UNIVERSITY OF READING

Department of Mathematics

Numerical prediction of ood plains using a Lagrangian approach

Jean-Francois VUILLAUME

October 29, 2012

Declaration A dissertation submitted in partial ful lment of the requirement for the degree of "Mathematics and Numerical Modelling of Atmosphere and Ocean"

" I con rm that this is my own work, and the use of all material from other sources has been properly and fully acknowledged."
Reading, United Kingdom, October 29, 2012

Acknowledgements I would like to acknowledge Prof. Mike Baines for supervising this project and expanding the subject with ourishing numerical ideas.

Thanks to:
Nicholas Bird, Dr Anne Verhoef and Dr. Bruce Main for discussions;

Industri Energi (IE),industry worker union in Norway to provide me with a study grant to nance partially the study

Jenjira, Tar and Laszlo for sharing their warm friendship

Dedication I would like to dedicate this thesis to my family ...

Abstract

Flood events have large consequences on human society in terms of impact on human life and economy. In the current climate change situation, increase of heavy rain will contribute to an increase of ood events both in intensity and frequency. Groundwater ood is a particular ood event which involves the rising of the groundwater table to the surface due to previous in ltration. Numerical modelling codes based on physical laws describing the velocity and water column change are powerful tools for ood simulating extensions and intensity forecasts. We have developed a coupled code consisting of a one dimensional shallow water equation approximation together with a thin lm equation to describe the behaviour of the groundwater ood. A Lagrangian description is used throughout because it is particularly adapted to the problem of both the shallow water and thin lm equations. In addition, high order numerical resolution of high order equation is sought using an

Contents

3.2 Semi-Lagrangian 18

List of Figures

List of Tables

De nition *Groundwater* : Aquifer which is located underground. In this case the aquifer is uncon ned and pressure increase results in the rise of the water column

Open channel ow : Flow of water in a channel in which the top part con-

Chapter 1

Introduction

Flooding research constitutes an important research topic driven by its major impact on society. The damage due to flood is costly and insurance costs to prevent flood damage are high. Worldwide, coastal, riverine and flash floods are responsible for more than 50% of fatalities and for about 30% of the economic losses caused by all natural disasters.

In the United Kingdom, property, land and assets to the value of $214 billion are at risk of flooding in England and Wales. The Environment Agency spends $300 millions a year on flood defences, 43% of existing flood defences being in a fair, poor or very poor state of repair. The damage bill from the devastating floods of 2007 was in excess of $3 billion. A map of England exposure risk to groundwater flood was published by the Environment Agency (Figure 1.1)

These natural hazards have a certain degree of predictability and the keys

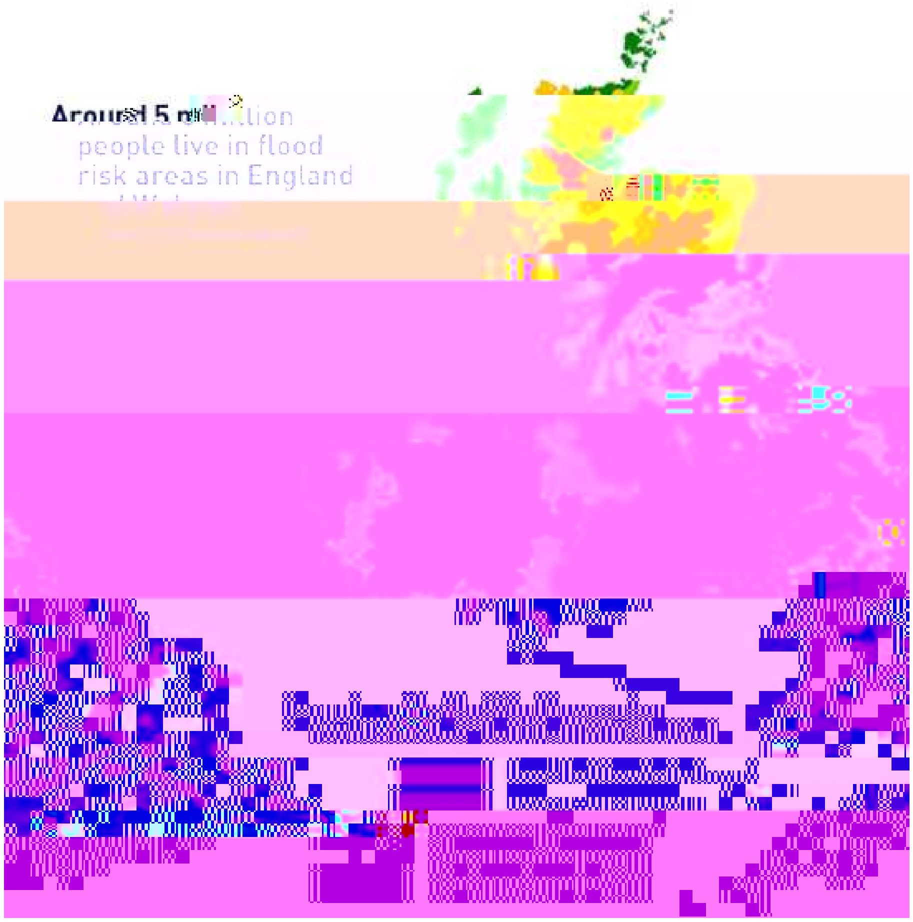


Figure 1.1: Flooding facts in the UK

During this study, only the groundwater flood process after reaching the surface was studied. To model this effect, a one dimensional depth averaged approximation of the Navier-Stokes was used called the Saint-Venant equations.

The Navier-Stokes equations describe the motion of fluid and arise from applying Newton's second law to fluid motion. The Shallow Water Equations (SWEs) are a vertically averaged approximation of the Navier-Stokes equation. The vertical averaging is determined using the boundary conditions and by averaging the velocity over the depth. This dissertation describes the derivation steps of the averaged depth Shallow water equation and the associated approximations. The depth averaged method is a standard current technique to approximate the Navier-Stokes equations to shallow water approximation. The one dimensional approximation of the Shallow water equations are the Saint-Venant equations or open channel flow equations (Olsen, 2012 [1]).

The latest numerical scheme and strategies to optimize flood depth averaged free surface problem have been compiled by Delis et al., 2010 [2]. Furthermore, detail of the derivation of the approximation of the Shallow water equations currently used are developed by Dawson and Mirabito, 2008, [3]. Commercial numerical codes developed since the '60's, include MIKE from Danish Hydrological Institute (DHI), HEC-RAS from the American Hydraulic Engineering Centre, TUFLOW. Recent open source code are now available for flood modelling purpose such as OPEN OpenCFD [4]. Those models are used for channel, overbank flood or heavier rain flood. At present, no model is specifically dedicated to groundwater flooding but there is a growing interest related to the groundwater issue.

Ouput of numerical model are used to produced flooding risk maps as illustrated Figure 1.2 The Saint-Venant equations are coupled to a thin film equation to model the diffusion of the water with low water fluxes.

The thin film equation is a non-linear 4th order equation which describes the spreading of a fluid on a surface. The two equations, Saint-Venant for linear uniform flow, and diffusion, are coupled to describe more precisely a groundwater flood process.

Figure 1.2: Example of ooding risk map (From Ordnance Survey Ireland, All rights reserved. Licence number 2010/15CCMA/Galway County Council)

A recent research project called FUSE (Floodplain Underground SEnsors) [5] uses a high-density, wireless, underground Sensor Network to quantify oodplain hydro-ecological interactions. It investigates the Field groundwater table change with ood consequence. It allows monitoring of the groundwater level by geophysics, i.e. electromagnetic methods, with a high resolution. In the long term the project will improve our understanding of groundwater ood forecast and could possibly be used as an early alert tool for groundwater oods.

Numerical modelling software is a key tool to address the degree of predictability of a ood event: Numerical modelling improvements will minimize the damage by improving the precision of ood front location and the height of the water wave. The present project also makes use of a Lagrangian frame of reference, resulting in interesting and challenging numerical modelling issues with maximum time steps in coupling the systems of equations: Saint-Venant for laminar ow and thin lm for non-linear di usion, using a moving mesh strategy based on velocities.

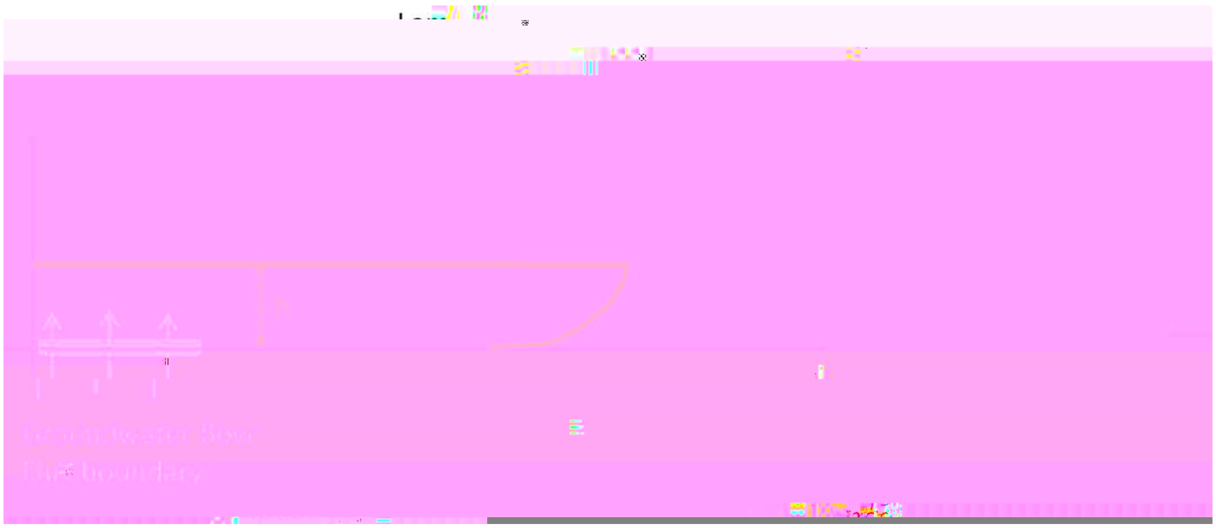


Figure 1.3: Schematic model of the numerical problem (*From Baines et al., 2012, Unpublished*)

The thin film equation is a diffusion problem described by a 4th non linear differential order equation. It describes the spreading of a thin film on a surface. Several publications are related to this problem which have several industrial applications (O'Brien and Schwartz, 2002 [6]). In our case, problem of capillary and inter-facial tension is neglected due to the dynamic of the fluid.

The coupling of the two problems is a based on theoretical assumptions which haven't been developed and published yet. Hence, the use of this idea will be submitted to criticism based on this first project. The problem covered by this work is illustrate by the Figure 1.3 where the two flow domains are illustrated. The location of the groundwater source flow at the left boundary and the moving free boundary at the right hand of the model are illustrated as well.

Summary The rst chapter introduces the objectives of the project and the limitations of the model and discusses recent advances in ood numerical modelling. Chapter 2 reviews the moving mesh method and the velocity moving mesh method used for this problem. Chapter 3 describes the Saint-Venant equations, 1D approximation of the shallow water equations (also called the open channel ow equations). Mathematical and numerical scheme development is described including the Lagrangian frame of reference. Chapter 4 describes a version of the 4th ib 4thIthe th(cal)]To/.oof

Chapter 2

A moving mesh : A velocity-based moving mesh method

This chapter describes a method based on a moving mesh strategy. This method uses a velocity based moving mesh method, particularly well-adapted for Lagrangian uid movement. Moving meshes or dynamic meshes are a numerical modelling strategy to minimize the number of grid points used for a dynamic problem compared to a static grid while preserving the physics of phenomena. The method chosen is based on local mass conservation for each discretized element (Baines et al., 2011 [7]), which is consistent global mass conservation. The method allows the con guration of the velocity of the mesh for each nodal point of the mesh (Bhattacharya, MSc 2004 [8]). The local consevation is assured for each time step. The velocity is obtained

2.1 Finite Di erence Methods

The Finite Di erence Method (FDM) is used is this project. Finite Element Method was used by Bhattacharya. B., 2004 [8] and Baines et al., 2005 [11] with successful results. The coupled method consists of discretizing the gives:

$$(x_{i+1} - x_{i-1})h_i = {}_i \tag{2.4}$$

for

$$i = 2, \dots, N - 1$$

Next, the new location of the node is computed and height of the water column is recomputed based on equation 2.3 for each h_i

2.4 Moving mesh

Knowing the velocity of the node at each node, the new location of the nodes x_i^{n+1} can be calculated form the previous mode location x_i^n by the expression:

$$x_i^{n+1} = x_i^n + v_i \quad dt \tag{2.5}$$

where v_i represent the velocity of each node and dt the time step used. Care have to be taken for the chose of dt, high value of the time step could conduct to node overlapping since the node velocity are different. In the other hand, low dt value will conduct to slow the computation code.

Chapter 3

One-dimensional shallow water equation approximation: The Saint-Venant approximation

The Saint-Venant equation describes a one-dimension (1D) approximate shallow water flow used currently for open channel flow. It can be used for a one dimension water flow problem as a simplification of a two-dimension (2D) problem in a 1D context. The equations which described the flow process are derived from the mass conservation and momentum conservation.

Eulerian and Lagrangian descriptions of fluid constitute two ways to describe fluid movement (Price (2006 [12]). The Eulerian approach supposes a fixed reference and the Lagrangian approach a coordinate system moving with the fluid particles. Due to the nature of flooding, the Lagrangian description

3.1 Eulerian description of fluid

There are several schemes available to solve the Saint-Venant equations. For the Eulerian approach, we used space central difference method corrected by a predicator corrector as illustrated by Olsen., 2012 [1].

3.1.1 Mathematics equations

We define the variables:
h: The height of the water column, a function of x and t
x: Position in the x direction
u: Velocity of the water in the x direction, also a function of x and t

where h_j^n is the height of the water column at node j and at time t_n of the domain $x = [0; 1000]$.
Also,

$$\frac{\partial(uh)}{\partial x} \approx u_i^n \frac{h_{i+1}^n - h_{i-1}^n}{2\Delta x} + h_i^n \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x}; \tag{3.3}$$

leading to :

$$\frac{h_i^{n+1} - h_i^n}{\Delta t} + u_i^n \frac{h_{i+1}^n - h_i^n}{2\Delta x} + h_i^n \frac{u_{i+1}^n - u_i^n}{2\Delta x} = 0 \tag{3.4}$$

Hence,

$$h_i^{n+1} = h_i^n - \frac{\Delta t}{2\Delta x}(u_i^n(h_{i+1}^n - h_i^n) + h_i^n(u_{i+1}^n - u_i^n)) \tag{3.5}$$

where dx is the spacial increment and dt the temporal increment. For the equation the explicit form is given by: Which lead to:

$$\frac{u_i^{n+1} - u_i^n}{\Delta t} = u_i^n \frac{u_{i+1}^n - u_i^n}{\Delta x} + g \frac{h_{i+1}^n - h_i^n}{\Delta x}$$

$$u_i^{n+1} = u_i^n - \frac{\Delta t}{\Delta x}(u_i^n(u_{i+1}^n - u_i^n) + g(h_{i+1}^n - h_i^n)) \tag{3.6}$$

The semi-implicit form is defined by:

$$u_i^{n+1} = u_i^n - \frac{\Delta t}{\Delta x}(u_i^n(u_{i+1}^n - u_i^n) + g(h_{i+1}^{n+1} - h_i^{n+1})) \tag{3.7}$$

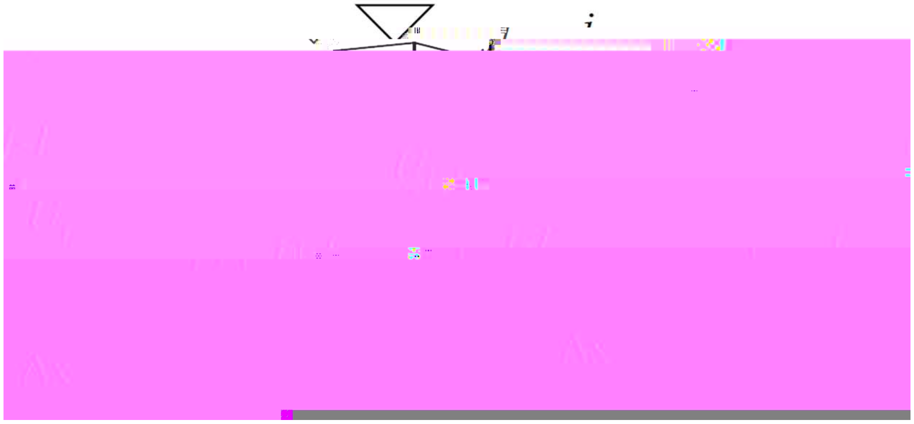


Figure 3.1: Figure for control volume approach to discretization of continuity equation [1]

where u is defined as:

$$u = \frac{u_i^n + u_i^{n+1}}{2}$$

An illustration of the of the approach related to the Equation 3.8 is illustrated Figure 3.1. $i-1; i; i+1$ represent three cross sections, $j-1; j$ two surfaces

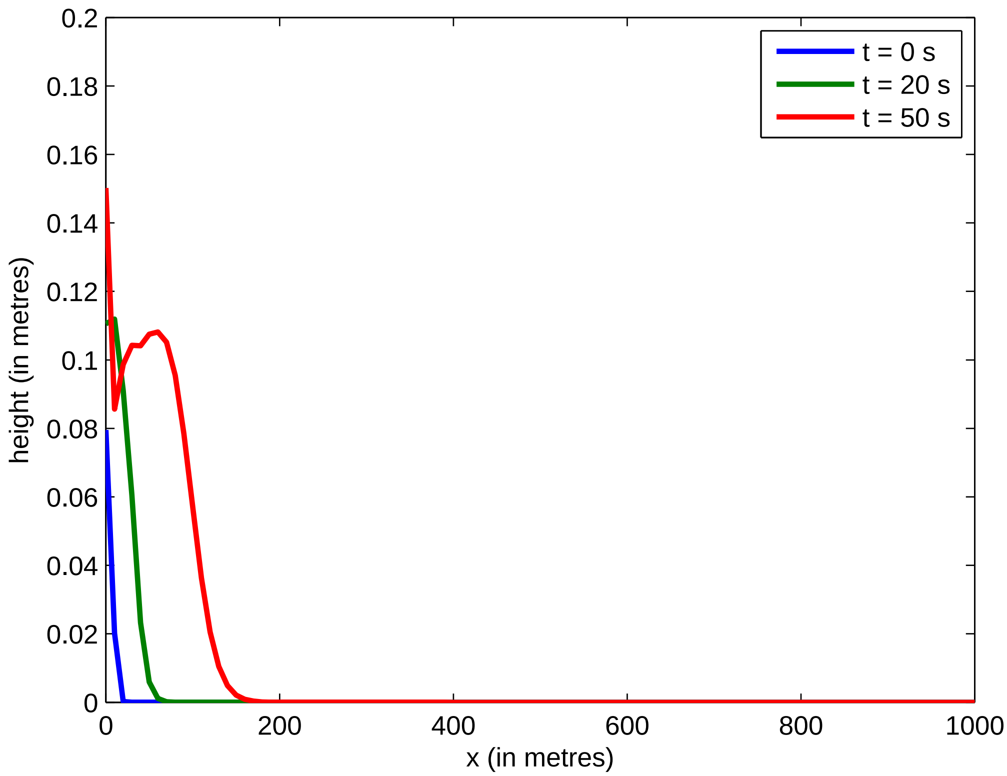


Figure 3.2: 1D Saint-Venant ow at t = 100 s (Computation FORTRAN, visualization MATLAB)

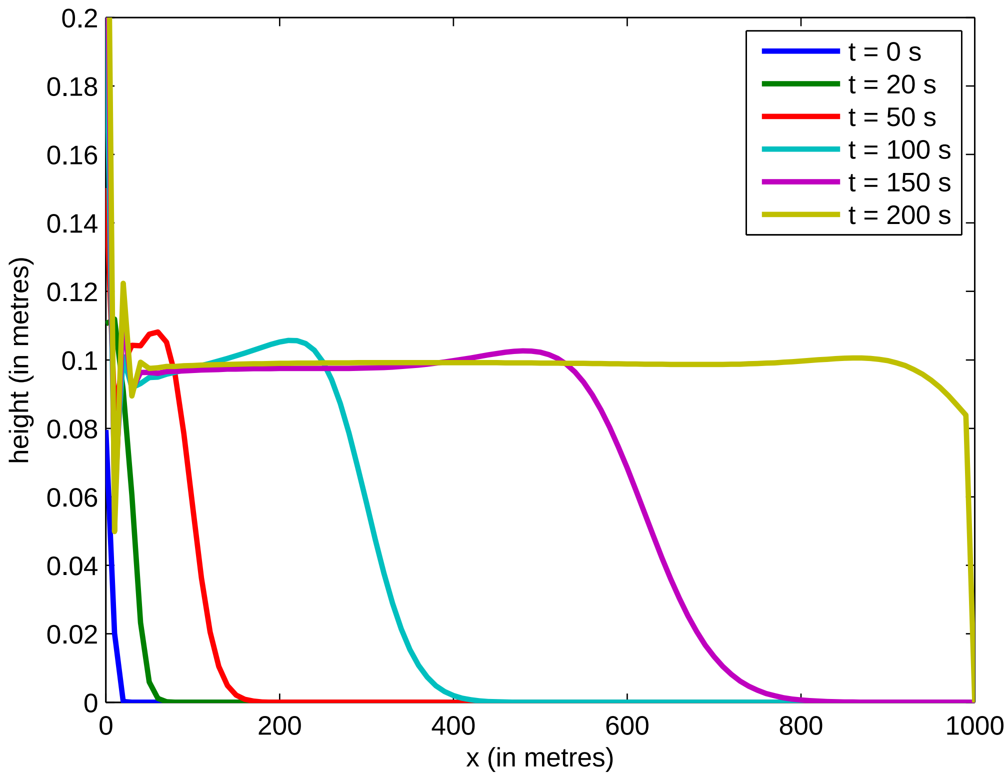


Figure 3.3: 1D Saint-Venant ow at t = 200 s (Computation FORTRAN, visualization MATLAB)

3.2 Semi-Lagrangian description of fluid movement or trajectory-based method

The Semi-Lagrangian description of fluid flow is used to simplify the problem. For the Saint-Venant approximation, two steps are needed for the method. First, the height of the water column has to be computed based on the velocity by the continuity equation. In the next step, the new velocity is determined by the momentum conservation equation. We used the definition of Lagrangian fluid movement:

$$\frac{Dh}{Dt} = \frac{\partial h}{\partial t} + u\frac{\partial h}{\partial x} \tag{3.9}$$

3.2.1 Mathematical equations

We have the following equations:
Continuity equation:

$$\frac{\partial h}{\partial t} + \frac{\partial (uh)}{\partial x} = 0 \ , \ \frac{Dh}{Dt} + h\frac{\partial u}{\partial x} = 0 \tag{3.10}$$

Momentum equation:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + g\frac{\partial h}{\partial x} = 0 \ , \ \frac{Du}{Dt} + g\frac{\partial h}{\partial x} = 0 \tag{3.11}$$

3.2.2 Finite difference discretization

We express the previous equation using a Lagrangian derivative, expressing the new height, new velocity and new position of the water wave: Continuity equation:

$$\frac{Dh}{Dt} + h\frac{\partial u}{\partial x} = 0$$

$$\Leftrightarrow \ \frac{h_i^{n+1} - h^n}{\Delta t} + h_i^n \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x} = 0$$

$$\Leftrightarrow \quad \frac{h_i^{n+1} - h^n}{\Delta t} = -h_i^n \frac{u_{i+1}^n - u_{i-1}^n}{2\Delta x}$$

$$\Leftrightarrow \quad h_i^{n+1} = h^n - h_i^n (u_{i+1}^n - u_{i-1}^n) \frac{\Delta t}{2\Delta x}$$

Momentum conservation equation:

$$\frac{Du}{Dt} + g\frac{\partial h}{\partial x} = 0$$

$$\Leftrightarrow \quad \frac{u_i^{n+1} - u_i^n}{\Delta t} + g\frac{h_{i+1}^n - h_{i-1}^n}{2\Delta x}$$

$$\Leftrightarrow \quad u_i^{n+1} = u_i^n - g\frac{\Delta t}{2\Delta x}(h_{i+1}^n - h_{i-1}^n)$$

3.2.3 Boundaries conditions

Left boundary condition

The Left boundary condition is set as constant flux boundary of water. It represents a local punctual groundwater flood in flux.

Right boundary condition

The right boundary x_N is constrained by the length location of the last point. Flux at the end point is computed using velocity and height at this location.

Visualization

The results of the Saint-Venant numerical model in Lagrangian reference are illustrated Figure 3.4 and Figure 3.5 at time t = 10 s, t = 70 s. The space time step was $dx = 50m$, with $N = 21$ nodes and $dt = 0.01s$.

3.3 Fully Lagrangian fluid dynamic description: The Verlet scheme

A fully Lagrangian description needs $x(t; \ )$ independent of dx, where is

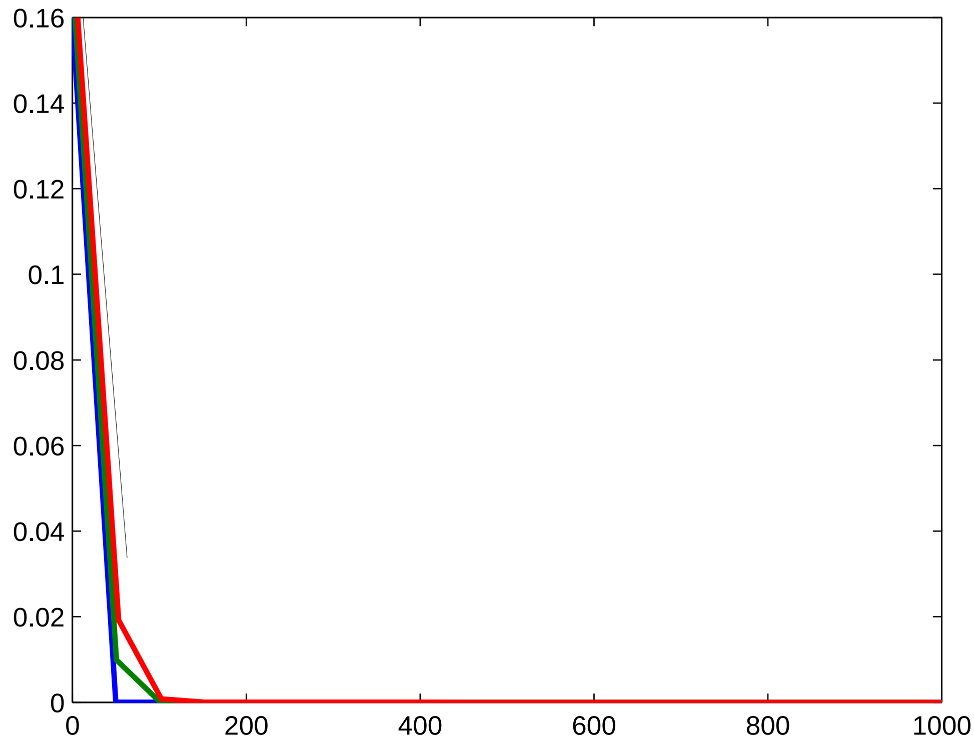


Figure 3.4: 1D Saint-Venant ow at t = 10 s (Computation FORTRAN, visualization MATLAB)

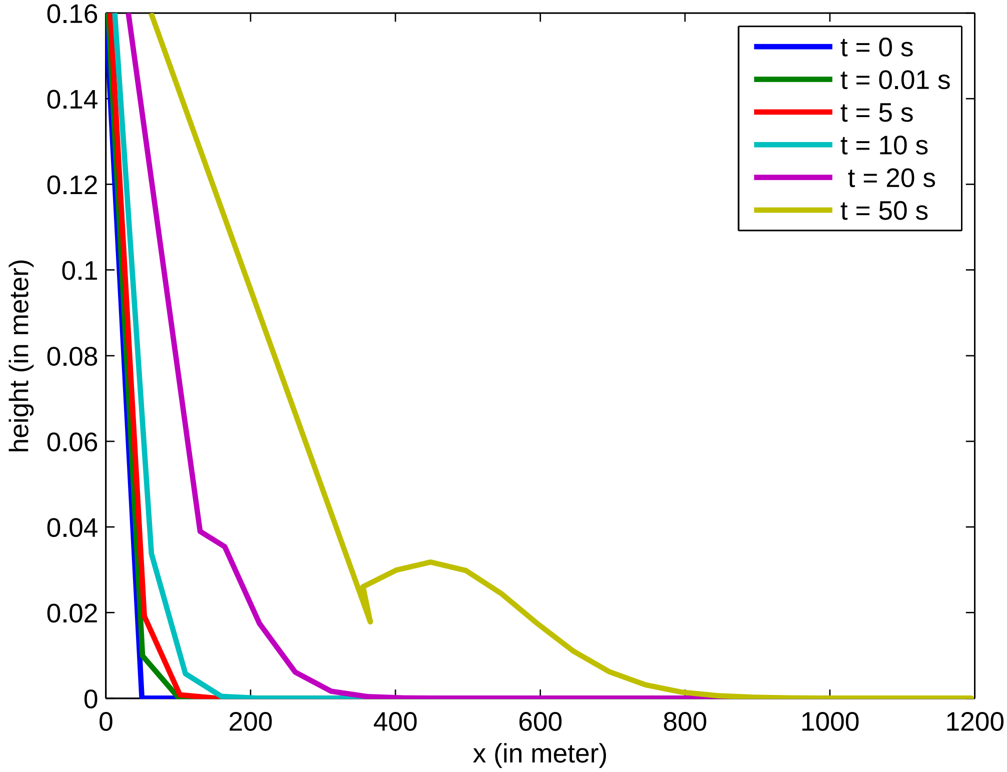


Figure 3.5: 1D Saint-Venant ow at t = 70 s (Computation FORTRAN, visualization MATLAB)

develop a full Lagrangian description of the uid particle.

3.3.1 Mathematical formulation

3.4.1 Discretization and algorithm implementation

We start from the momentum equation in a Lagrangian domain:

$$\frac{Du}{Dt} = \quad g\frac{h \quad h}{x} \tag{3.13}$$

To compute the mesh location, we use the Velocity Verlet formulation (Dummer et al., 2012 [13]).
First step :

$$\frac{x_i^{n+\frac{1}{2}} \quad x_i^n}{\frac{t}{2}} = u_i^n \tag{3.14}$$

Second step : $h^{n+\frac{1}{2}}$ is computed by the relation

$$h^{n+\frac{1}{2}} \quad x^{n+\frac{1}{2}} = \quad c \tag{3.15}$$

$$\frac{u_i^{n+1} \quad u_i^n}{t} = \quad \frac{gh^{n+\frac{1}{2}} \quad h}{c} \tag{3.16}$$

$$u_i^{n+1} = u_i^n \quad \frac{gh^{n+\frac{1}{2}} \quad h^{n+\frac{1}{2}}dt}{c} \tag{3.17}$$

Third step :

$$\frac{x_i^{n+1} \quad x_i^n}{t} = \frac{1}{2}(u_i^n + u_i^{n+1}) \tag{3.18}$$

3.4.2 Numerical Results of Saint-Venant equations using the Verlet scheme

Initial conditions

The initial conditions are not known for this equation because no exact solution is available for this problem. We tested three di erent height initial conditions for:

1. $h^0 = a \quad 1 \quad \frac{x^2}{4} \quad ^2$ with $a = 0.01; 1...10$ and $x = [0; 2]$ The result obtained are illustrated Figure 3.6. The displacement of the water wave, it location and height is illustrated. After the rst time step, the height of the water column at the left boundary drop to it real dynamic height maintain by the in ow.

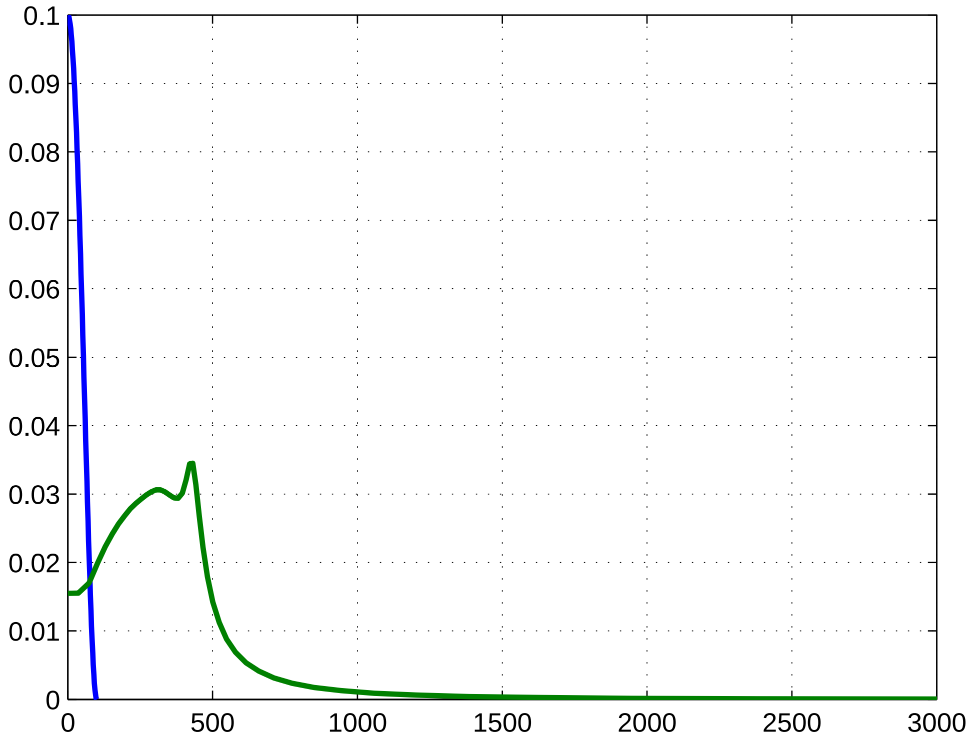


Figure 3.6: Results of the computation of the Saint-Venant equations using the Verlet scheme, N = 51 nodes and initial condition $h^0 = a\left(1 - \frac{x^2}{4}\right)^2$ (Computation and visualization tool MATLAB).

Figure 3.7: Results of the computation of the Saint-Venant equations using the Verlet scheme, N = 11 nodes and initial condition $h^0 = a \quad (1 \quad tanh(x))$ (Computation and visualization tool MATLAB).

2. $h^0 = a \quad (1 \quad tanh(x)$ with $a = 0.01, 1 \ldots 10$ and $x = [0, 2]$ The result of the computation of the Lagrange Saint-Venant equations using the Verlet scheme is illustrated Figure 3.7. It shows the evolution of the water height column over time.

3. $h^0 = 1$ for $x = [0, 50]$ and $h^0 = 1 \quad {\frac{x^2}{4}}^{2}$ for $x = [50, 100]$, , $dt = 0.00001s$, $dx = 1m$. The result of the computation of the Lagrange Saint-Venant equations formulation using the Verlet scheme is illustrated Figure 3.8 and zoom on the wave front illustrated Figure 3.9 and the left boundary Figure 3.10 and the oscillation on the front Fig-

ure 3.11. It shows the evolution of the water height column over time. Oscillation due to the sharp initial condition and the explicit method (Figure 3.11). The Figures 3.8 and 3.11 illustrates the instability of

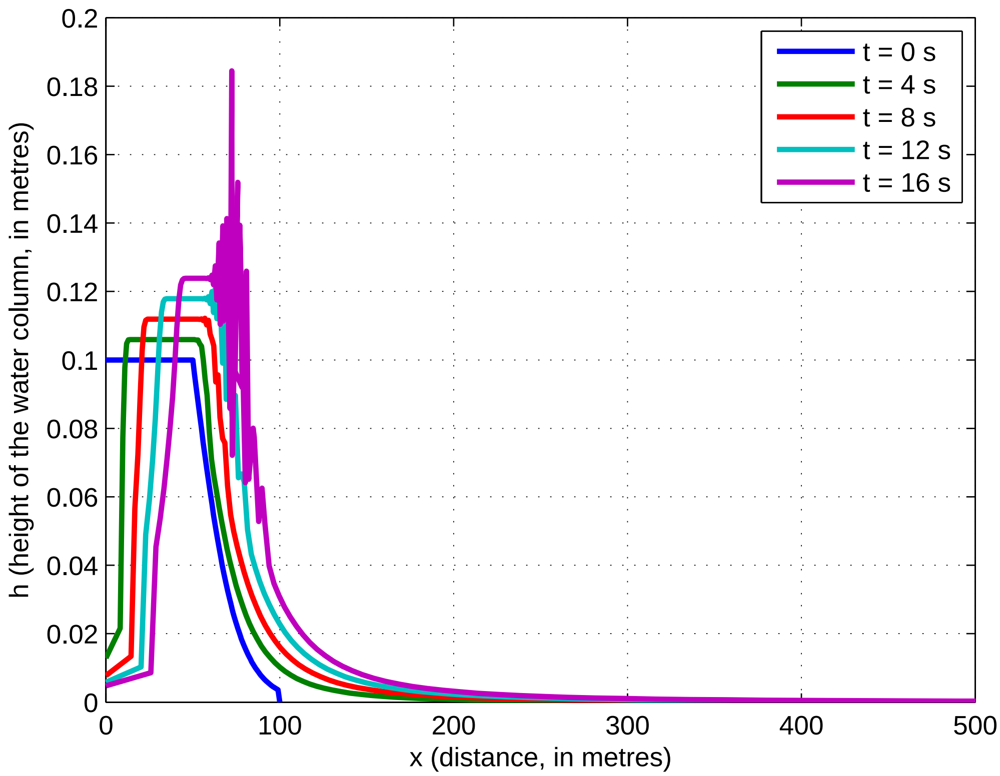


Figure 3.8: General results of the computation of the Saint-Venant equations using the Verlet scheme, N = 101 nodes and initial condition $h^0 = 1$ for $x = [0; 50]$ and $h^0 = \left(1 \quad \frac{x^2}{4}\right)^2$ for $x = [50; 100]$, $dt = 0{:}00001s$, $dx = 1m$ (Computation and visualization tool MATLAB).

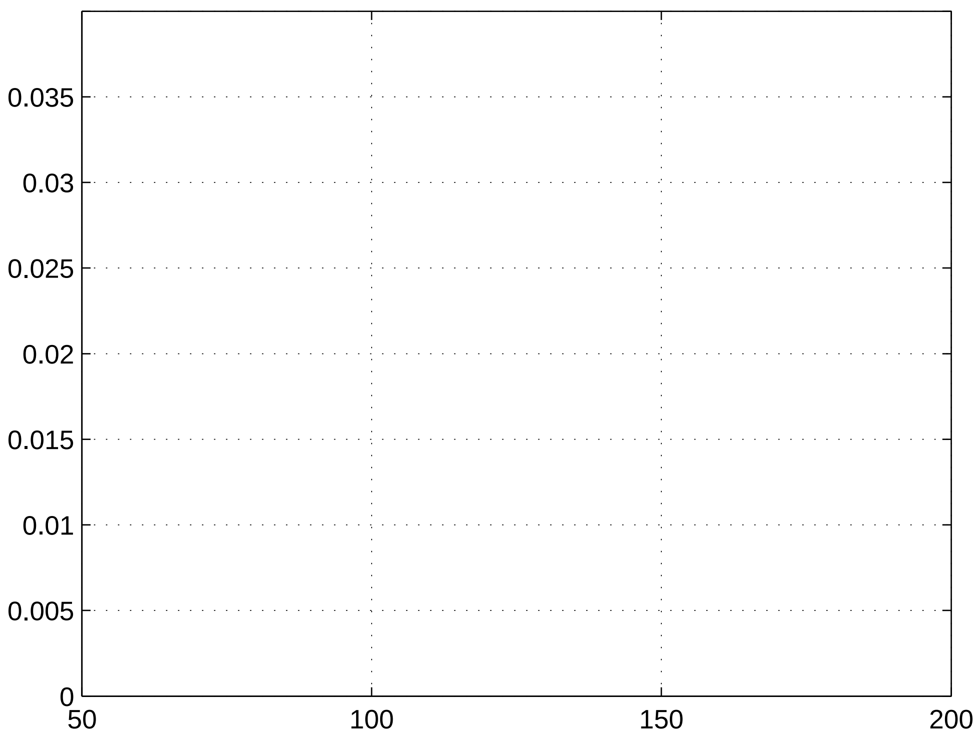


Figure 3.9: Front zoom results of the Saint-Venant equations computation using the Verlet scheme, N = 101 nodes and initial condition $h^0 = 1$ for $x = [0; 50]$ and $h^0 = \left(1 - \frac{x^2}{4}\right)^2$ for $x = [50; 100]$, $dt = 0.00001s$, $dx = 1m$. Zoom on the front wave (Computation and visualization tool MATLAB).

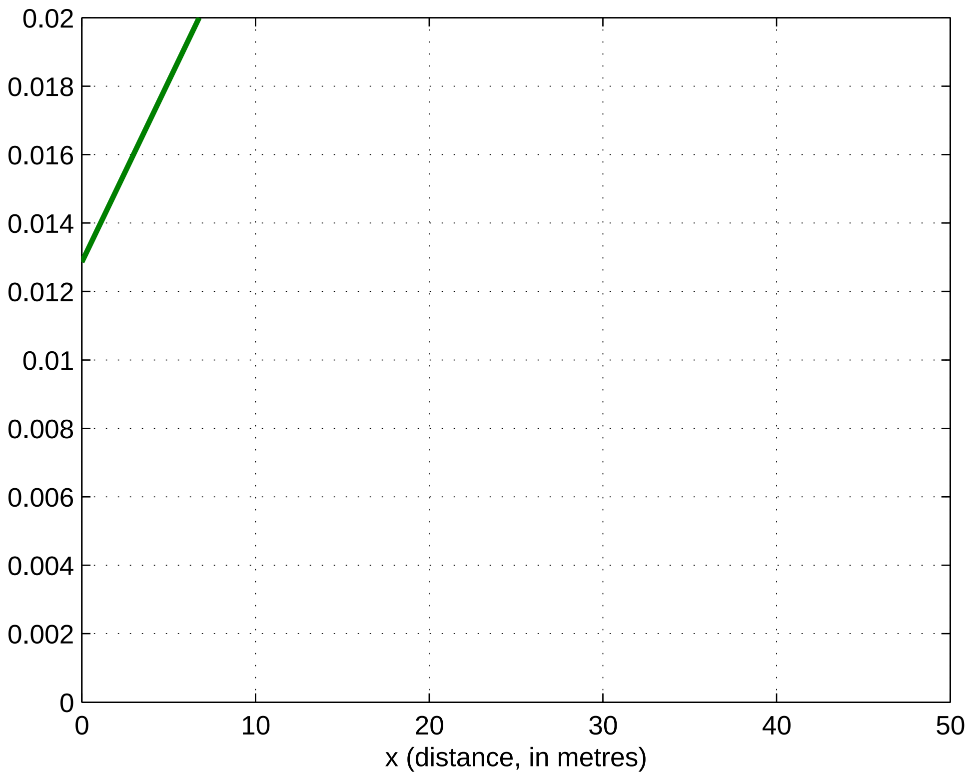


Figure 3.10: Left boundary results of the Saint-Venant equations computation using the Verlet scheme, N = 101 nodes and initial condition $h^0 = 1$ for $x = [0; 50]$ and $h^0 = \left(1 - \frac{x^2}{4}\right)^2$ for $x = [50; 100]$, $dt = 0.00001s$, $dx = 1m$. Zoom on the Left boundary (Computation and visualization tool MATLAB).

Chapter 4

Non linear diffusion: Thin film equation

The thin film equation, a non linear diffusion equation, also called the lubrication approximation, is used to model the front wave of the groundwater flood. The thin film equation is a 4th order in space 1st order in time, partial differential equation. It is used to compute the height versus spreading of a thin film of liquid on a surface over time. The equation is defined as :

$$\frac{\partial h}{\partial t} = \frac{\partial}{\partial x} \left(h^n \frac{\partial h}{\partial x} \right) \tag{4.1}$$

where $n = 3$. The solution of the self similar problem when $n = 1$ has been study before with Finite Element (Bhattacharya, MSc 2004 [8]) and Finite Difference (Baines et al., 2011 [7]) and Bird, 2012 [14]). The analytical solution for the case of $n = 3$ haven't been discovered yet, no initial condition can be surely used for that case.

4.1 Mathematical formulation

The algorithm to model the thin film equation is again based on local mass conservation.

1. Advance x the position of the film and h the height of the water using

$$\frac{dx}{dt} = h^3 h_{xxx}$$

2. Advance h using local mass conservation

$$h \quad x = \quad c$$

We use the interface boundary conditions: continuity of $h; u; h_x$ and the free boundary conditions: $h \; = \; h$

points x_2 until x_N can be determined using an explicit Euler method. We used forward Euler to compute the new nodes location

$$x_i^{n+1} = x_i^n + u^n dt$$

The height is determined using the local mass. From the initial condition using central differences, we have

$$_i = (x^0_{i+1} - x^0_{i-1})h^0_i$$

for $i = 2, \ldots, N-1$ Here, x^0 is a vector containing the initial mesh location

and h^0 is the vector containing the initial water column height.

4.3 Boundary conditions

4.3.1 Left boundary, velocity of nodal point 0

The left boundary is a flux boundary. A amount of water filled the first grid block at each time step. The mass of the local block c_0 is defined as:

$$c_0(t+1) = c_0(t) + dt \quad q(t) \tag{4.4}$$

In our case, a constant flux is used for simplification and $q(t) = 1 m^3.s^{-1}$. The left boundary is computed using symmetry principle or mirror point along the y axis defined by the equation:

$$h_0^n = \frac{c_0^n}{2 \quad x_0^n} \tag{4.5}$$

with $c_0 = cst$ computed at the time $t = 0$ at initial condition.

4.3.2 Right boundary, velocity of nodal point N: Thin film precursor

The right boundary is defined by the moving node boundary x_{N+1}, at that location the local mass is considered as $c_N = 0$ and in theory, the height of the water column is $h_N = 0$. Considering that height and the fact that h is used to compute the velocity of the node, velocity will be 0. A thin lm precursor has to be used. A thin lm precursor has to be added to the numerical model. O'Brien [6] described the precursor thin lm as a thin layer to allow the liquid to move. On a numerical point of view, the thin lm precursor is computed trough the height given at the node $x_N(t)$. If $h_N(t) = 0$, we have $v_N(t) = 0$ and the node N doesn't move as illustrated by the equation 4.6:

$$v_N(t) = \quad h_N(t)^2 \quad \frac{(q_N(t) \quad q_{N \;\; 1}(t))}{(x_N(t) \quad x_{N \;\; 1}(t))} \tag{4.6}$$

4.4 Numerical Results of the thin lm equation

The Figure 4.1 illustrates a half domain time stepping method evolution of the thin lm equation including a precursor lm at the front of $h_N(t) = 0{:}05m$. The line shows the shape of a droplet spreading over a surface at time t = $[0s; 2s; 4s; 6s; 8s; 10s; 12s; 14s; 16s; 18s; 20s]$. The time step $dt = 0{:}000001s$ for a space step of $dx = 0{:}2m$ over 2 metres long. The high of the water column is x at a maximum of $h_0(0) = 0{:}1m$. To optimize the computation velocity, the number of nodes have been chosen to $N = 11$ including a x node at $x_0(t) = 0$ and a free boundary at the node $x_N(t)$. For higher time or space step, the solution blows up. The result shows the free moving boundary node moving on the right as expected by previous author publication. The curvature of the droplet contact between water and air is getting atter as expected to assure mass conservation. The left boundary present trend $\frac{dh}{dx} = 0$. The dynamic angle as illustrated by O'Brien [6] decreases. The Figure 4.2 resumes the main features of the half thin lm equation problem.

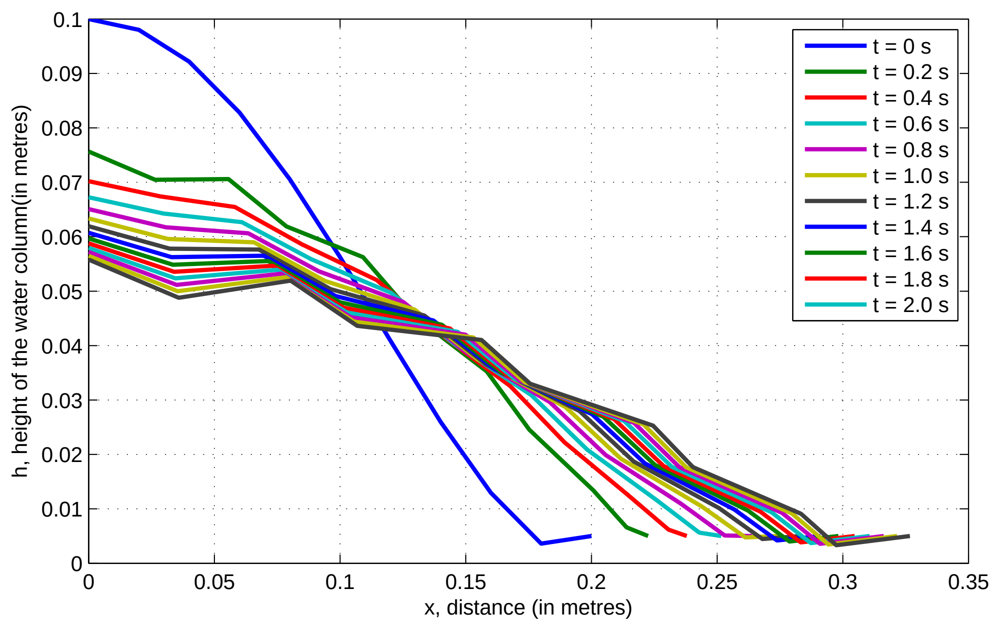


Figure 4.1: Results of the computation of the thin lm equation for N = 11 nodes.(Computation and visualization tool MATLAB)

Figure 4.2: Illustration of the main features of the thin lm script (Computation and visualization tool MATLAB).

Chapter 5

Coupling model for ood modelling

The coupling of the two numerical schemes needs careful examination of both ow domains. Time step are di erent and the interfaces point has to be identi ed. At the interface of the two ows domains, there is continuity of the water column height, velocity of the uid, the water ow, slope of the continuity of the water column height.

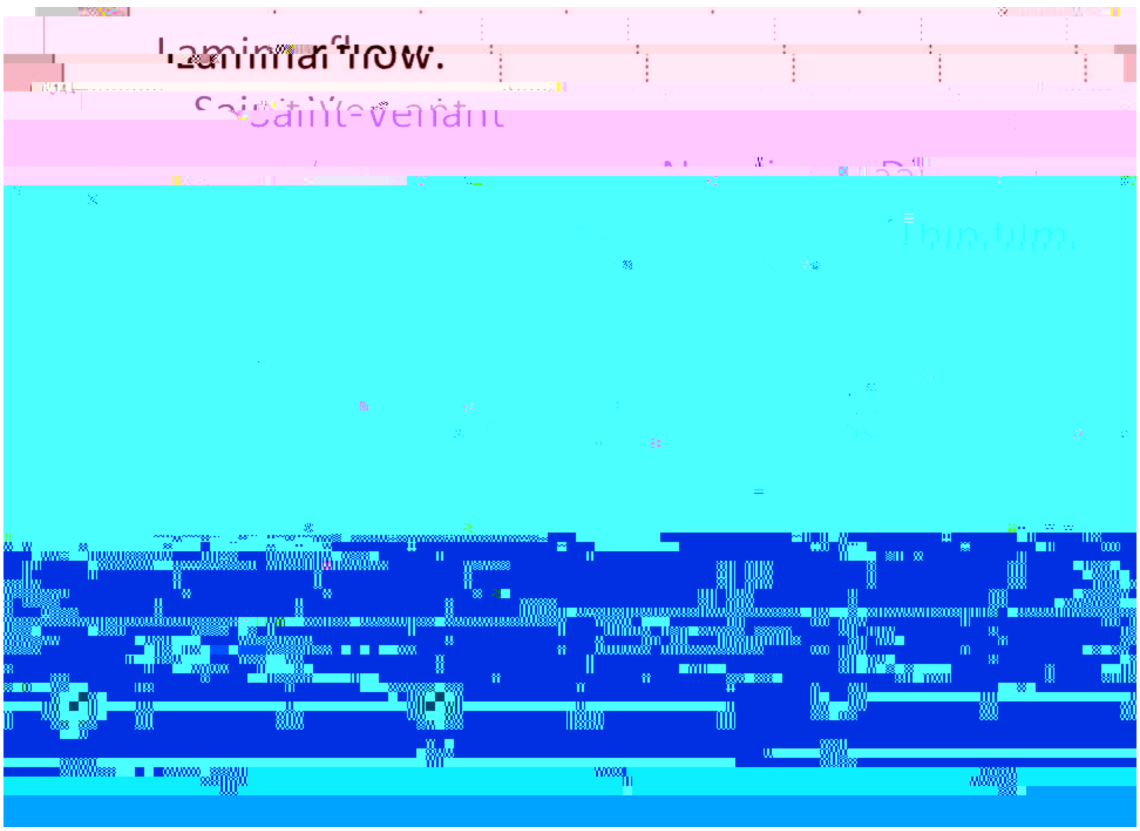
We introduce x_I, the left boundary point which is at a constant location $x_I = 0$. $x_C(t)$, the moving boundary point between the two domains and $x_F(t)$, the right boundary point at the front of the water wave. The moving mesh strategy is used for both domains through the Lagrangian frames of reference used. The Figure 5.1 resumes and illustrates the numerical problem with the main features associated. We have a 4th order equation; we need 4 boundaries conditions with 2 moving boundaries, we need also 2 more boundary conditions.

5.1 Mathematical formulation

We de ne the condtion at the node $0; C$ and N. At $x_C(t)$, we have:

$$v_{SW} = v_{TF}$$

$$h_{SW} = h_{TF}$$


Laminar flow.
Saint-Venant
Thin film.

and

$$\left.\frac{\partial h}{\partial x}\right|_{SW} = \left.\frac{\partial h}{\partial x}\right|_{TF}$$

The boundary conditions at

$$x_F(t)$$

are

$$h_{TF} = 0$$

$$\left.\frac{\partial h}{\partial x}\right|_{TF} = 0$$

The condition $h_N = 0$ gives a zero ux condition on the right boundary.

5.2 Global and local mass conservat43f81(.)]l-27(8al)2 [(=

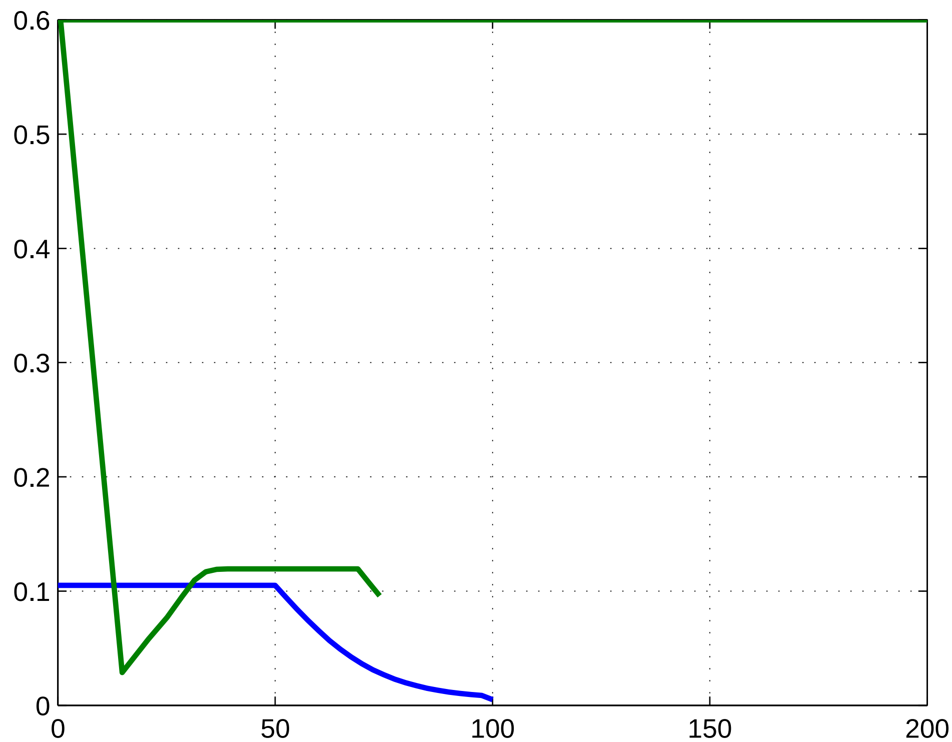


Figure 5.2: Numerical result of the coupled model at time t = 0 s, t = 20 s and t = 40 s

5.3 Numerical modelling results

Result obtained by the coupled model is illustrated Figure 5.2 and Figure 5.3. The two domains of ow are visualized on the model as well as there evolution over time. The number of node was xed to $N = 41$, the time step to $dt = 0.00005s$ and the space discretization to $dx = 2.5m$. The solution shows a problem of continuity of the left border due to the ow boundary and the change of the high due to the change of velocity. Another issue came at the point x_C where the continuity seems to be assured. Oscillation on the left border arises after $t = 200s$ and cause a solution blow up later. The solution proposes is on it draft level and will need further testing and improvement to propose a better solution even if the moving mesh of the two

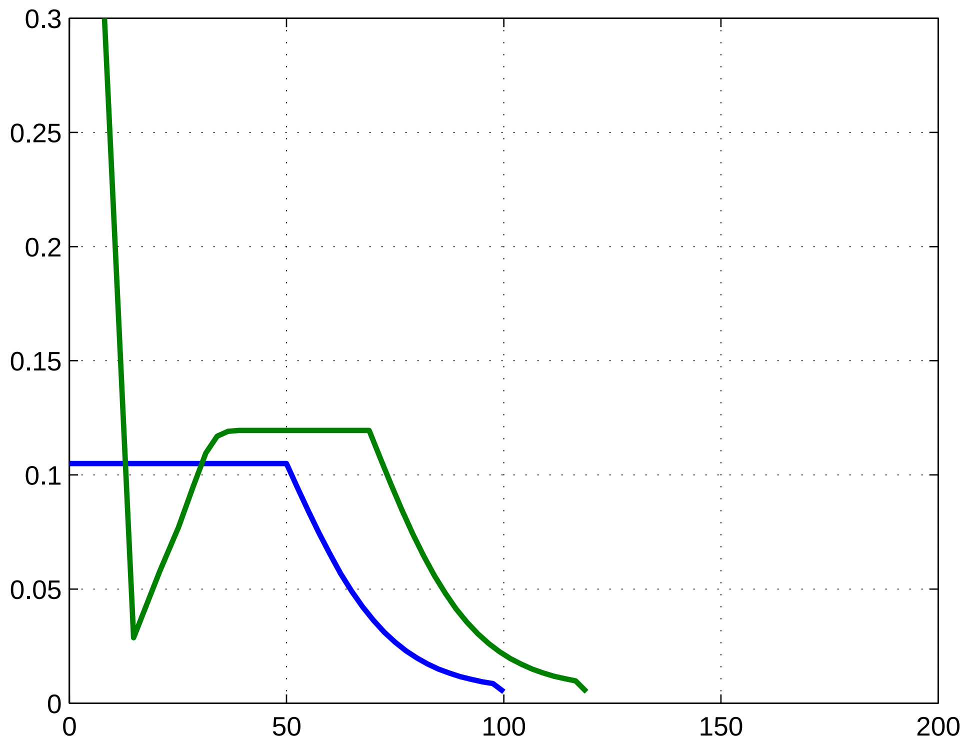


Figure 5.3: Numerical result of the coupled model at time t = 0 s, t = 20 s, t = 40 s, t = 60 s, t = 80 s, t = 100 s

ow domain is assured.

Chapter 6

Conclusions and Further Work

6.1 Conclusions

During this project, the Saint-Venant and thin film equations have been used. Several schemes were tested to optimize the resolution of the Partial Differential Equations. To solve the Saint-Venant set of equations, explicit Eulerian and Lagrangian methods were used. The central space difference was used for the Eulerian method, the scheme show some instability at it left boundary. Furthermore, a volume corrector is used to recalculate the height of the wave at the location of the node based on flux difference between the next and the previous node. Semi-Lagrangian and fully Lagrangian method were used as well. The semi-Lagrangian offers an intermediate method more accurate than the Eulerian with the simple formulation of a transformed Euler formulation. The fully Lagrangian formulation was solve using the "Verlet method" based on two half location time step computation with a height computation between the two half time step. It offers a strong stability of the scheme. The "Verlet method" is a moving mesh method; the method we used is a velocity based method. The importance of Lagrangian method is highlighted by the results. Nevertheless, the initial condition plays a strong role in the scheme stability. Even if Saint-Venant equations have analytical solution in certain situation, it doesn't apply to all configurations. We designed the initial condition based on supposition like a $1 - tanh(x)$ function. Eulerian methods are very popular because of it visualization. Lagrangian methods are more accurate but need some practice to be familiar with. The left boundary considered as a flux boundary represents the inflow of the flood

in the model. The amount of flux was fixed based on the volume of water already set by the initial condition. The flux boundary was handled by two methods, the filling of the first node by the flux and a distribution of the flux over the whole water domain using partial local mass coefficient.
The thin film equation using dynamic mesh introduces to interesting applied numerical modelling problem, it resolution is a well known problem. Method, time step and mesh discretization size as well as initial condition have strong consequence on the stability of the solution and the rise of oscillations.

Due to the high instability of some methods, some schemes have to be preferred or corrector applied like the volume corrector for the central space difference method or FTCS method (Forward Time Central Space). Nevertheless, the results obtains are similar to current publication in term of expected results. Some problem arises at the left flow boundary due to the calculation of the water high based on local mass conservation issue.

The result obtained for the coupled model need further work to improve the model and constitute only a promising draft test to model groundwater flood in a more accurate manner than actual model.

The use of a thin film to describe a flood could be seen as "strange". But coupled with Saint-Venant set of equations and for low water level between 0 and 0.1 metres, it could improve the description of the front wave in a more accurate manner for groundwater flood event.

The slope of the area was not considered as well as the roughness of the area. Those parameters will increase or decrease the velocity, the travel time and the height of the water wave. Measuring the roughness of area based on satellite image is a research subject.

6.2 Results

Results obtained during this thesis project gives direction to the development of improve flood model based on coupled physical flow model. The first draft coupled model is under development and the result constitutes only a first draft result. Limitation of the method and gap in the flow continuity are evident. The use of coupled flow domain approach to model a groundwater flow is original for multidisciplinary research purpose on hydraulic, groundwater model and advance numerical modeling. The ideas developped during this thesis will need further validation, implementation and development.

6.3 Further Work

Several direction have to be considered for further work:

1. A first improvement will be to develop the model with implicit method which offers better result in term of scheme stability and accuracy. Nevertheless, computation time will be considerably increased. In the case of the thin film equation which need small time step the computation time could became a problem.

2. A second direction will concerned the two dimensional (2D) version of the models which will have an interesting ouput on the visualization of the wave.

3. A third direction will concerned the develpment of a complex topography solving method which take in count the altitude change of the area.

4. A last direction will concern the source point spread origin of groundwater flood source: Our simple 1D model only considered a single water

[8] Bhattacharya. B., 2004 A moving nite element method for high order nonlinear di usion problems.

[9] Budd, C.J., Huang, W., Russell, R.D., 2009. Adaptivity with moving grids. Acta Numerica (2009), Cambridge University Press, 2009 pp. 1131.

[10] Weller, H., Ringler, T., Piggott, M. and Wood, N., 2009 CHALLENGES FACING ADAPTIVE MESH MODELING OF THE ATMOSPHERE AND OCEAN, Meeting summary January 2010, American Meteorological Society p. 105-109.

[11] Baines, M., Hubbard, M. E. and Jimack, P.K., A Moving Mesh Finite Element Algorithm for the Adaptive Solution of Time-Dependent Partial Di erential Equations with Moving Boundaries (with), Applied Numerical Mathematics, 54, 450-469 (2005).

[12] Price, J.F., 2006, Lagrangian and Eulerian Representations of Fluid Flow: Kinematics and the Equations of Motion, Woods Hole Oceanographic Institution, Woods Hole, MA, 02543 http://www.whoi.edu/science/PO/people/jprice June 7, 2006. Consulted the 24th July 2012.

[13] Dummer, J., A Simple Time-Corrected Verlet Integration Method, accessed June 8, 2012. http://lonesock.net/article/verlet.html.

[14] Bird, N., 2012 A Finite Di erence Scheme for Solving the Fourth Order Problem, June 1, 2012. University of Reading 9 pp. Unpublished.