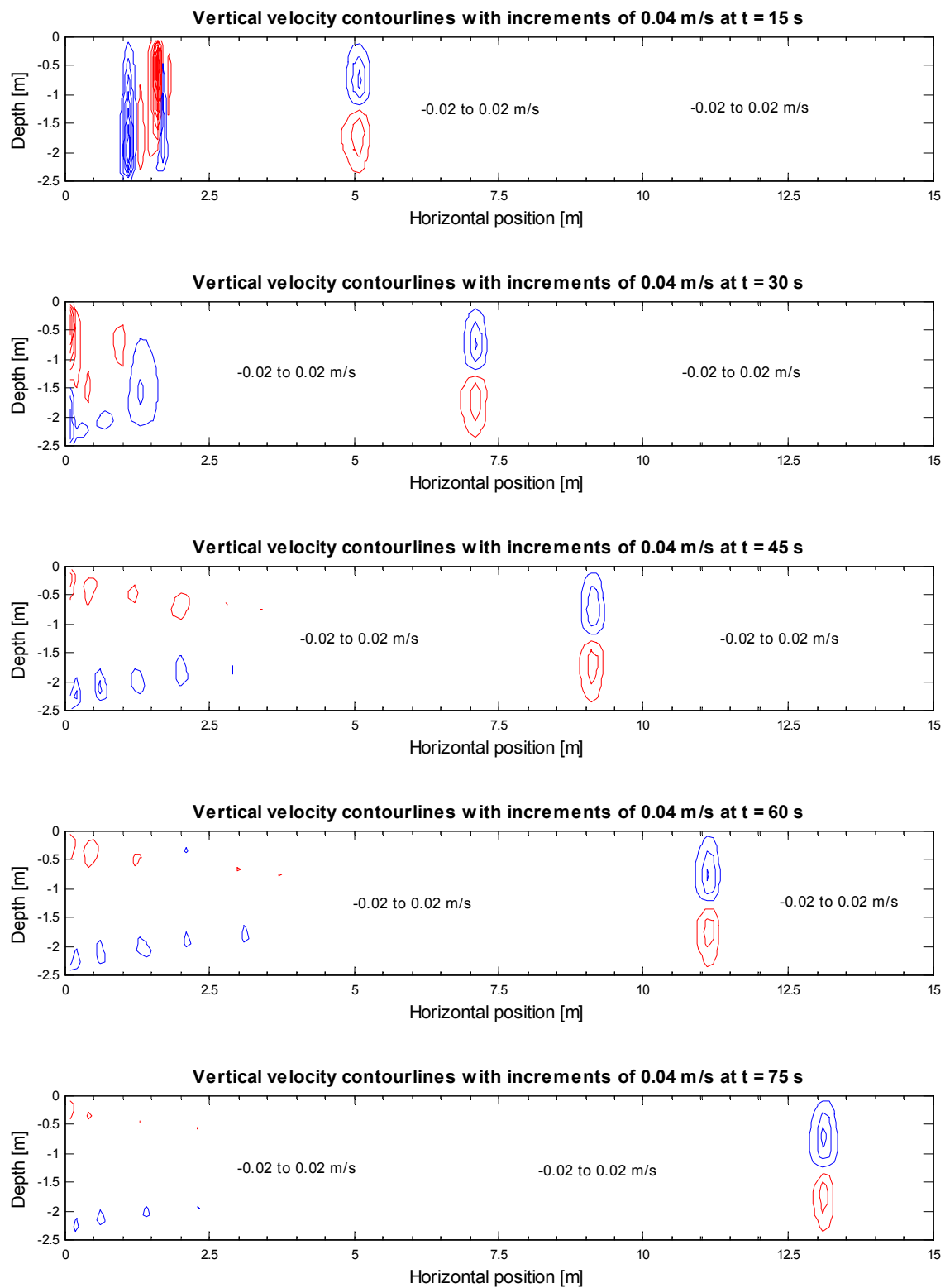


Figure 7-18: *Vertical velocity contourlines for the intrusive gravity current at $t = 15$; 30; 45; 60; 75 s. The red and blue lines mark areas with respectively negative and positive velocities. **Hydrostatic** simulation.*

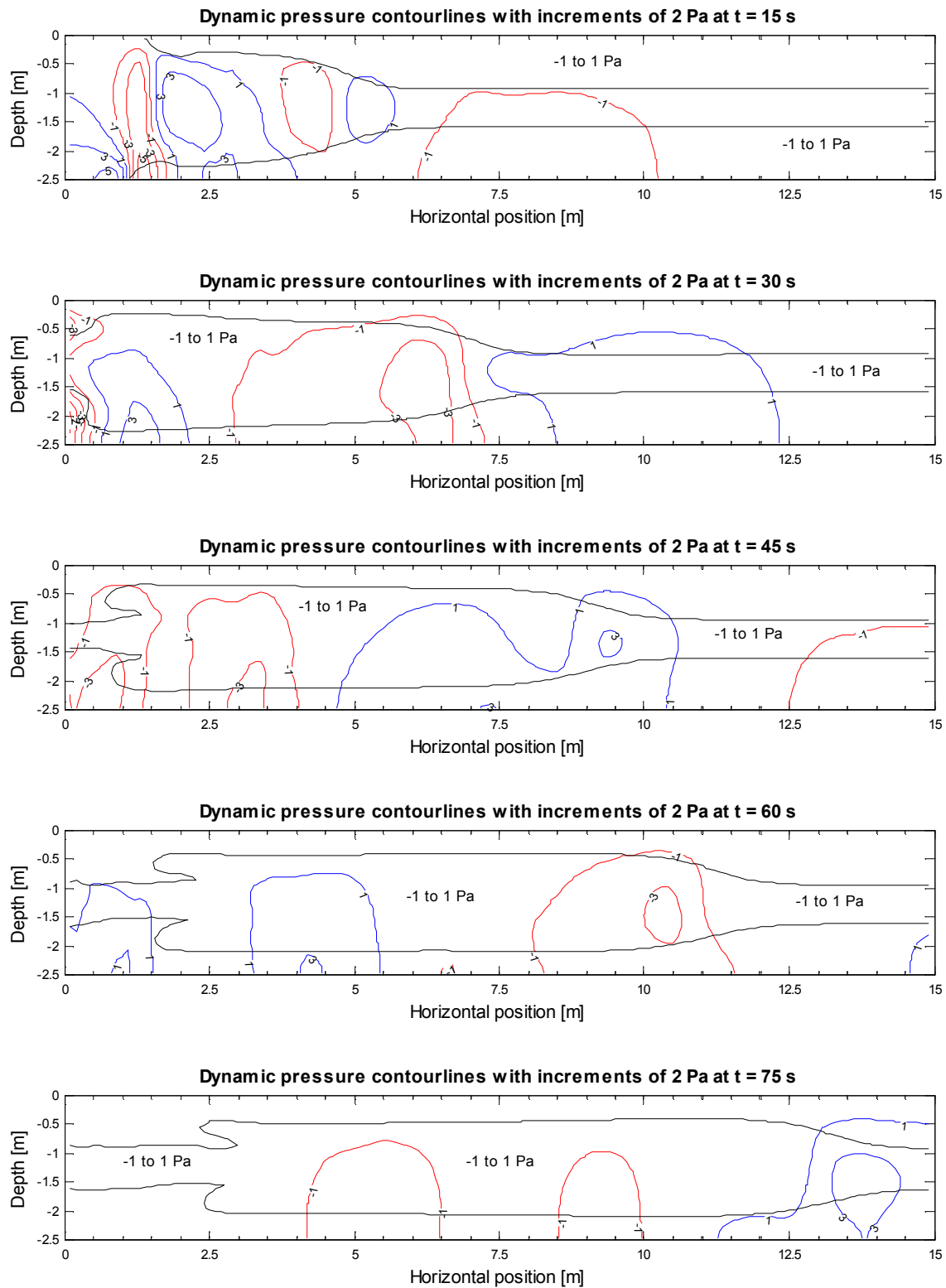


Figure 7-19: *Dynamic pressure contourlines for the intrusive gravity current case current at $t = 15$; 30; 45; 60; 75 s. The red and blue lines mark areas with respectively under- and over-pressure. **Non-hydrostatic** simulation.*

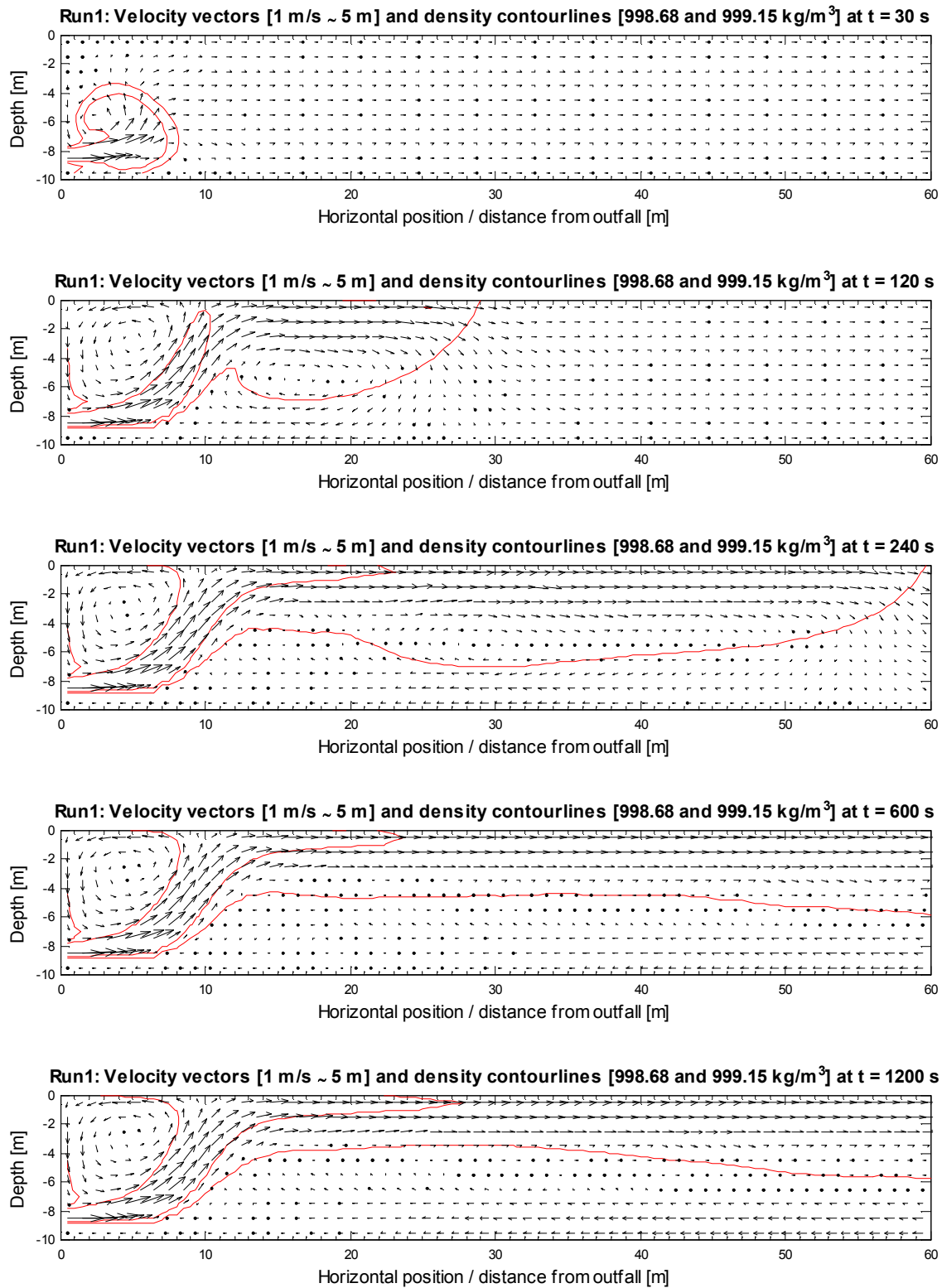


Figure 7-20: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 1** at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

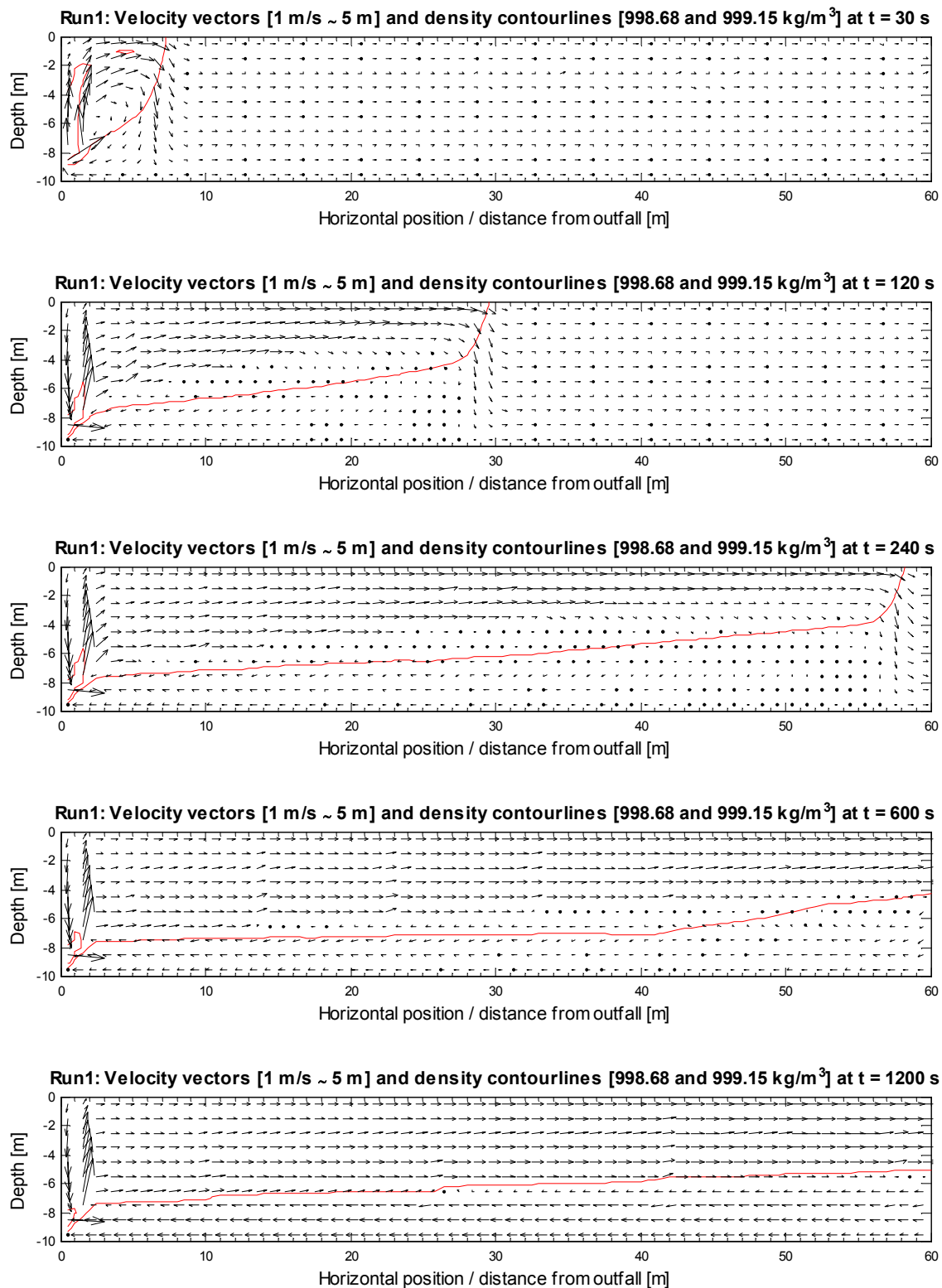


Figure 7-21: *Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for run 1 at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.*

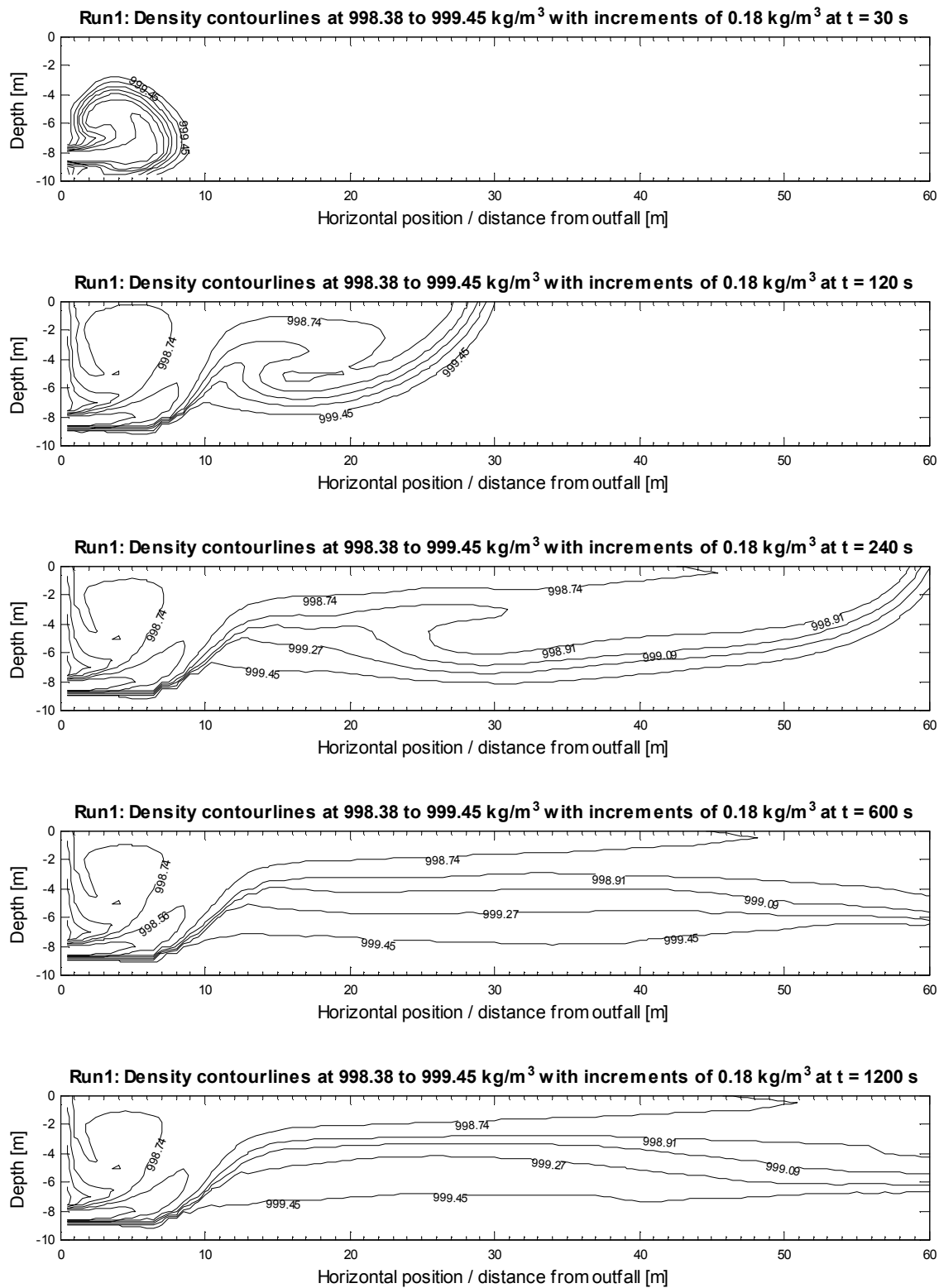


Figure 7-22: *Density contourlines for run 1 at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; Non-hydrostatic simulation.*

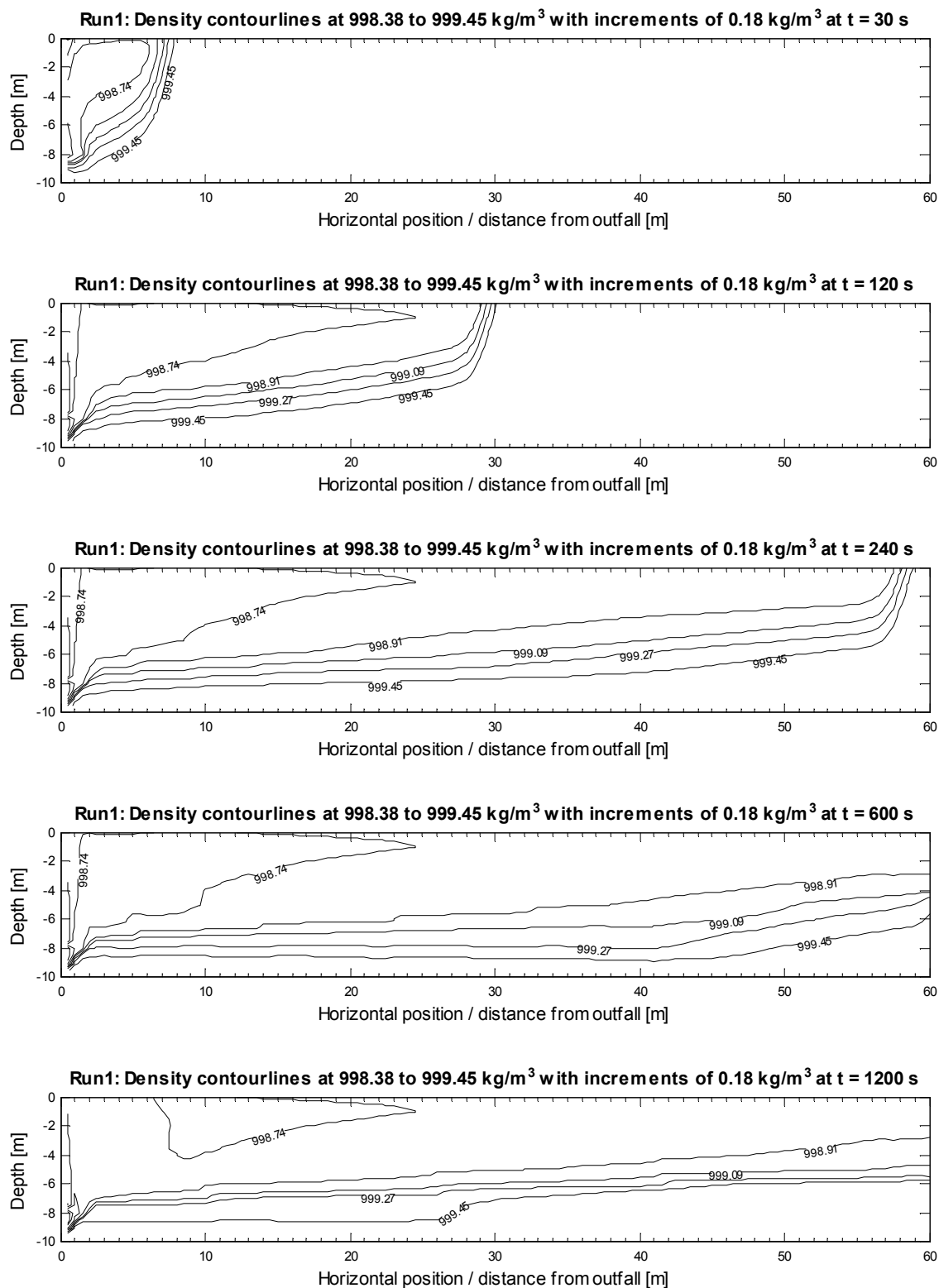


Figure 7-23: *Density contourlines for run 1 at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.*

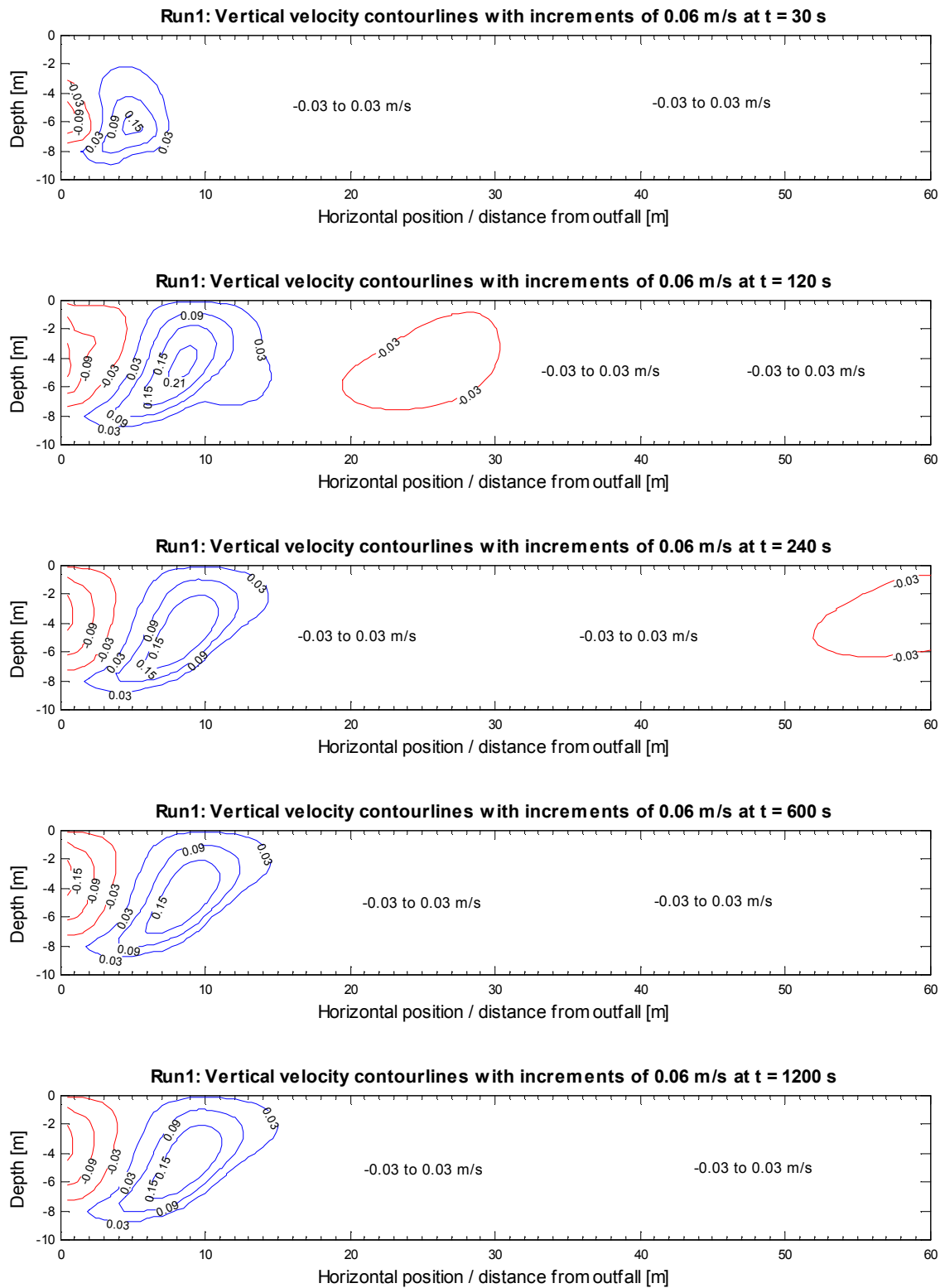


Figure 7-24: Vertical velocity contourlines for *run 1* at $t = 30; 120; 240; 600; 1200$ s. The red and blue lines mark areas with respectively negative and positive velocities. 2DV model with press. correction; **Non-hydrostatic** simulation.

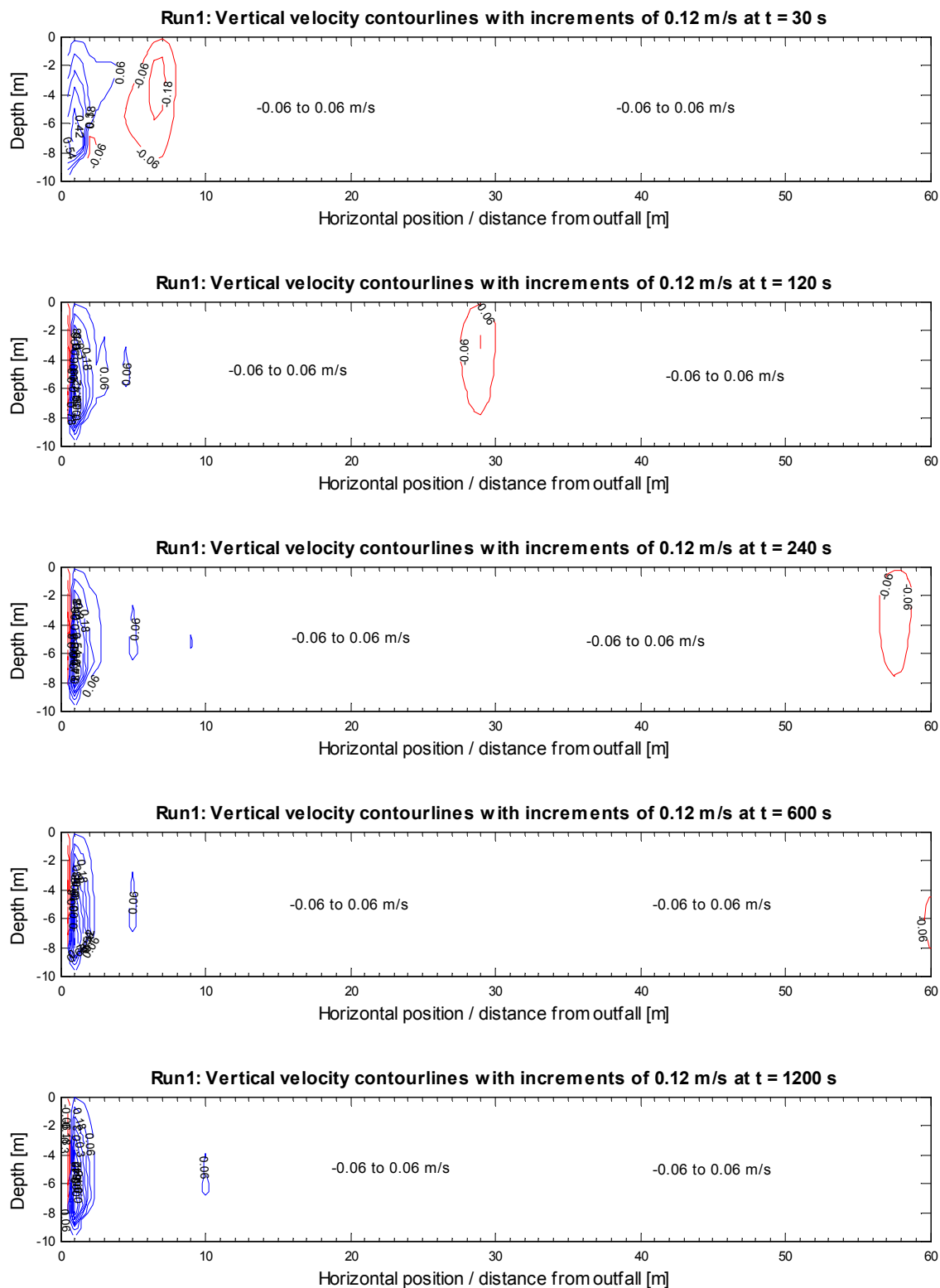


Figure 7-25: Vertical velocity contourlines for **run 1** at $t = 30; 120; 240; 600; 1200$ s. The red and blue lines mark areas with respectively negative and positive velocities. 2DV model with pressure correction; **Hydrostatic** simulation.

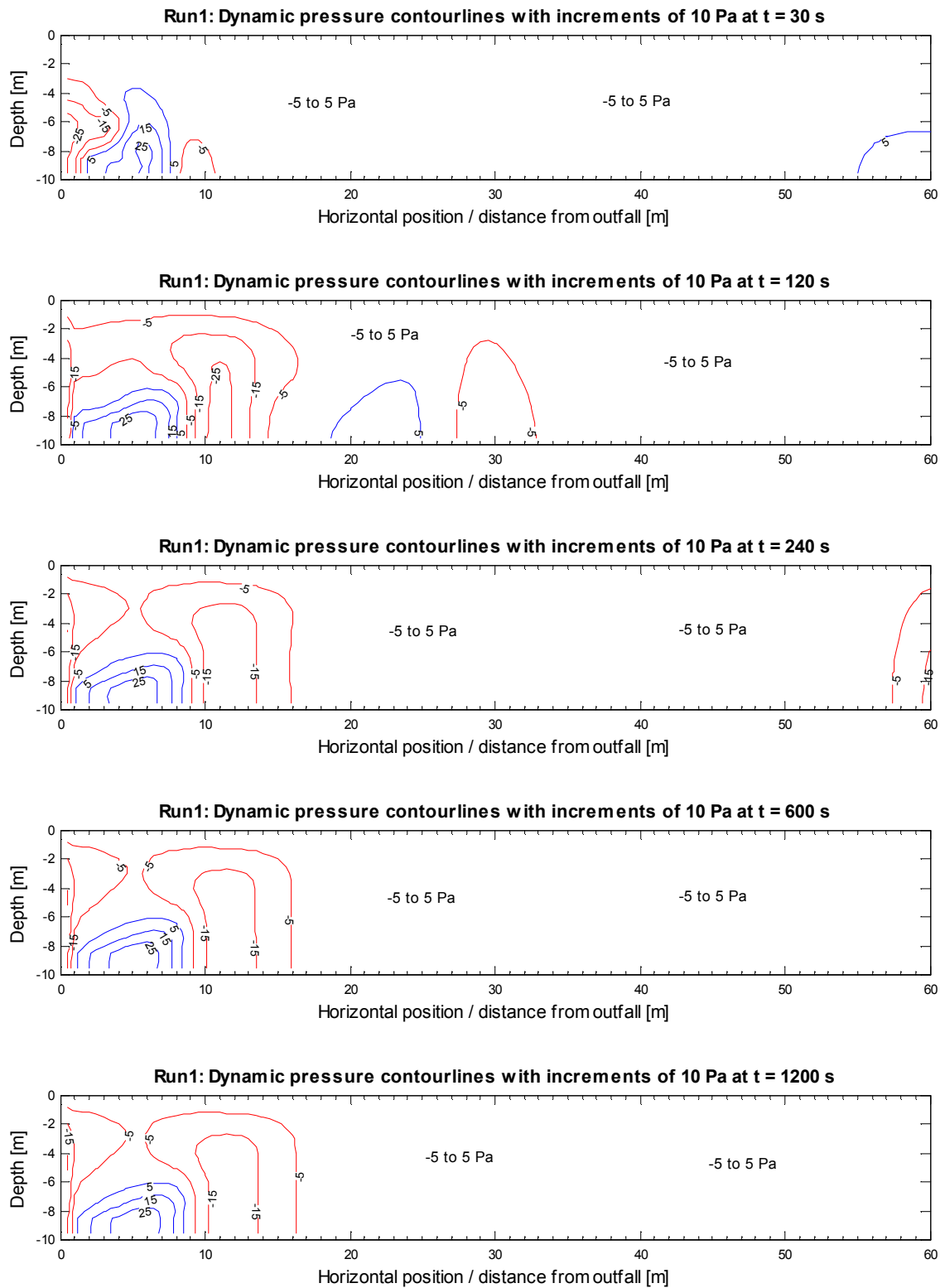


Figure 7-26: *Dynamic pressure contourlines for **run 1** at t = 30; 120; 240; 600; 1200 s. The red and blue lines mark areas with respectively under- and over-pressure; 2DV model with press. correction; **Non-hydrostatic** simulation.*

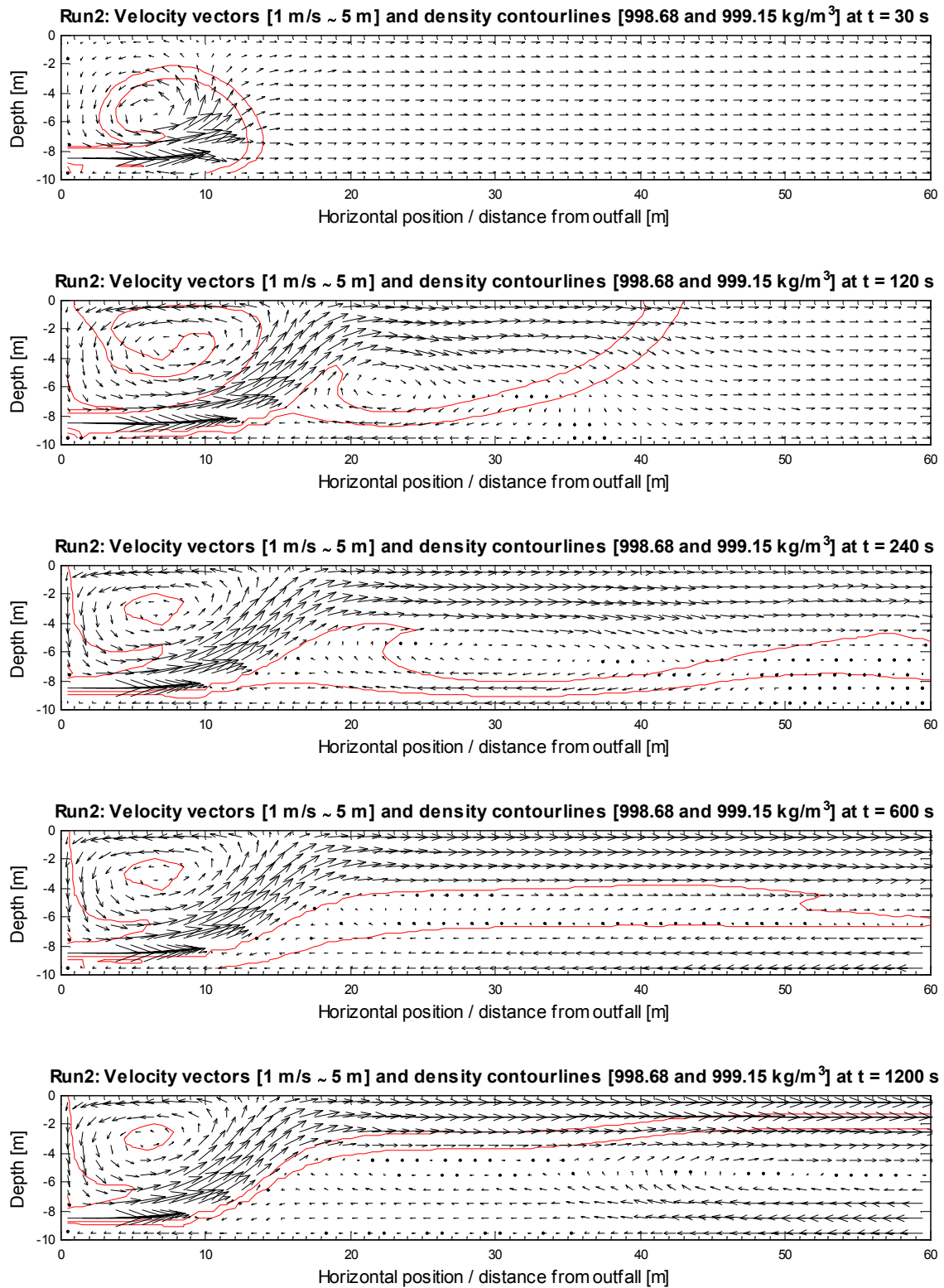


Figure 7-27: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 2** (increased jet velocity) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

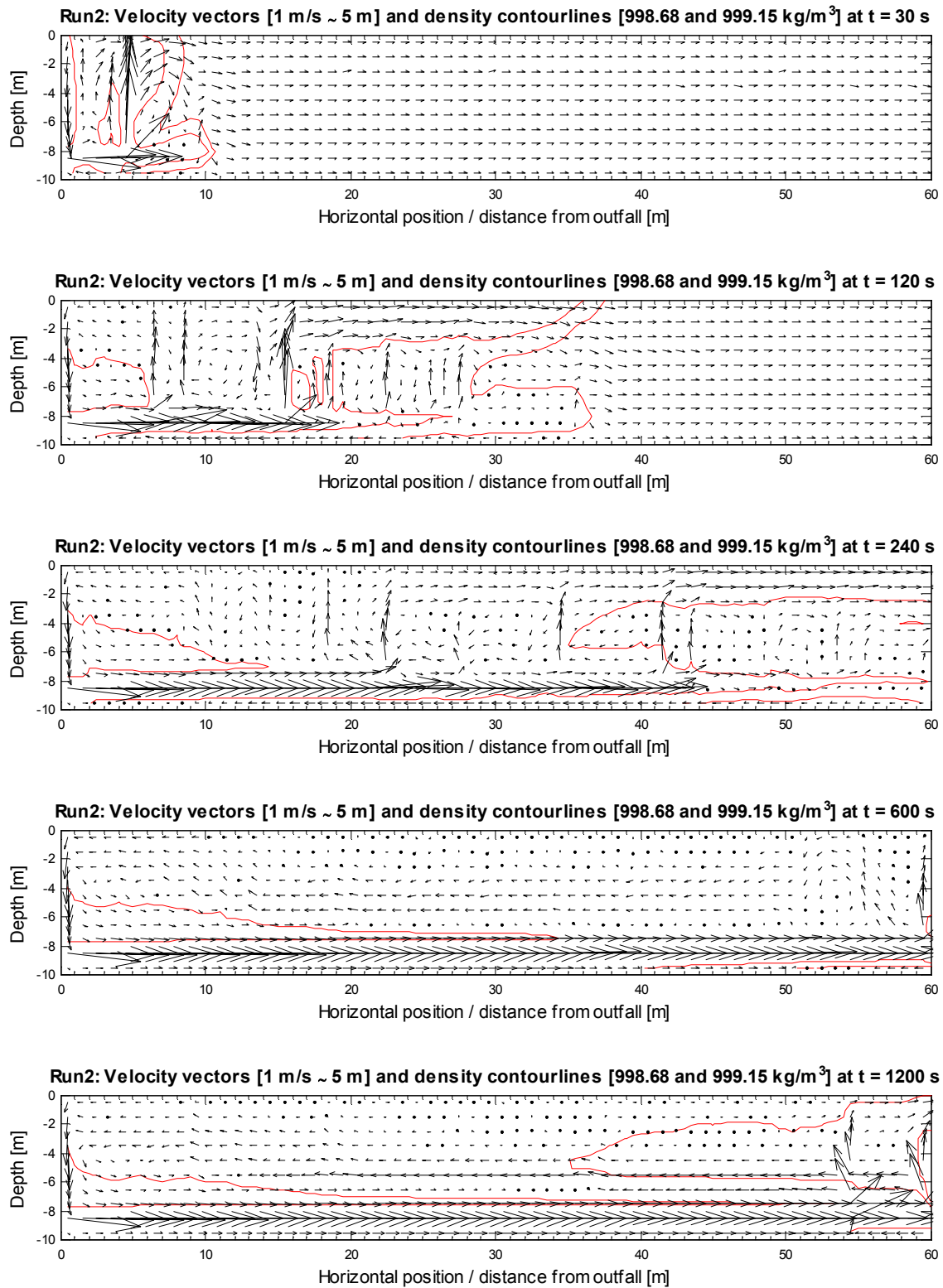


Figure 7-28: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 2** (increased jet velocity) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

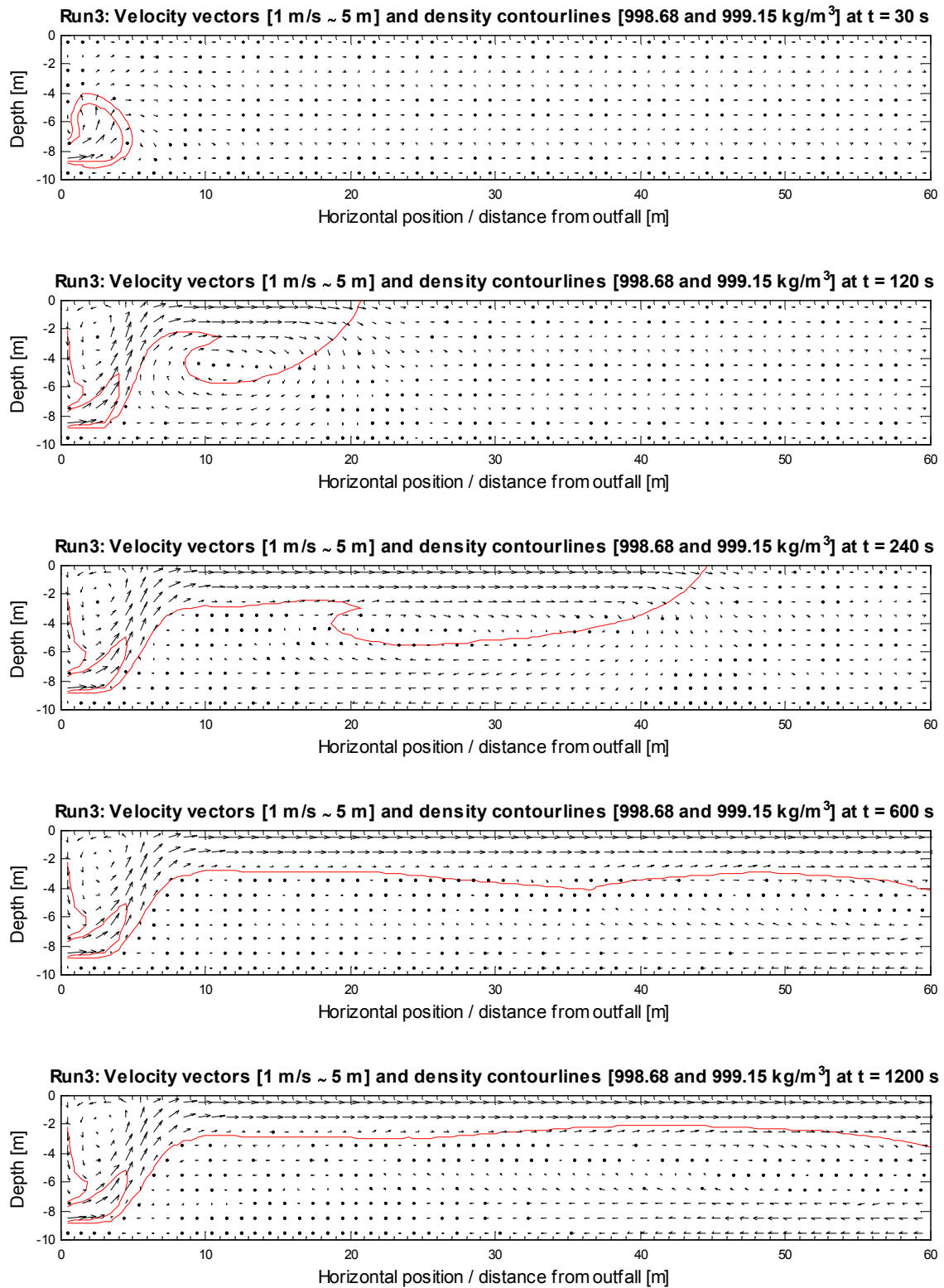


Figure 7-29: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 3** (decreased jet velocity) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

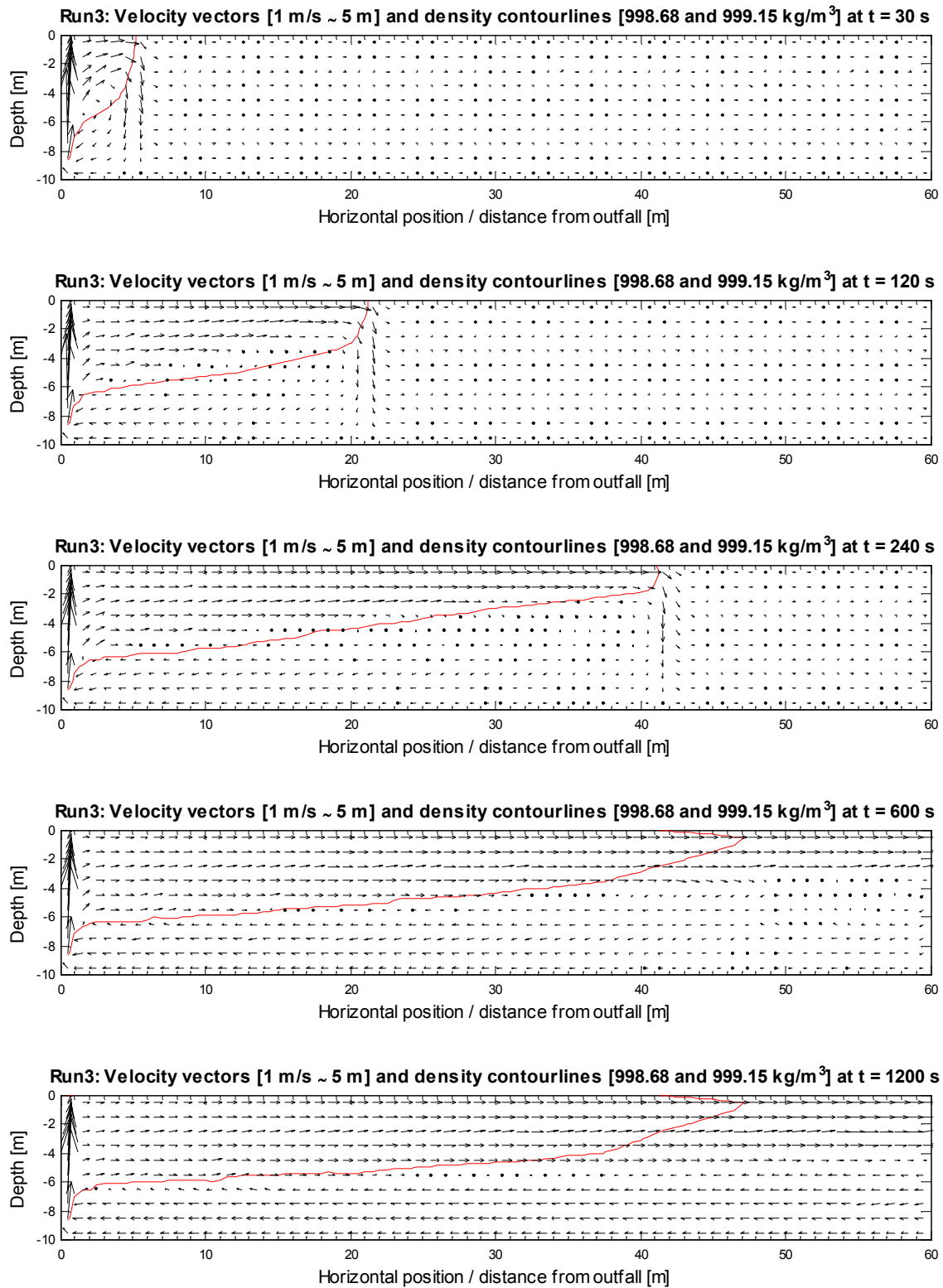


Figure 7-30: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 3** (decreased jet velocity) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

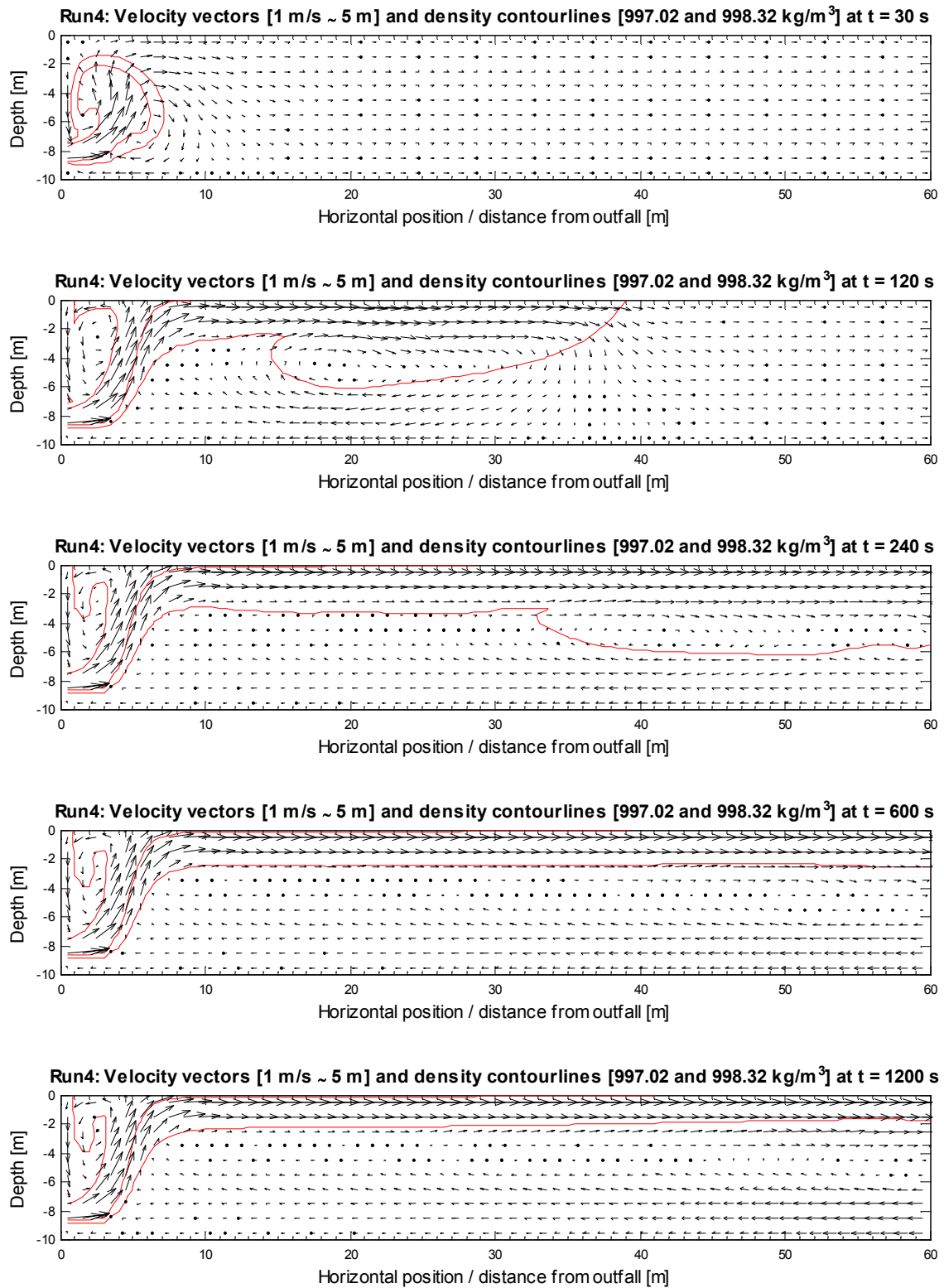


Figure 7-31: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 4** (increased density diff.) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

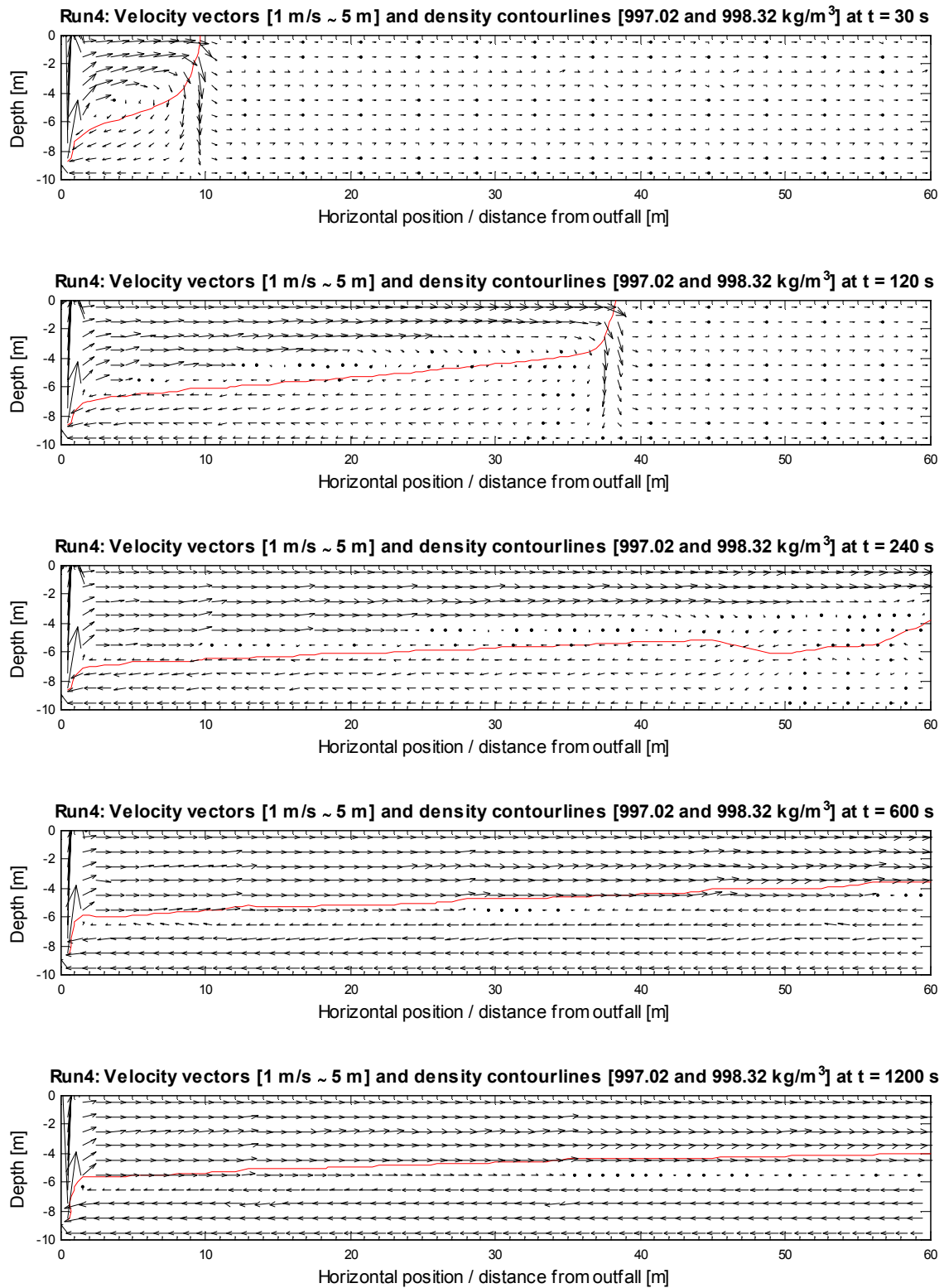


Figure 7-32: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 4** (increased density diff.) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

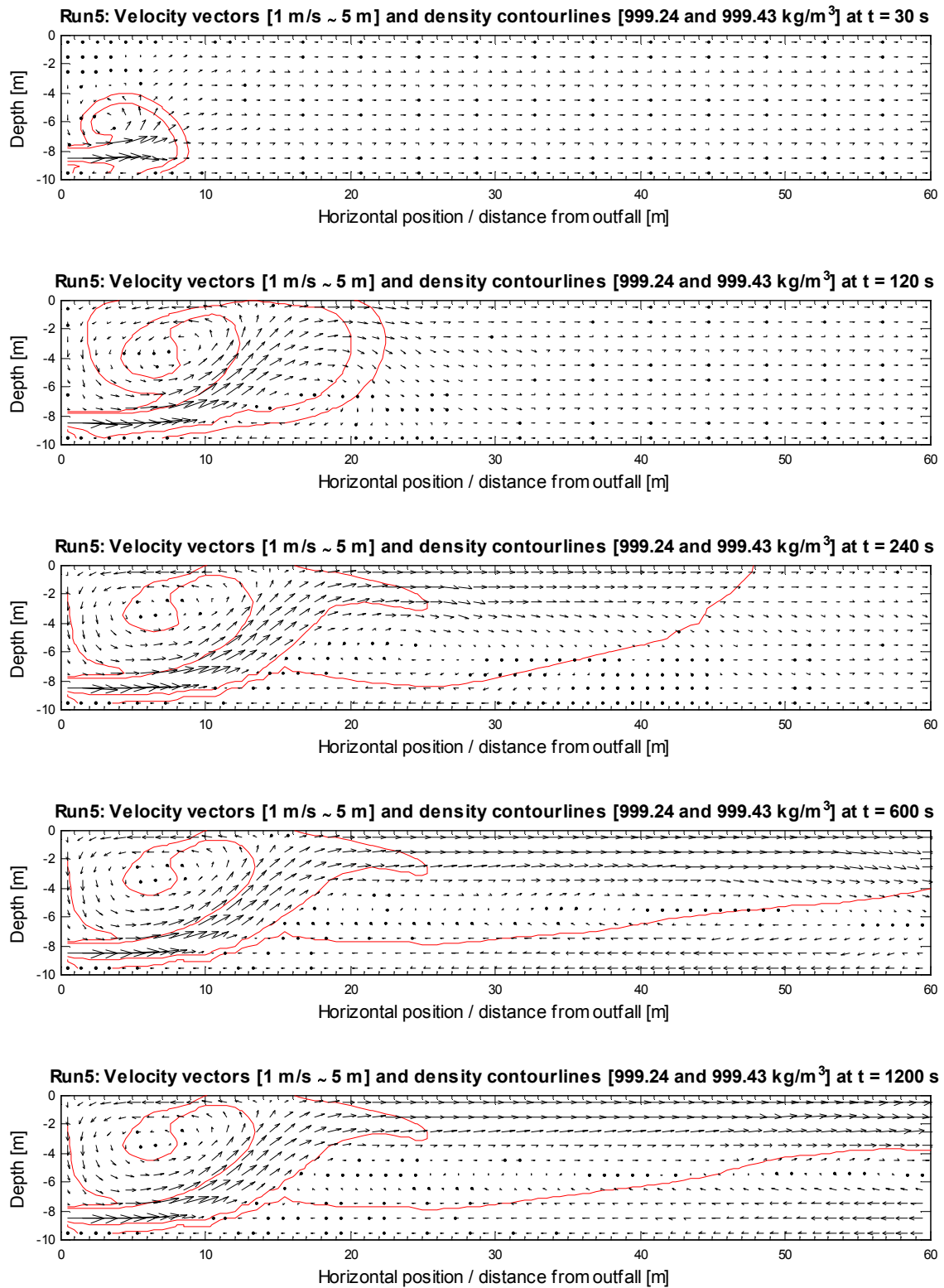


Figure 7-33: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 5** (decreased density diff.) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

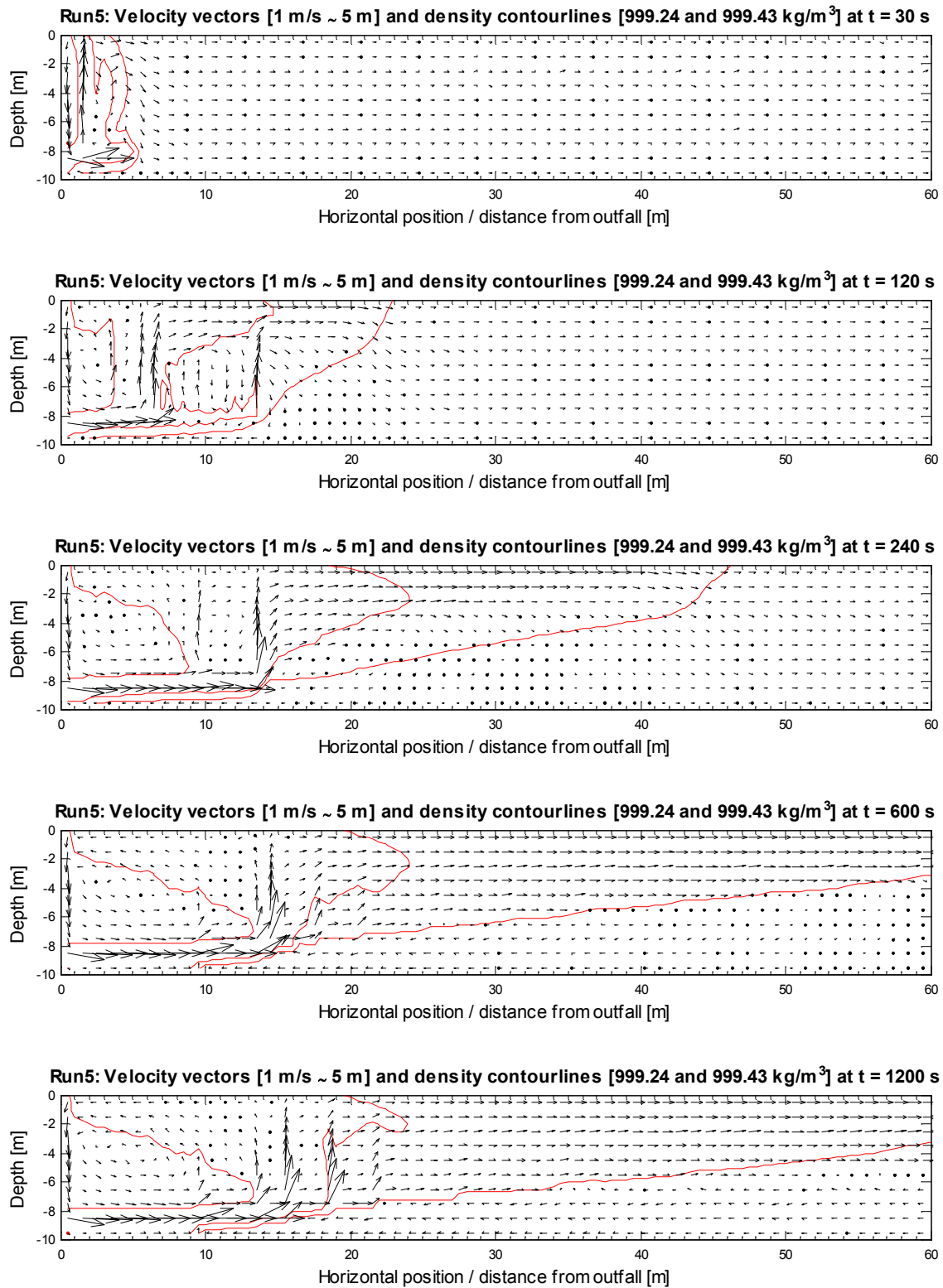


Figure 7-34: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 5** (decreased density diff.) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

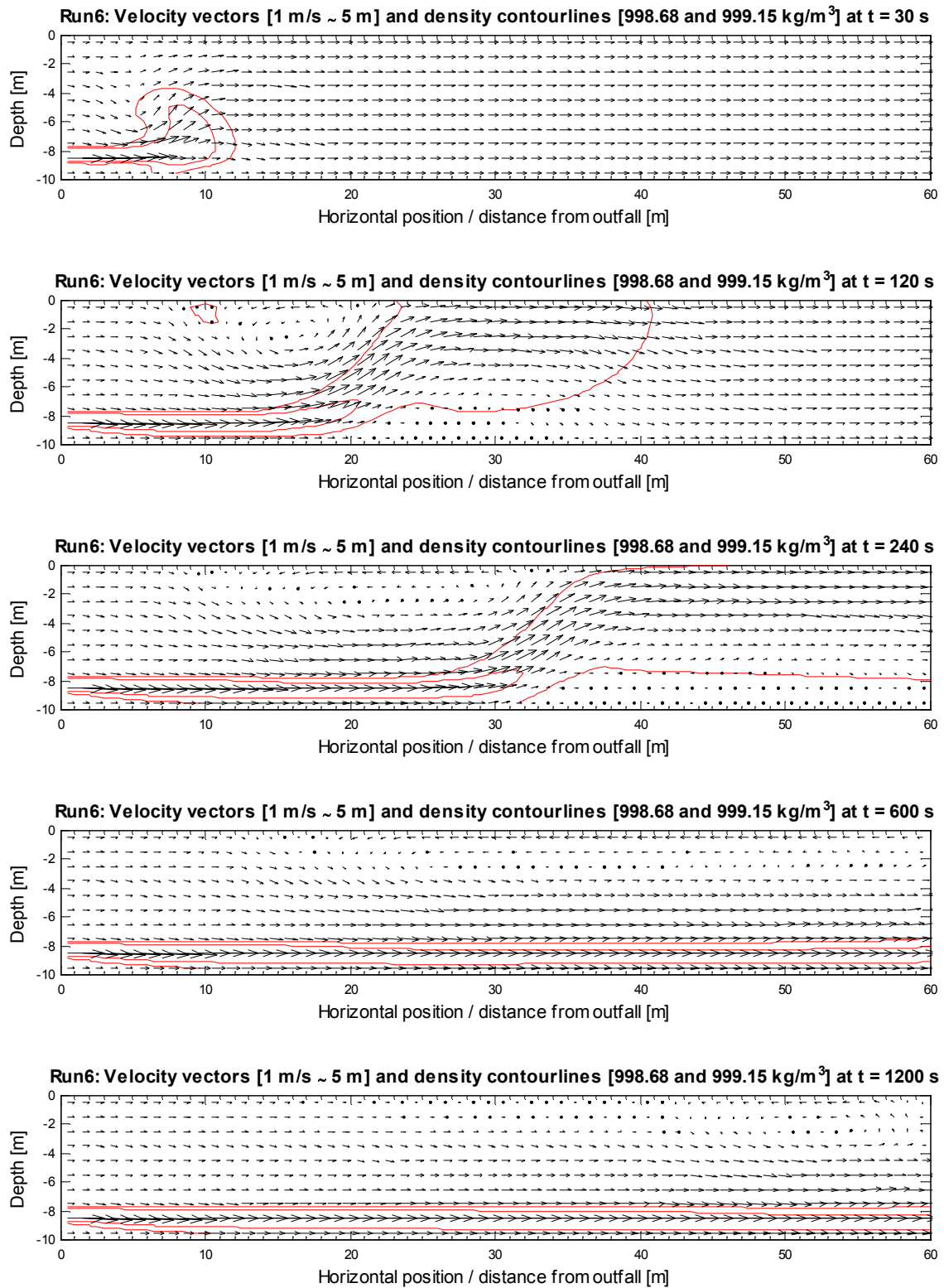


Figure 7-35: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 6** (increased ambient velocity) at $t = 30; 120; 240; 600; 1200$ s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

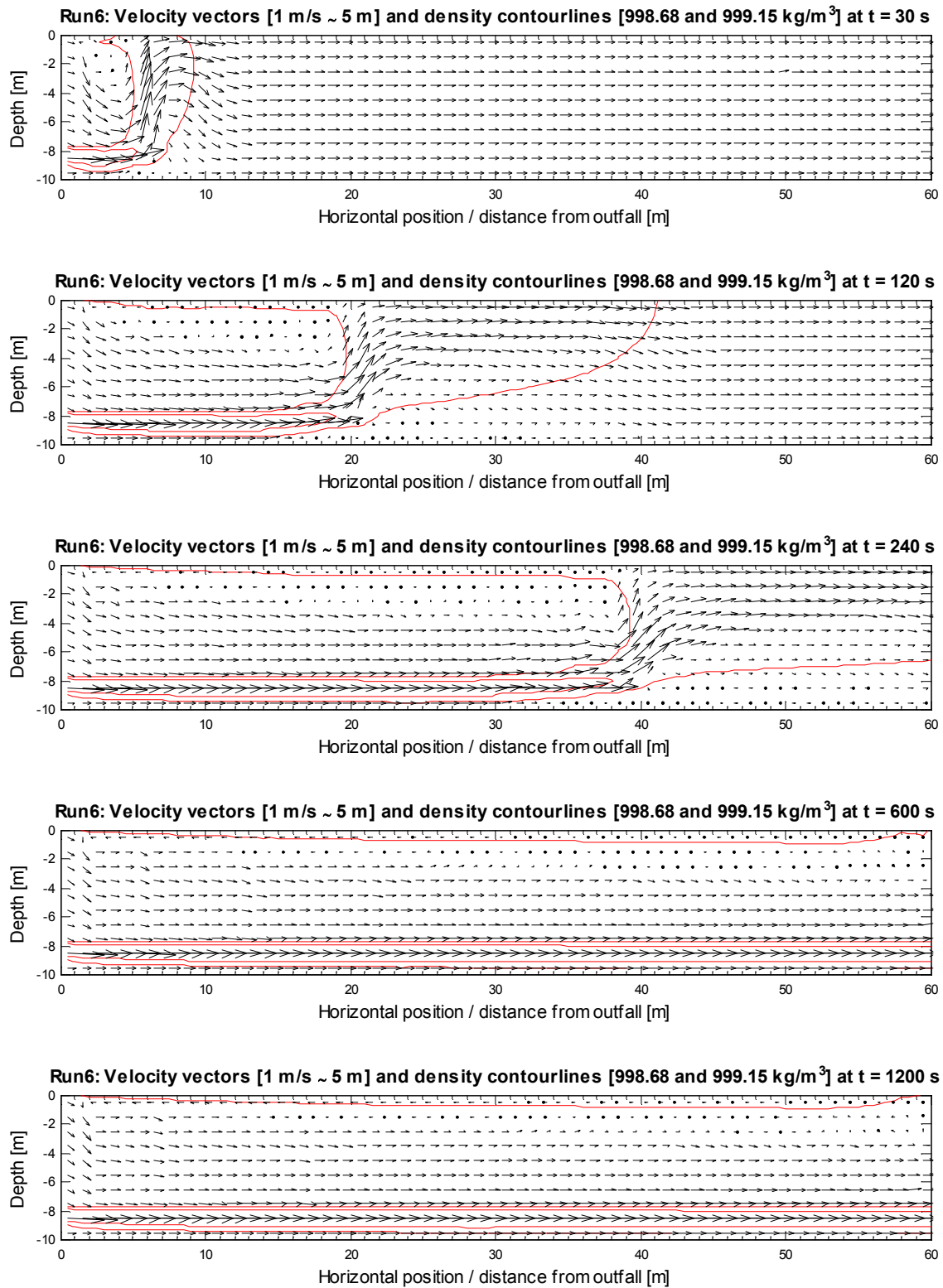


Figure 7-36: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 6** (increased ambient velocity) at $t = 30; 120; 240; 600; 1200$ s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

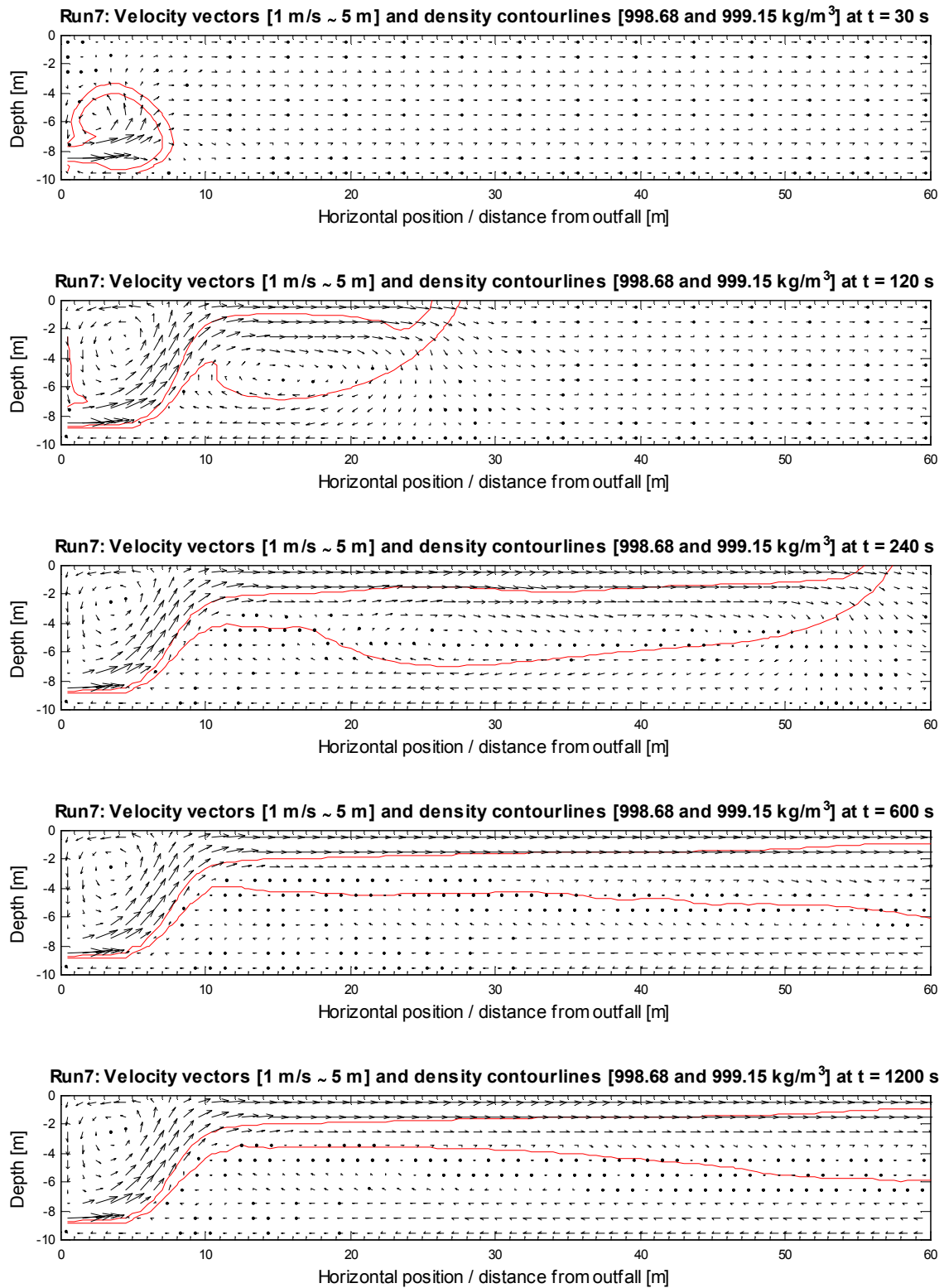


Figure 7-37: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 7** (zero ambient velocity) at $t = 30; 120; 240; 600; 1200$ s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

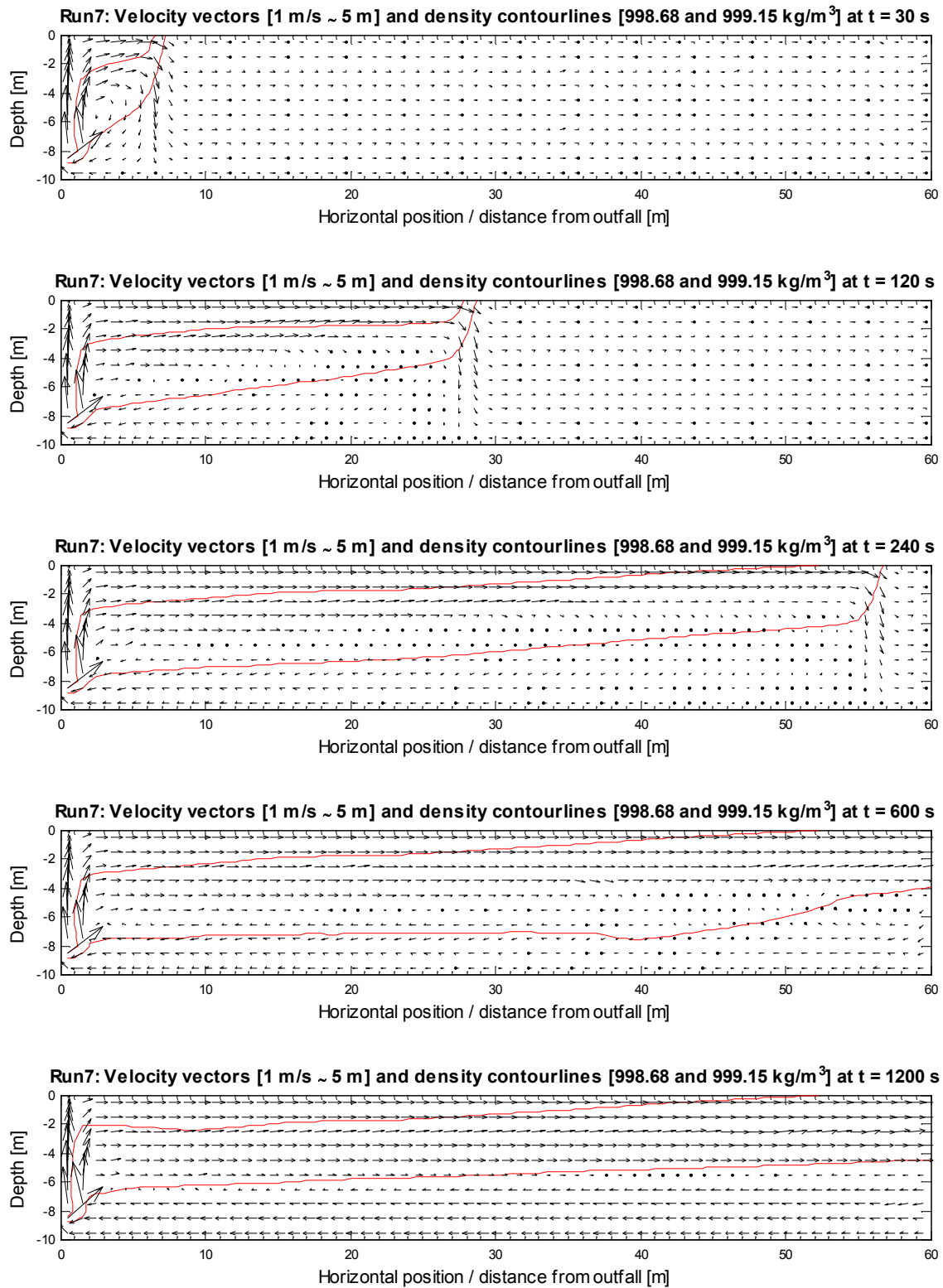


Figure 7-38: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 7** (zero ambient velocity) at $t = 30; 120; 240; 600; 1200$ s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

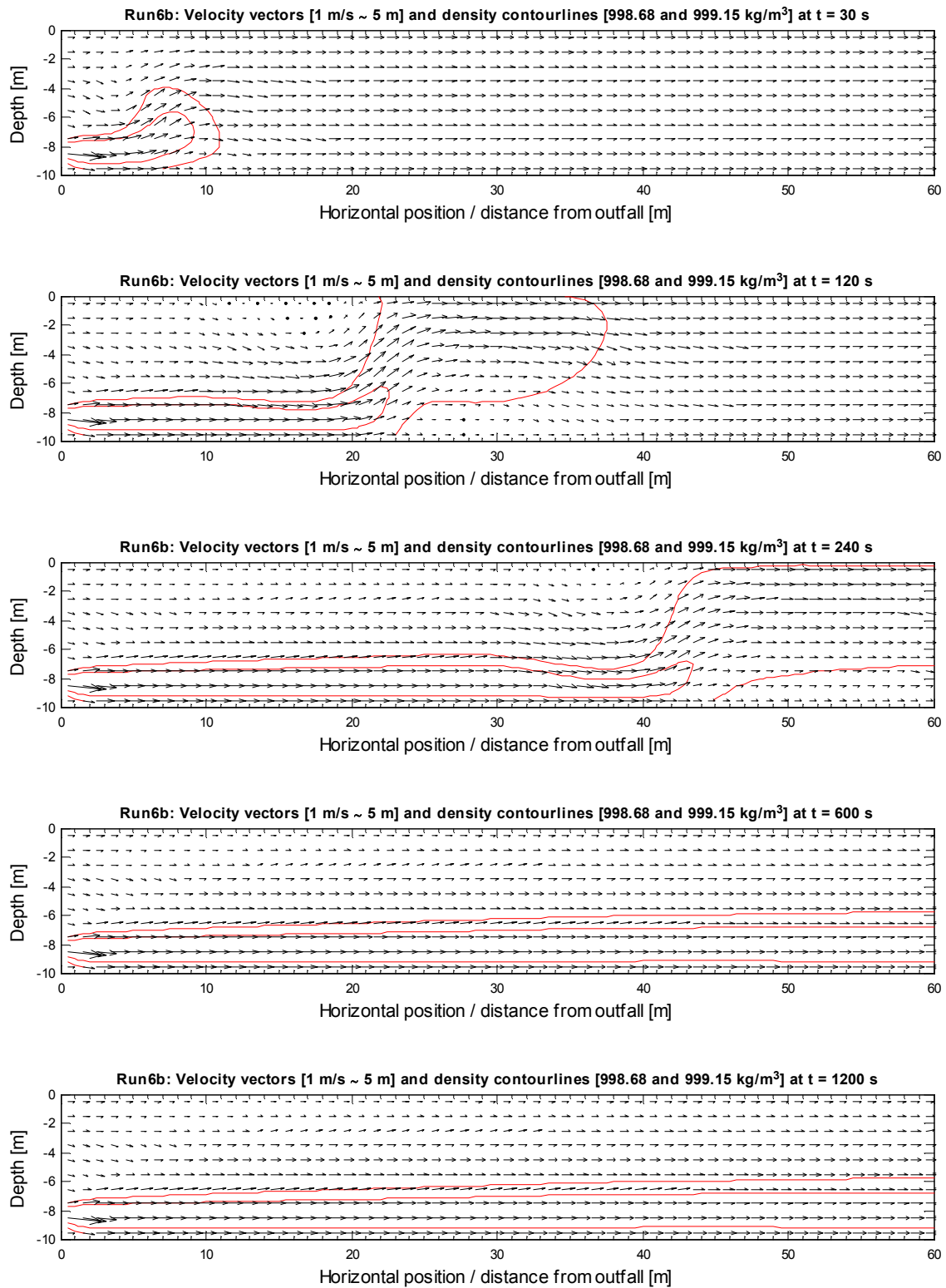


Figure 7-39: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 6b** (increased viscosity) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Non-hydrostatic** simulation.

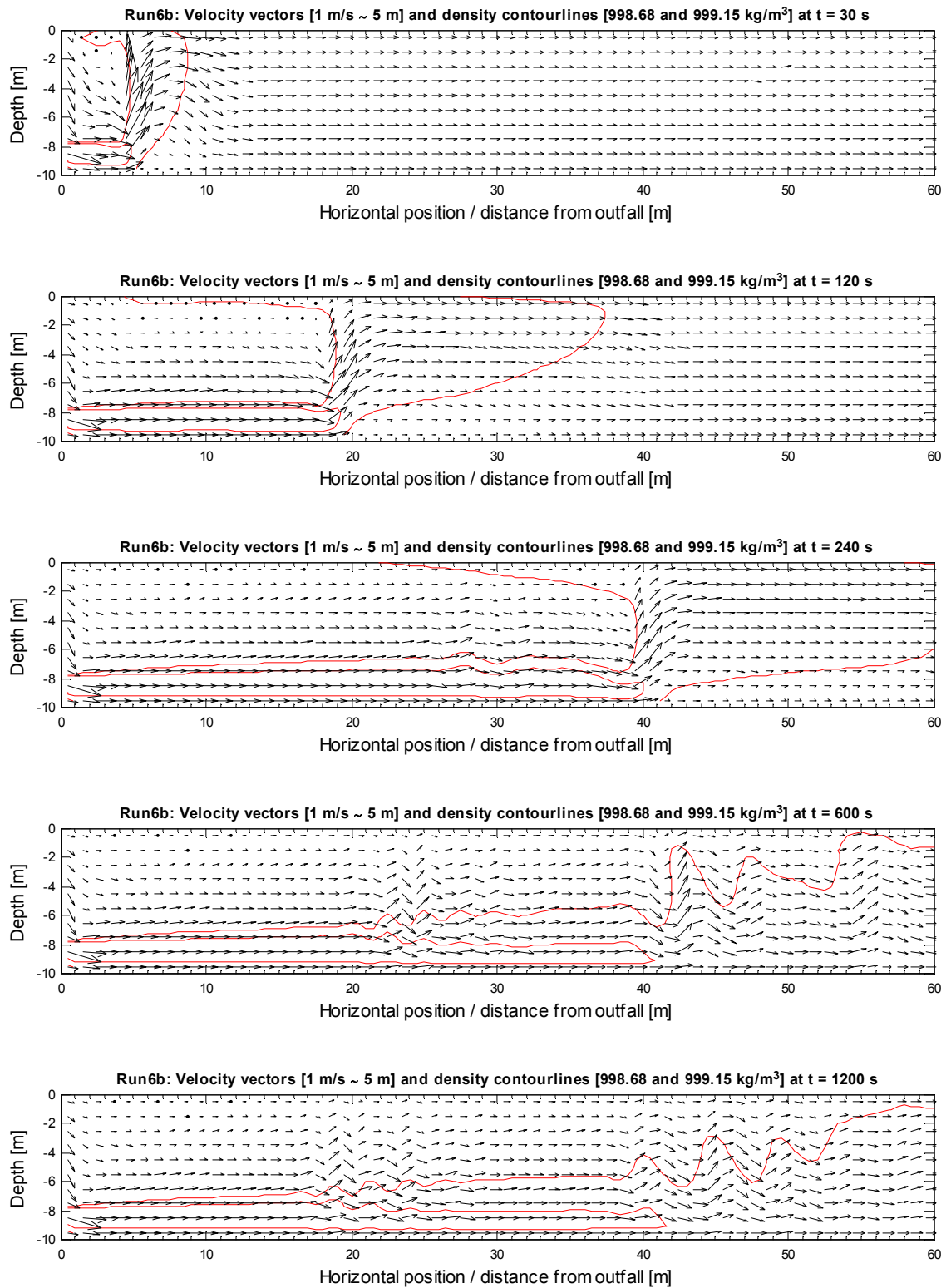


Figure 7-40: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 6b** (increased viscosity) at t = 30; 120; 240; 600; 1200 s. 2DV fixed layer model with pressure correction; **Hydrostatic** simulation.

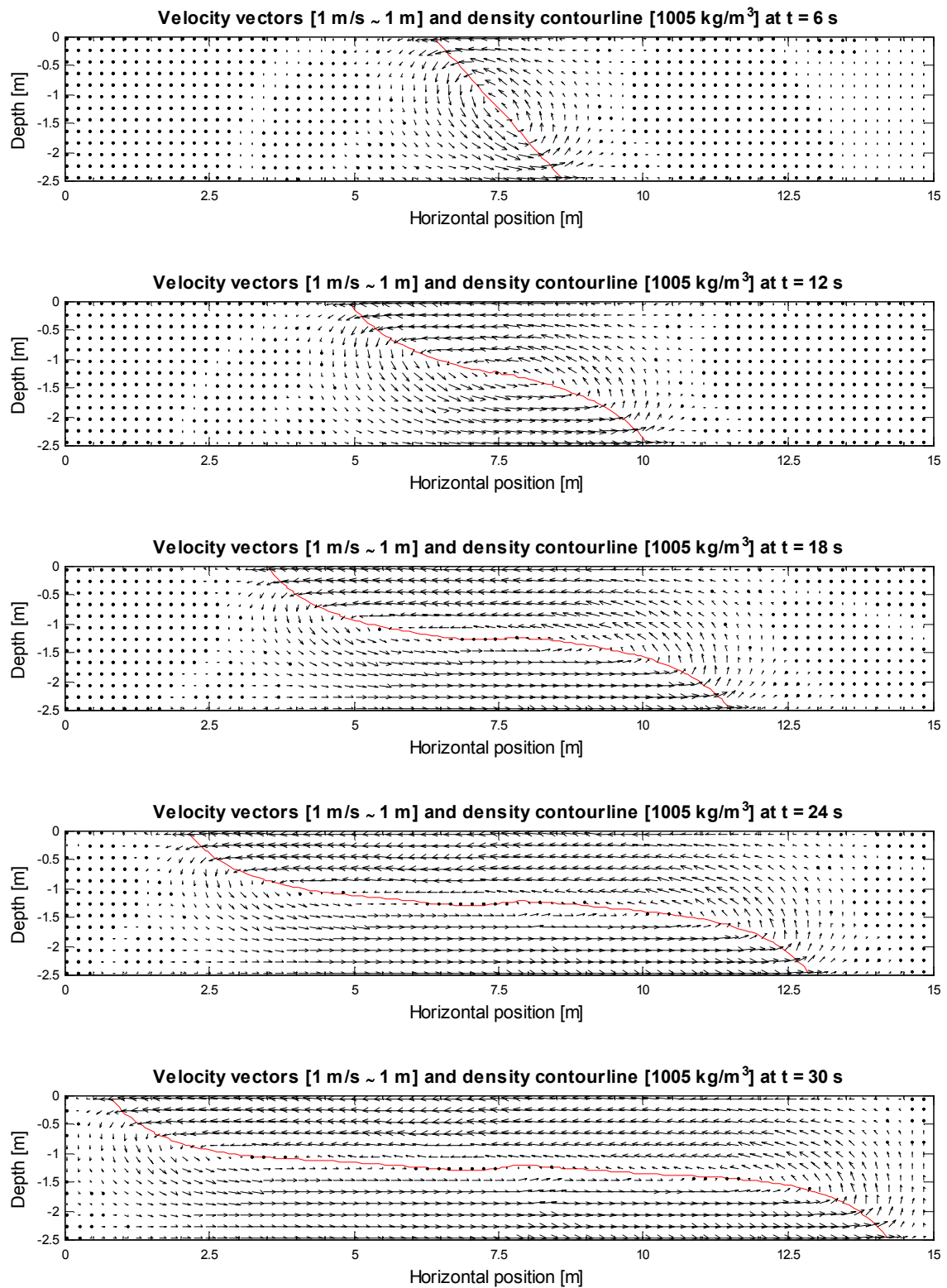


Figure 8-13: Velocity vectors and density contourlines for the Lock Exchange case at $t = 6; 12; 18; 24; 30$ s. Non-hydrostatic simulation - sigma model.

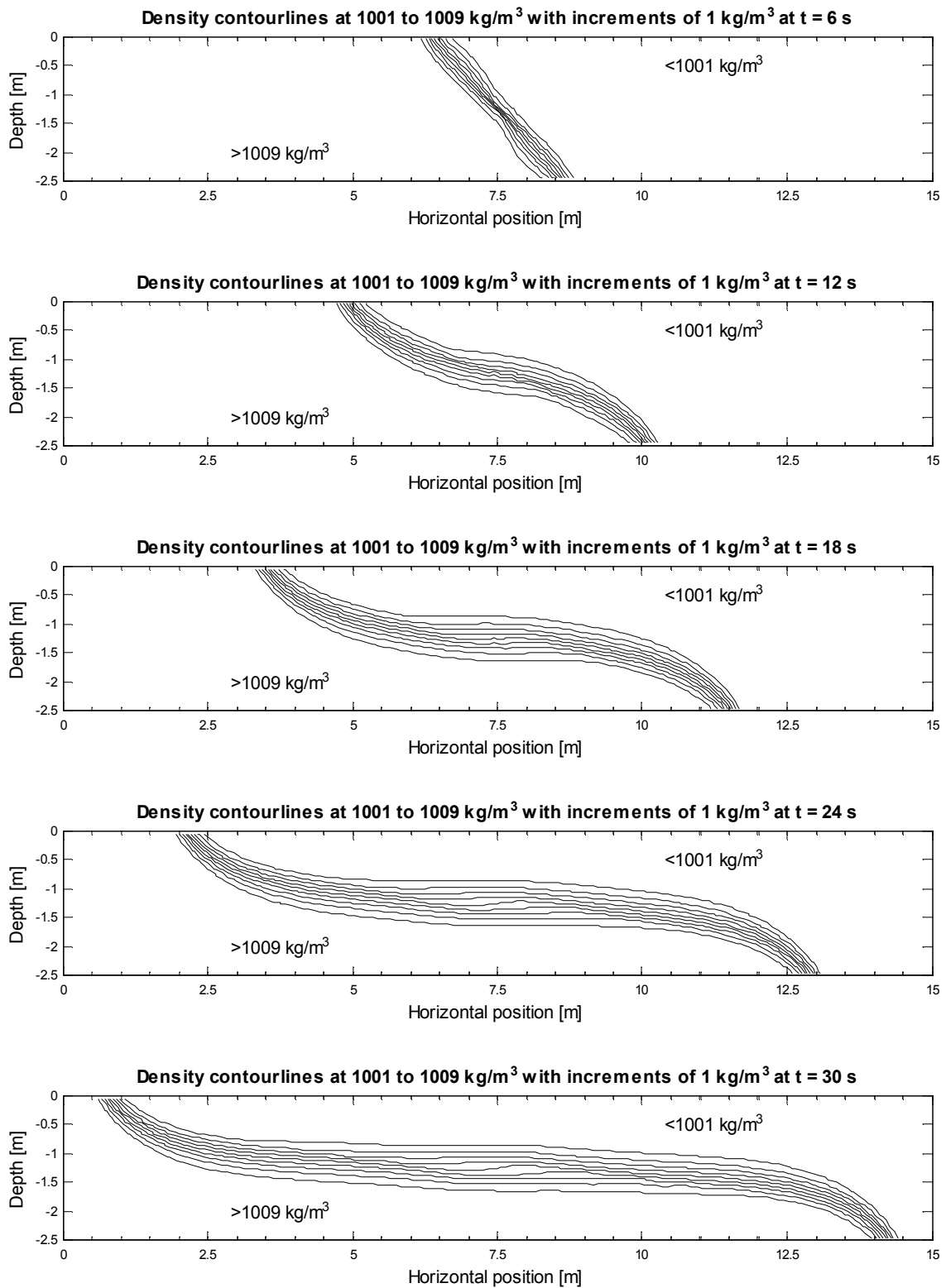


Figure 8-14: *Density contourlines for the Lock Exchange case at t = 6; 12; 18; 24; 30 s. Non-hydrostatic simulation - sigma model.*

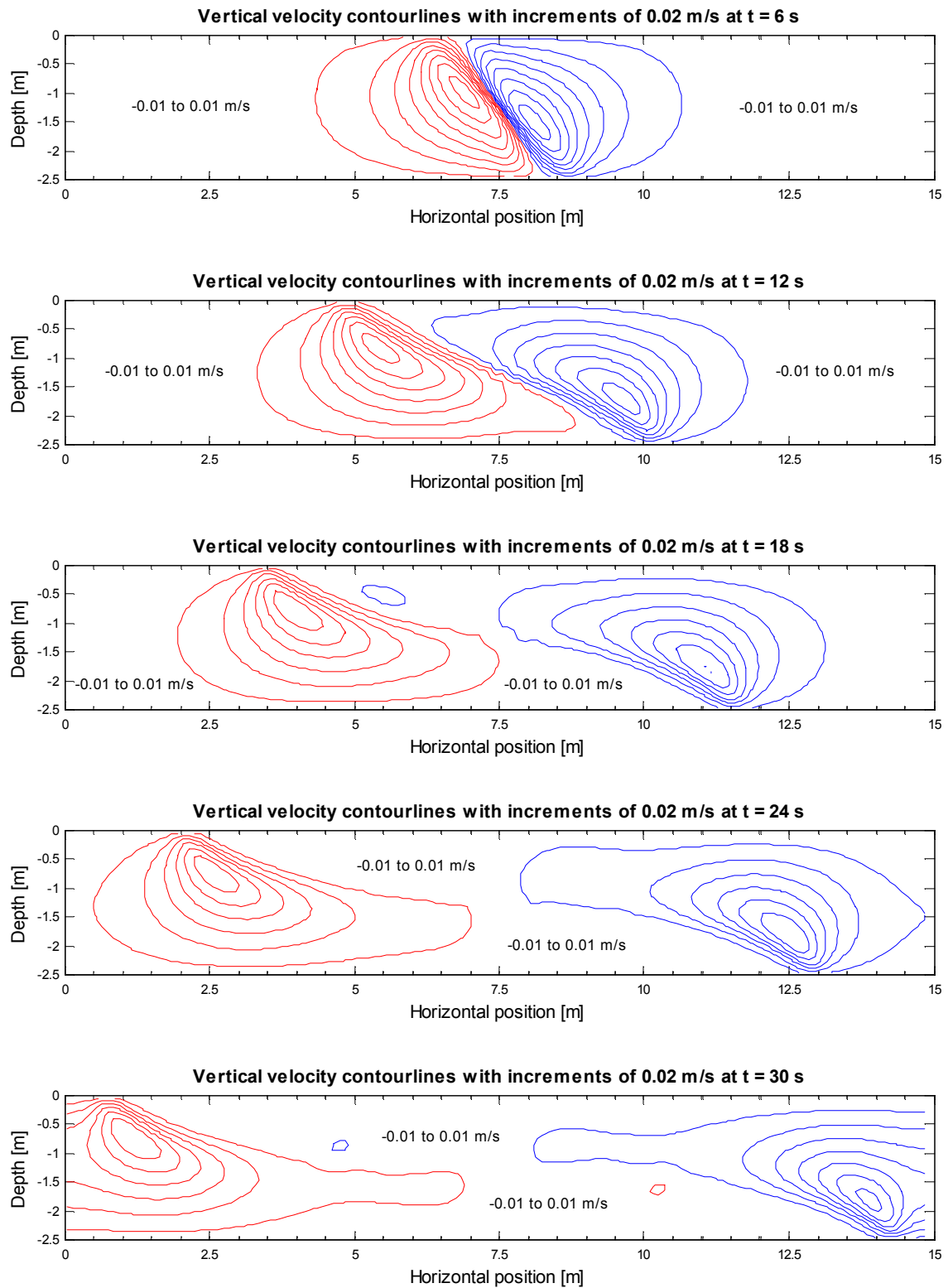


Figure 8-15: Vertical velocity contourlines for the Lock Exchange case at $t = 6$; 12; 18; 24; 30 s. The red and blue lines mark areas with respectively negative and positive velocities; Non-hydrostatic simulation - sigma model.

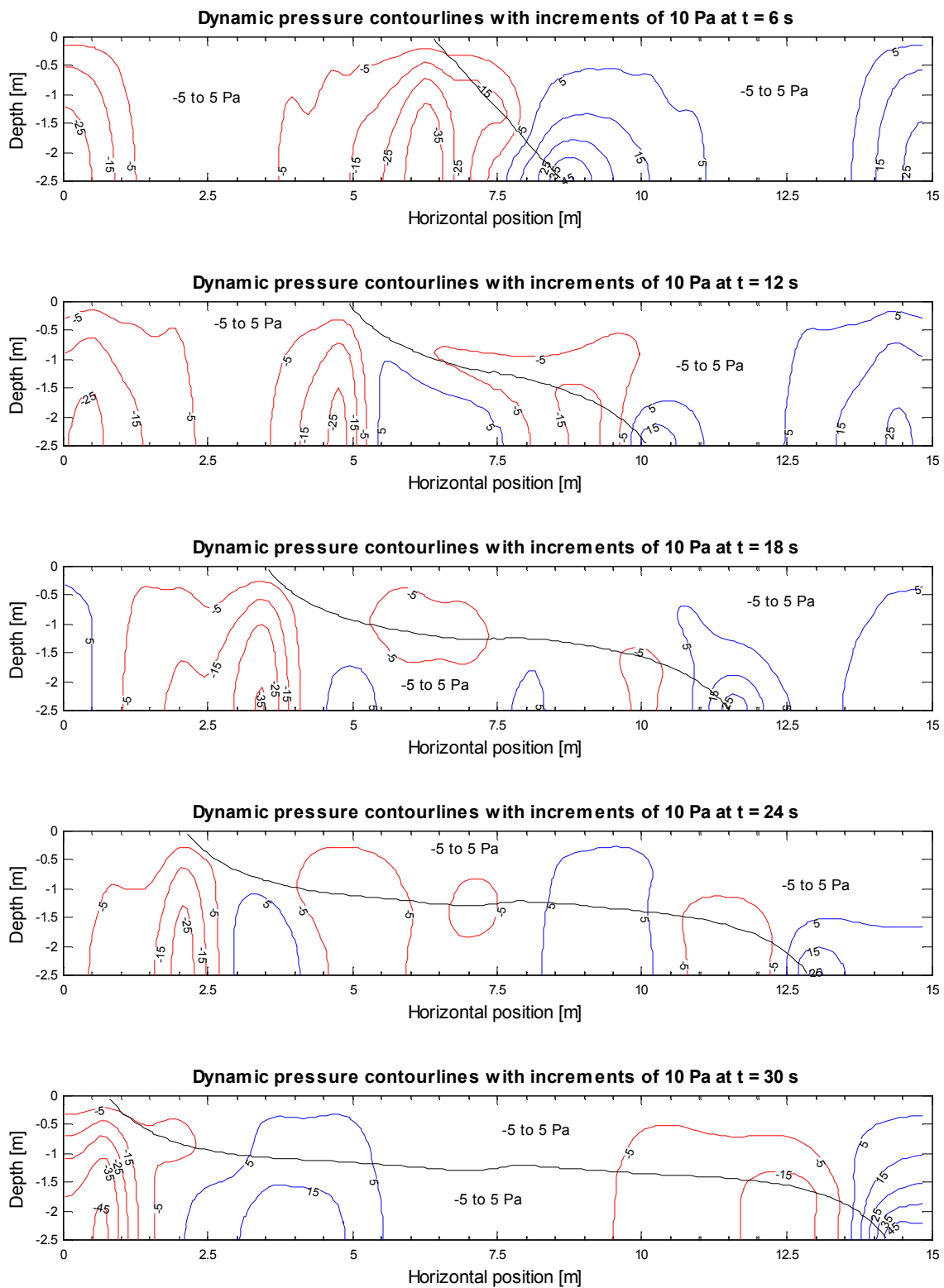


Figure 8-16: *Dynamic pressure contourlines for the Lock Exchange case at $t = 6$; 12; 18; 24; 30 s. The red lines mark areas with respectively under- and over-pressure. Non-hydrostatic simulation - sigma model.*

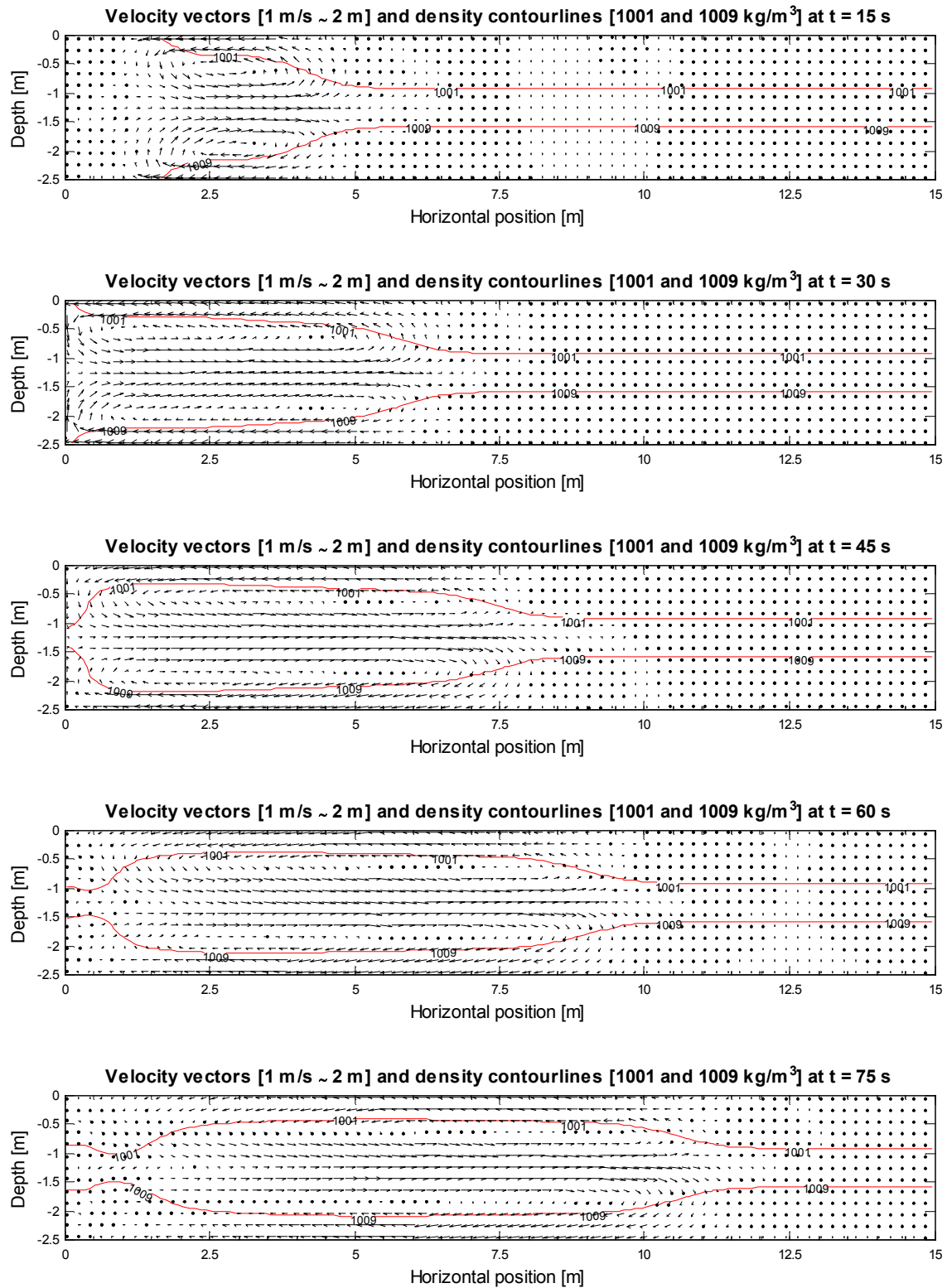


Figure 8-17: Velocity vectors and two density contourlines for the intrusive gravity current case at $t = 15; 30; 45; 60; 75$ s. Non-hydrostatic simulation - sigma model.

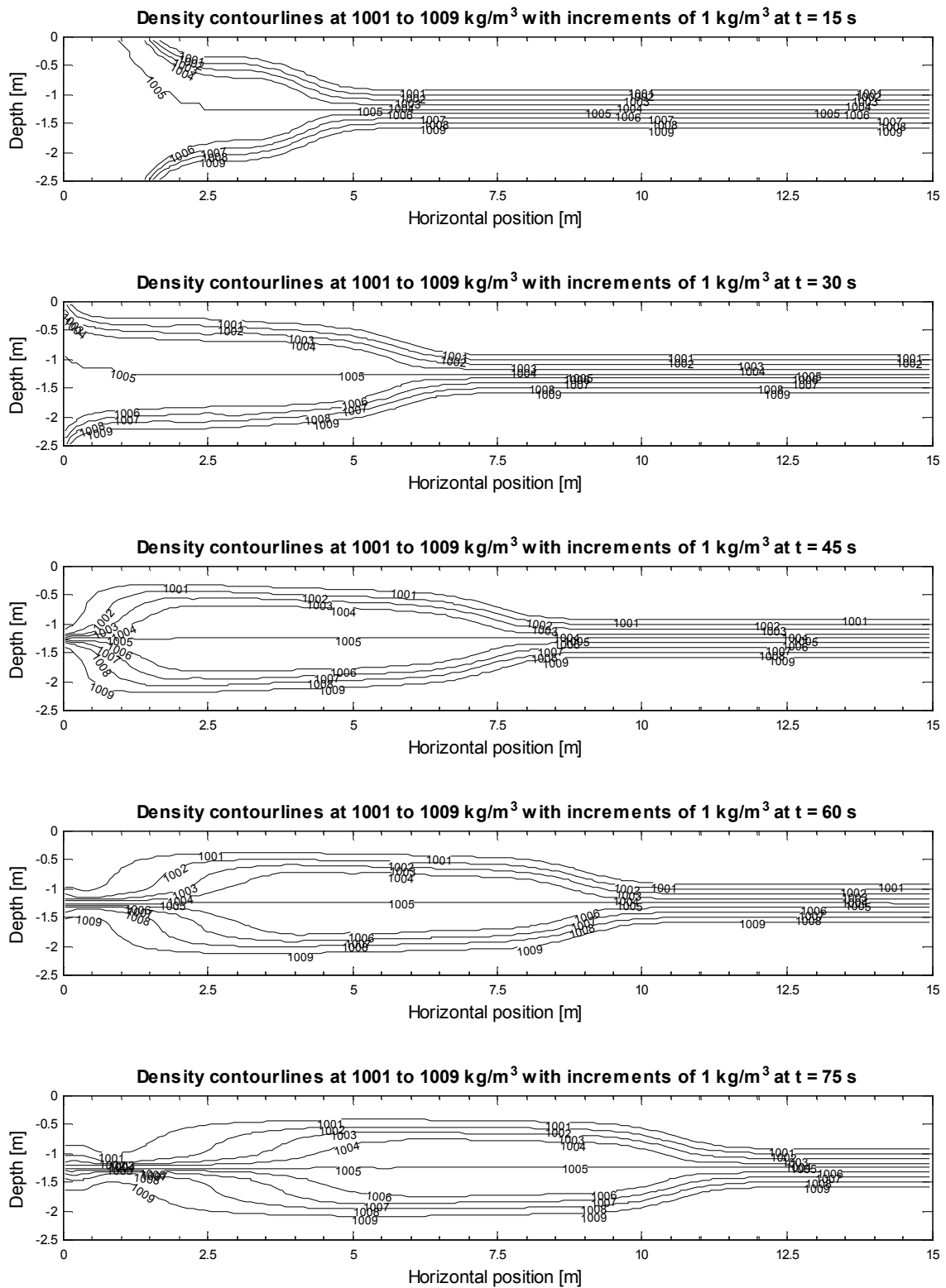


Figure 8-18: *Density contourlines for the intrusive gravity current case at t = 15; 30; 45; 60; 75 s. Non-hydrostatic simulation - sigma model.*

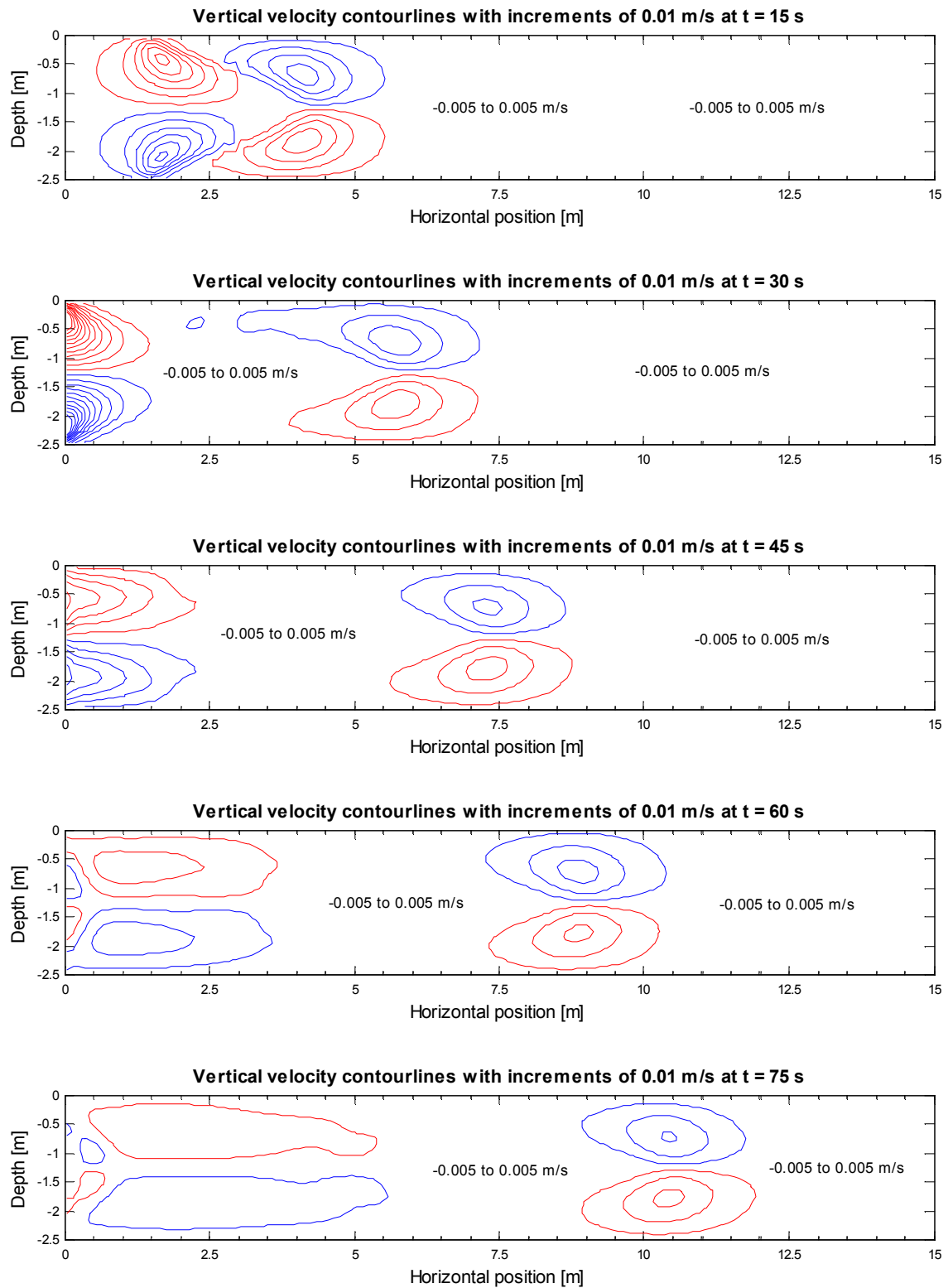


Figure 8-19: Vertical velocity contourlines for the intrusive gravity current at $t = 15$; 30; 45; 60; 75 s. The red and blue lines mark areas with respectively negative and positive velocities. Non-hydrostatic simulation - sigma model.

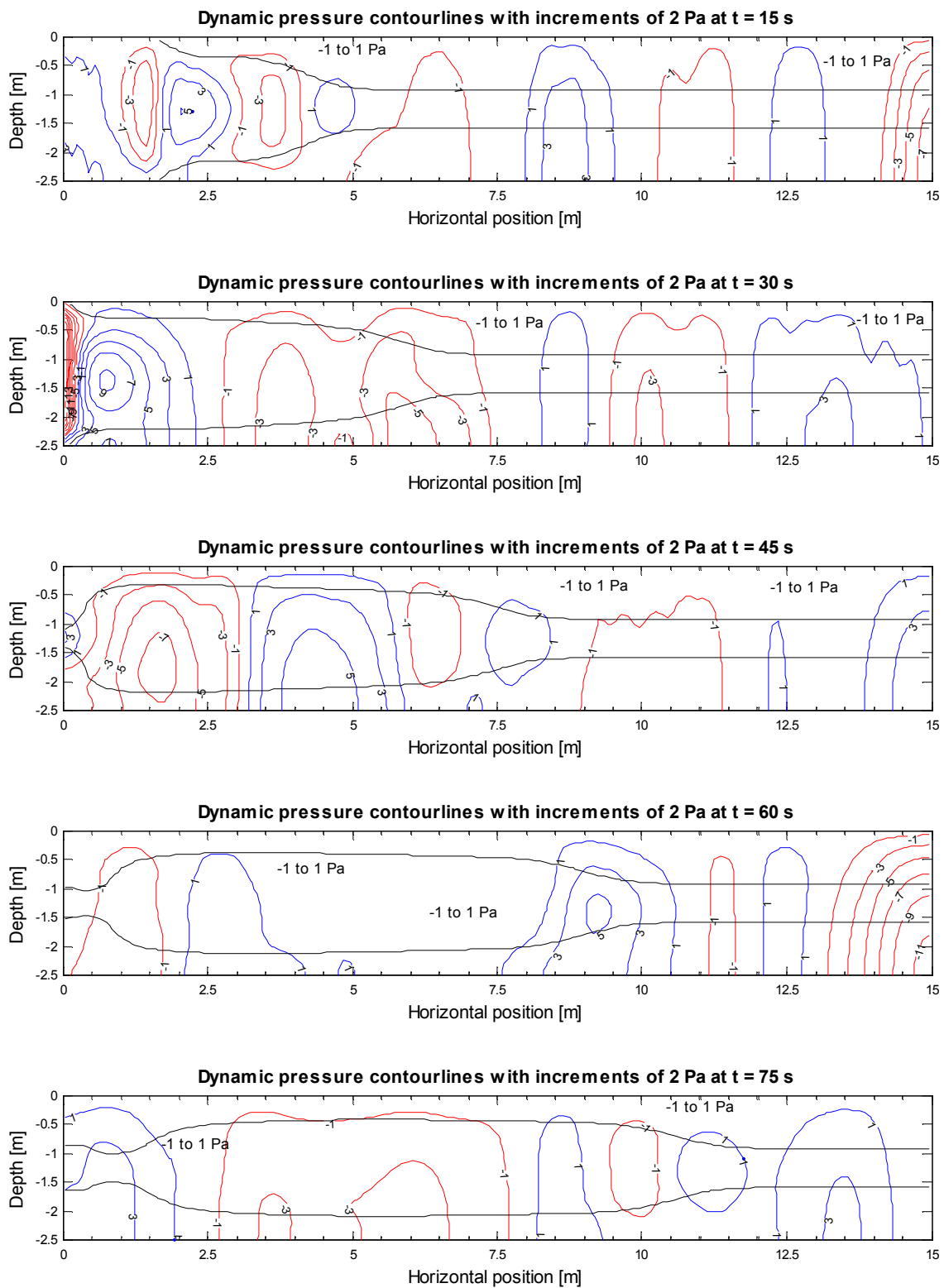


Figure 8-20: *Dynamic pressure contourlines for the intrusive gravity current case current at $t = 15; 30; 45; 60; 75$ s. The red and blue lines mark areas with respectively under- and over-pressure. Non-hydrostatic simulation - sigma model.*

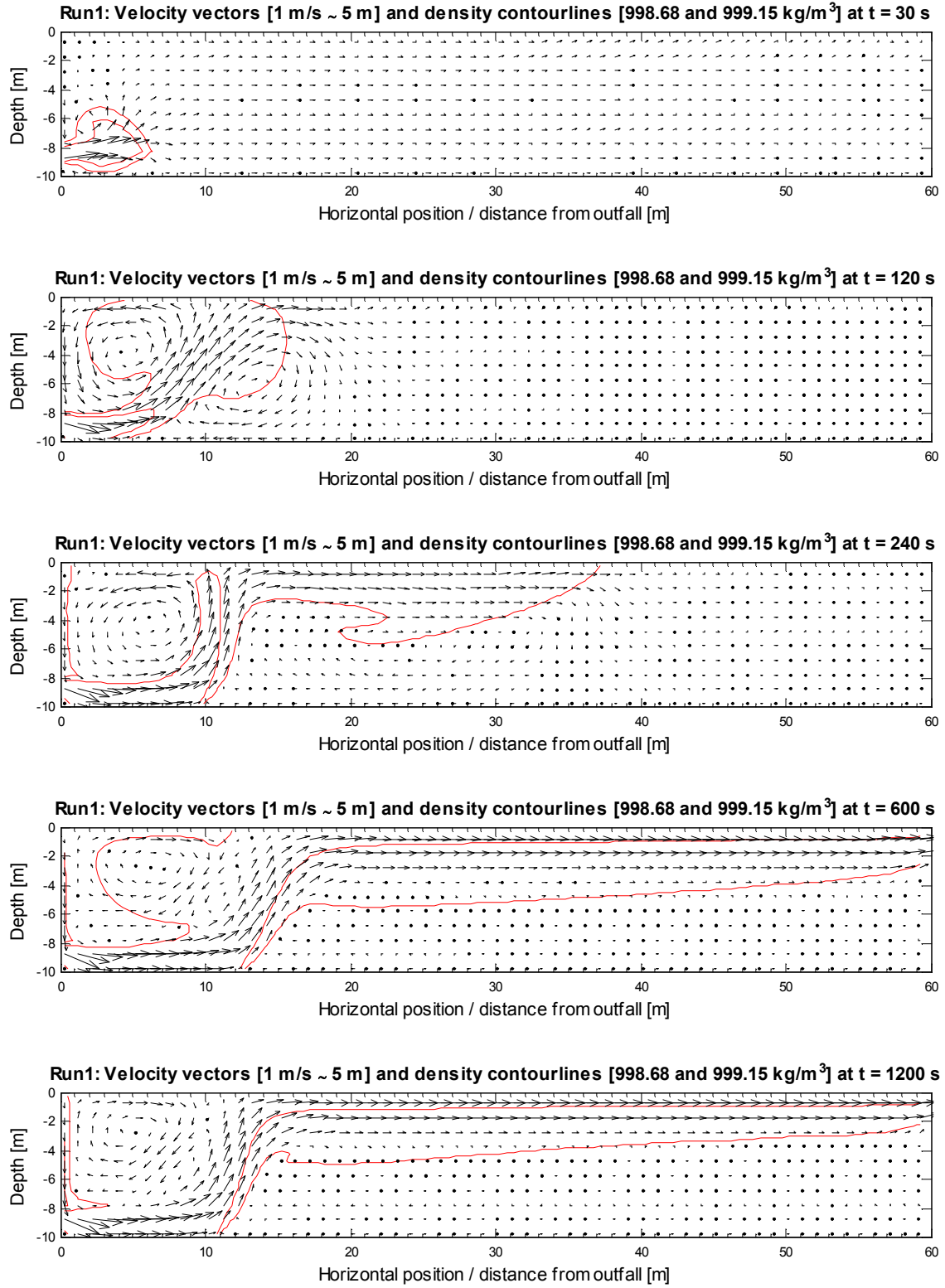


Figure 8-21: *Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for run 1 at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.*

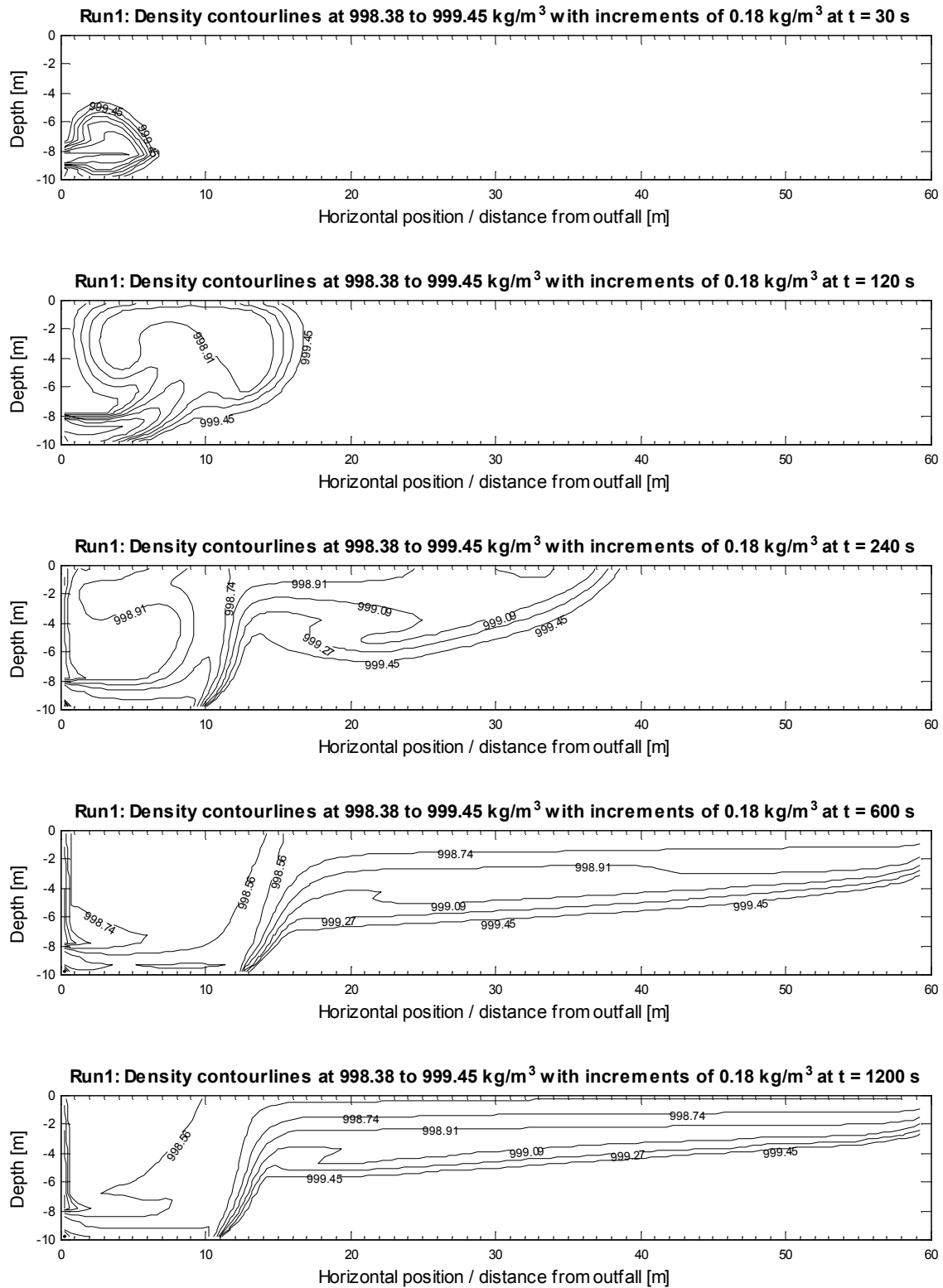


Figure 8-22: *Density contourlines for **run 1** at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.*

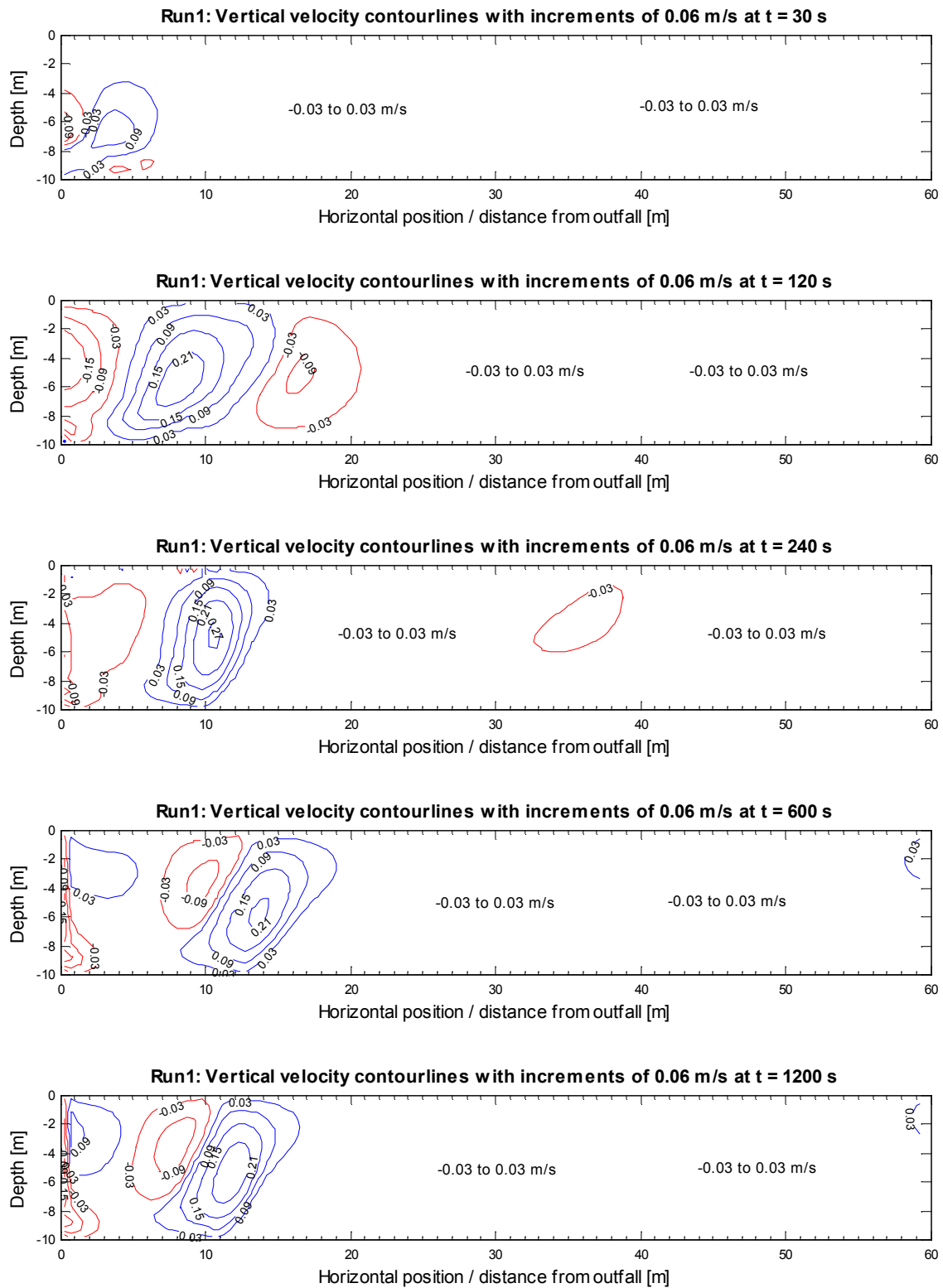


Figure 8-23: Vertical velocity contourlines for **run 1** at $t = 30; 120; 240; 600; 1200$ s. The red and blue lines mark areas with respectively negative and positive velocities. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.

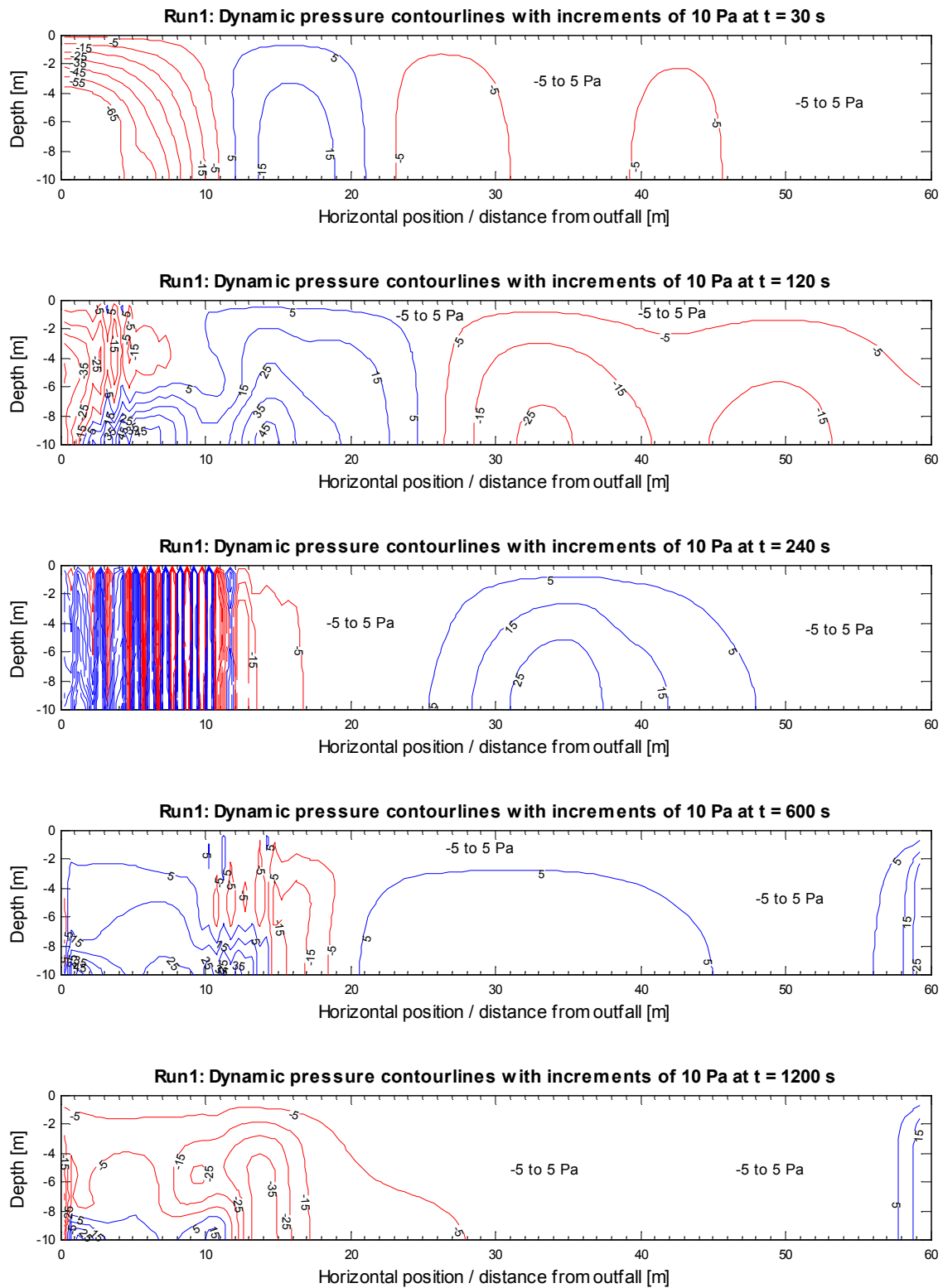


Figure 8-24: *Dynamic pressure contourlines for run 1 at t = 30; 120; 240; 600; 1200 s. The red and blue lines mark areas with respectively under- and over-pressure. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.*

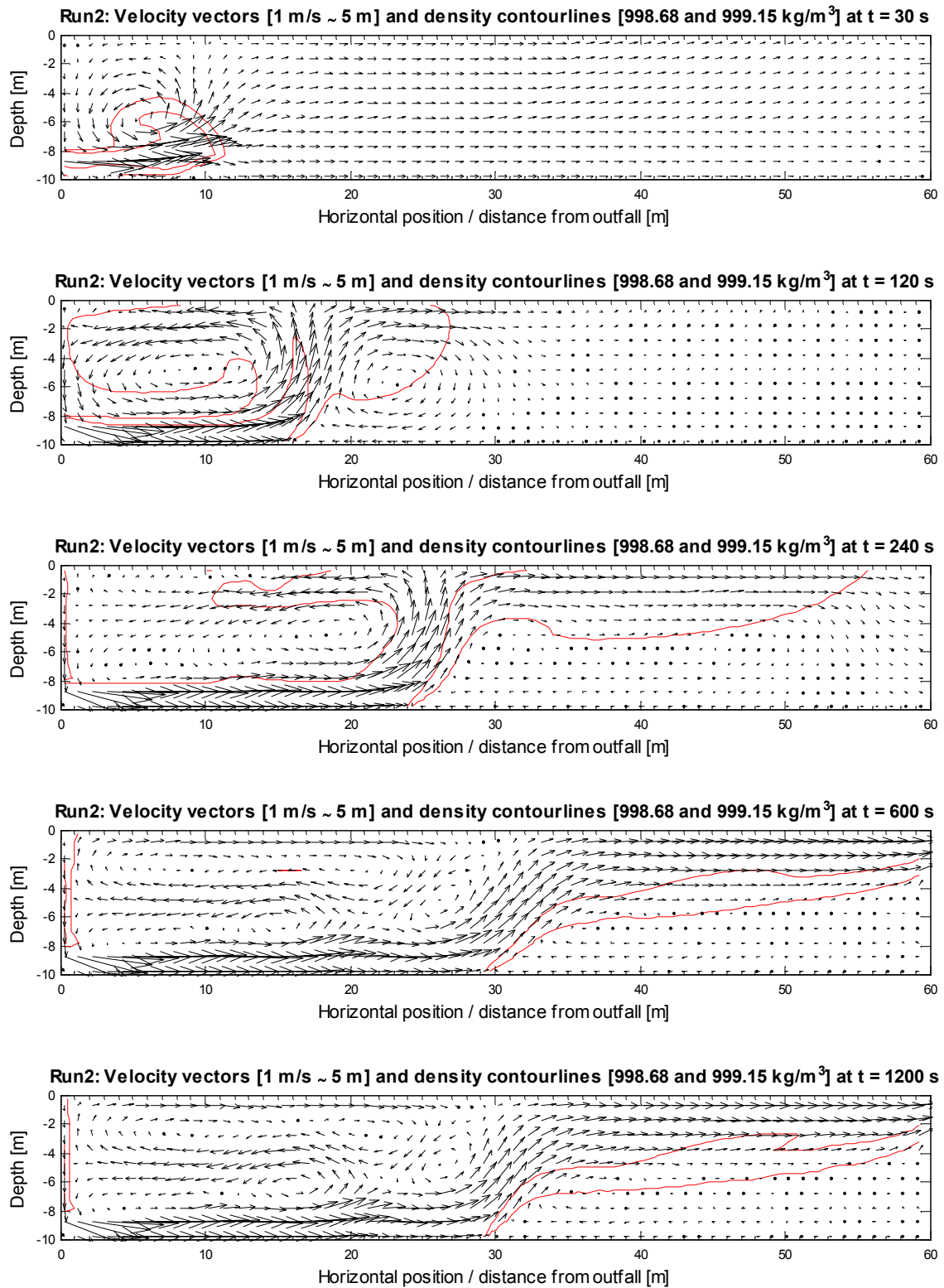


Figure 8-25: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 2** (increased jet velocity) at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.

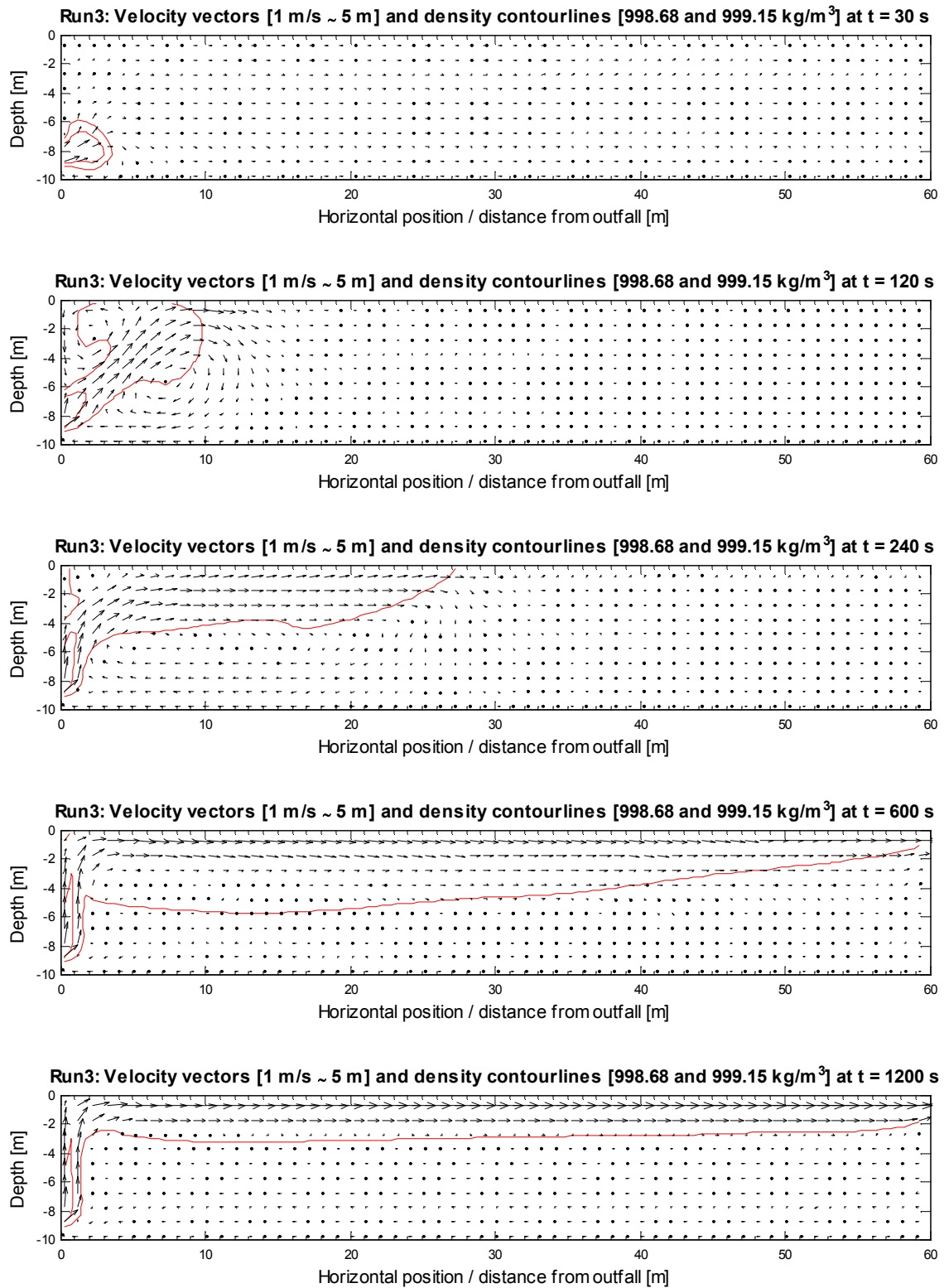


Figure 8-26: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 3** (decreased jet velocity) at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.

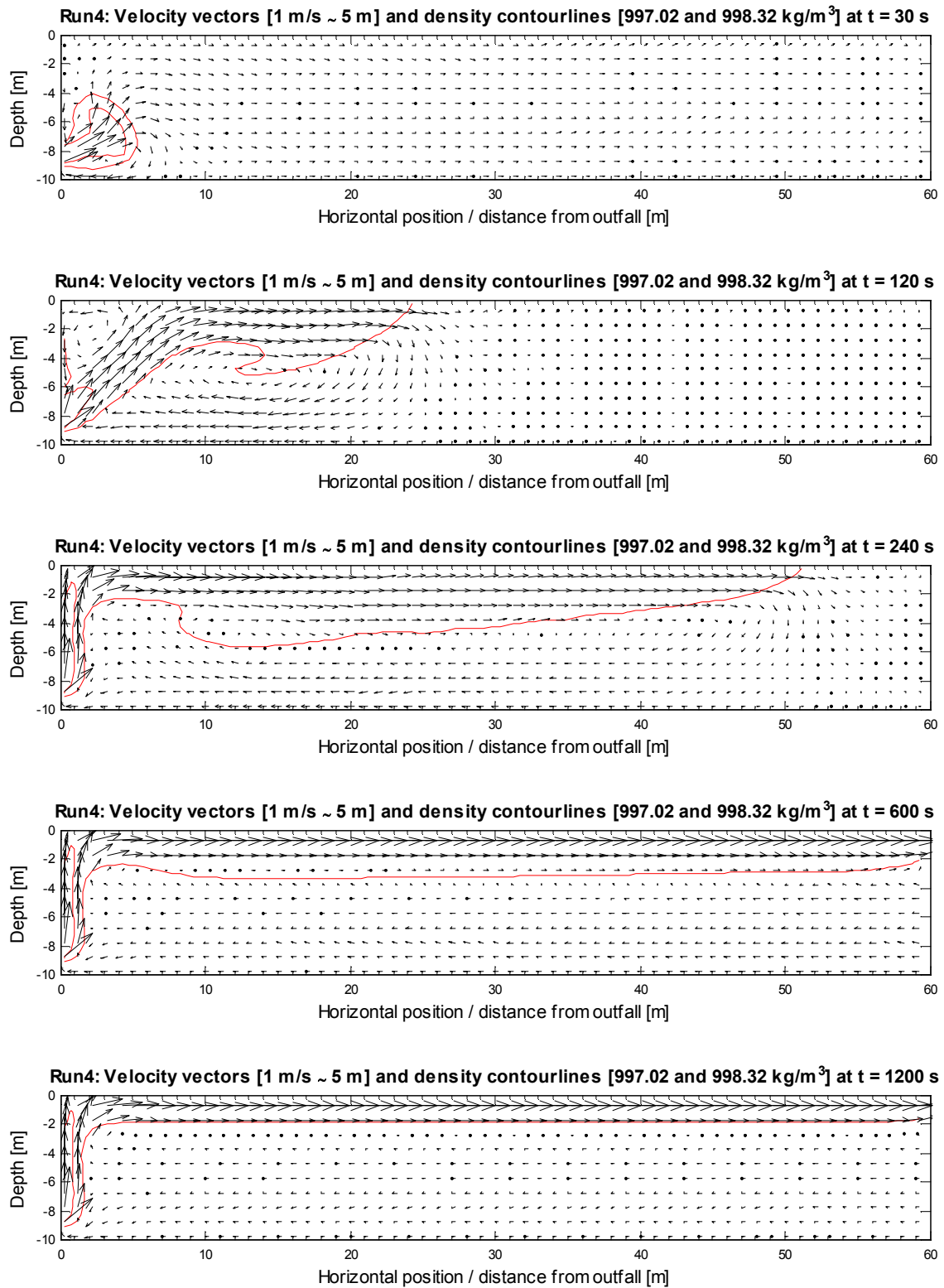


Figure 8-27: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 4** (increased density difference) at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.

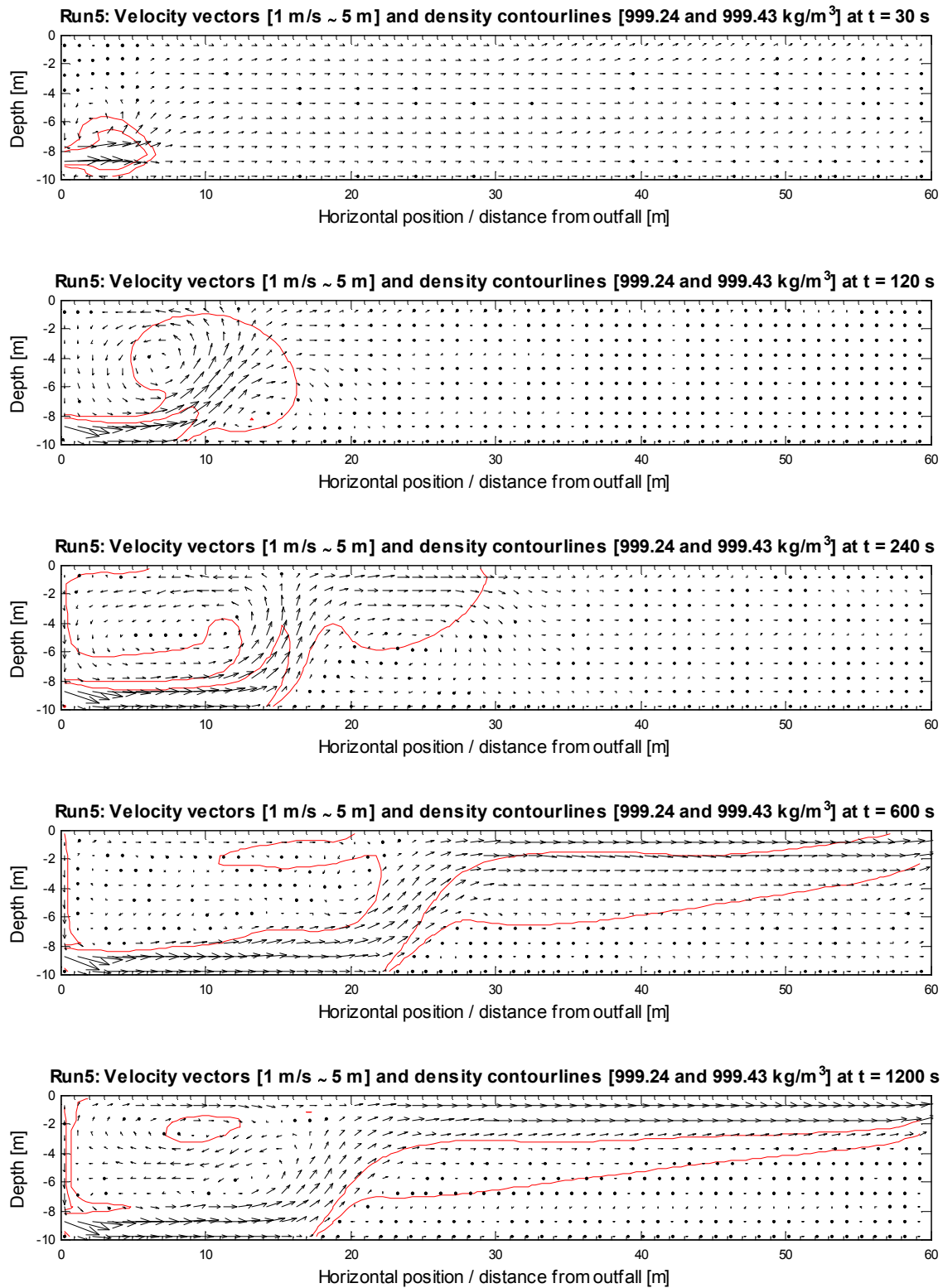


Figure 8-28: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 5** (decreased density difference) at $t = 30; 120; 240; 600; 1200$ s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.

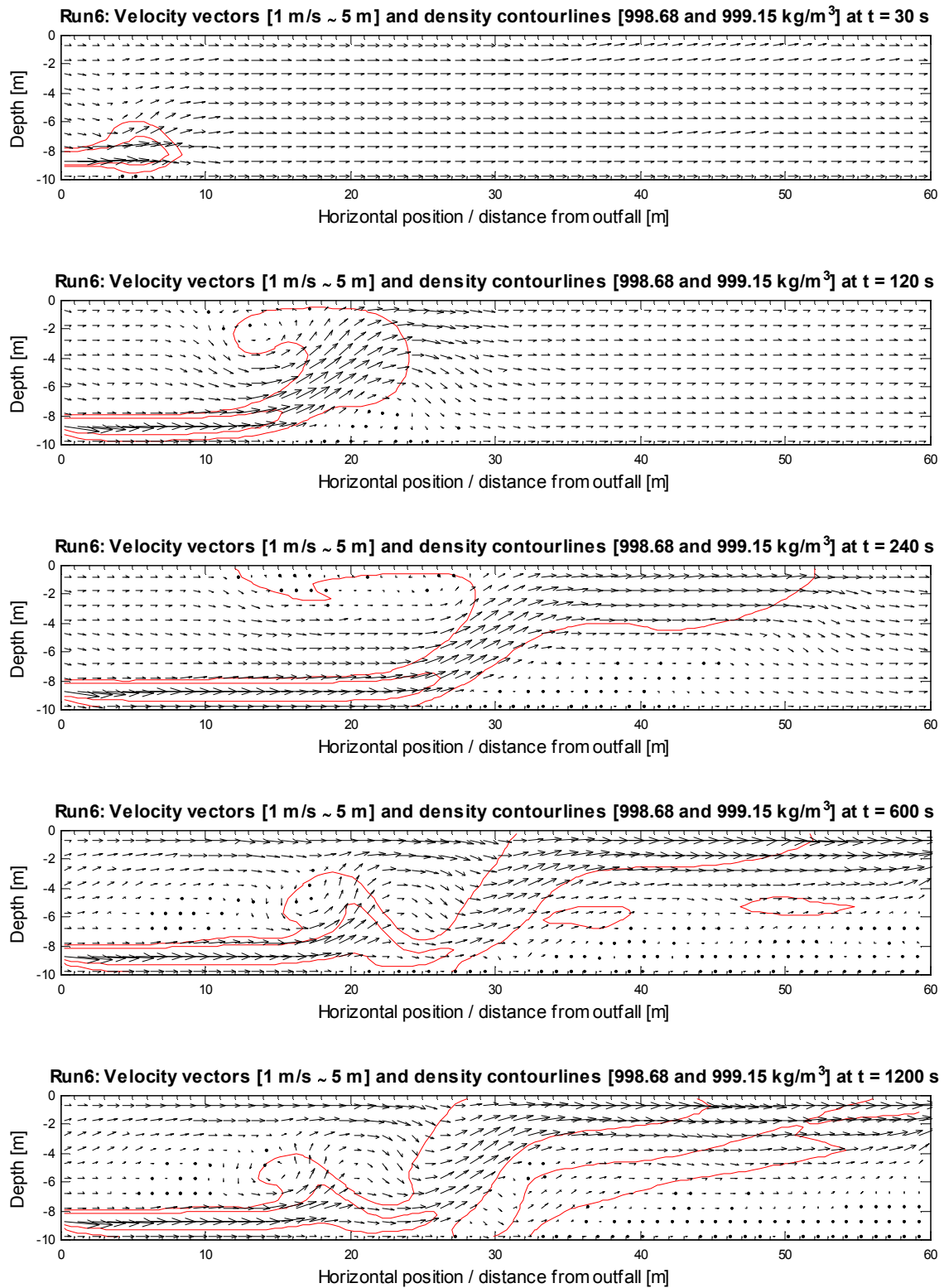


Figure 8-29: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 6** (increased ambient velocity) at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.

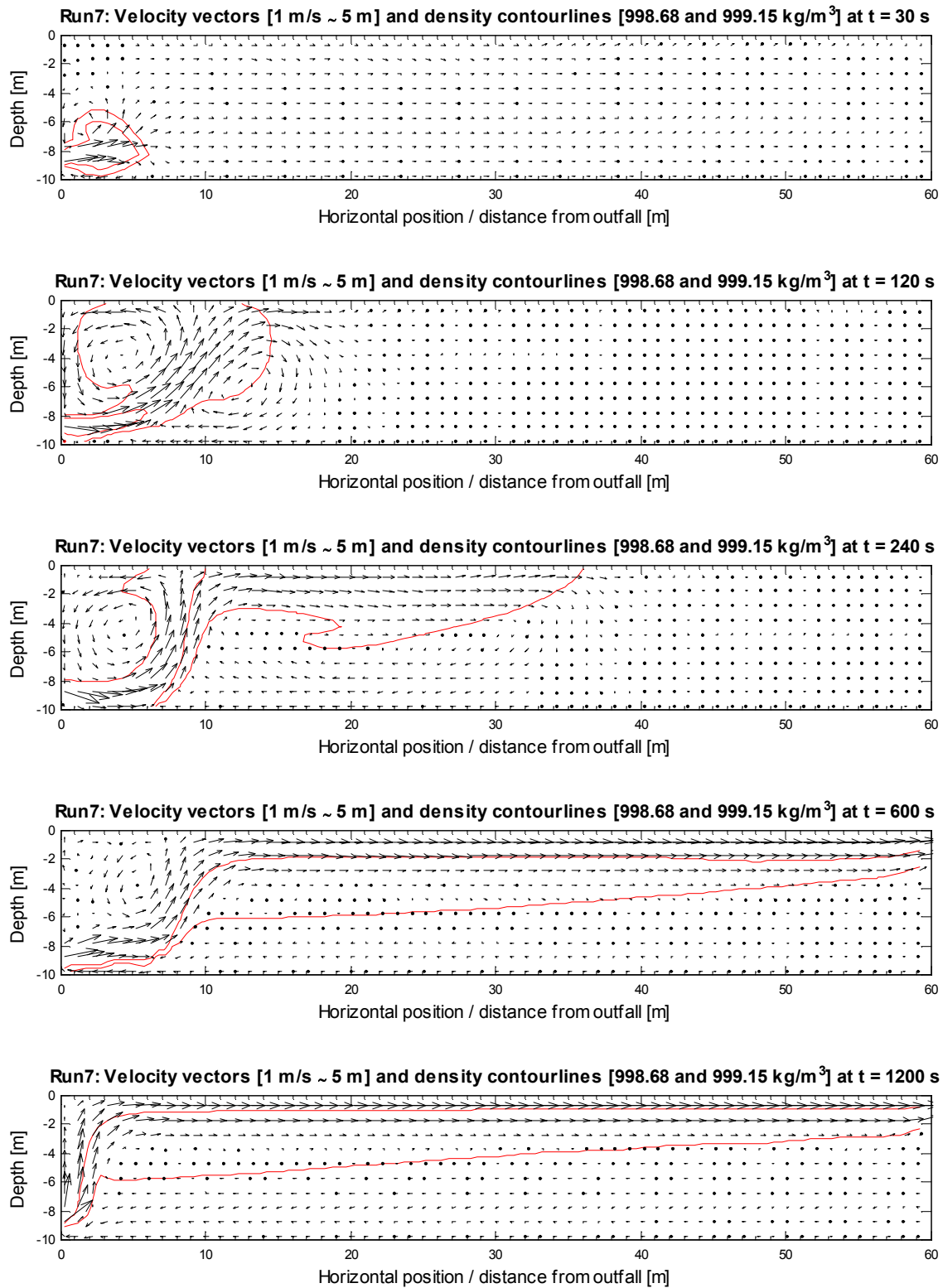


Figure 8-30: Velocity vectors and density contourlines (at 1/3 and 2/3 of the initial density difference) for **run 7** (zero ambient velocity) at t = 30; 120; 240; 600; 1200 s. 2DV hydrodynamic sigma layer model; **Non-hydrostatic** simulation.