Simulation of A Rapidly Varied Shallow Water Flow in Complex River Geometry

Jean Belanger[1]

[1]**Jean Bélanger** is an industrial physicist who has expertise in scientific computing and physical modeling. He can be contacted at **belanger@elligno.com.**

Contents

List of Figures

Abstract

The purpose of this thesis is an investigation on solutionning an environmental problem ... in the development of scientific software. .... study on aspects dealing with scientific software (mathematical modelling) in the environment industry. Complex case with all the difficulties present.... This thesis is made with the collaboration of the environment industry (Hydrosoft Energie Inc. and Genivar Inc.)

In the context of hydraulic we have considered the problem of the numerical simulation of a river. The case under consideration exibits torrential flows, transition to fluvial condition through an hydraulic jump and complex bed elevation. Moreover the physical problem is characterized by the presence of steep gradient. We consider a $2km$ section of the Ste-Margurite river located in the province of Quebec. This river reach was chosen for the complexity its river flows and for the quality of the field data available. We are concerned with the full non-linear de Saint-Venant equations to account for dynamical effects such as the propagation of water fronts and bores.

First, we have considered the non-conservative form of the equations. The calibration and validation step ..... To achieve this a two-dimensional finite element code (Galerkin type), AquaDyn[1] has been used to solve numerically the Saint-Venant equations. A numerical simulation of the steady flow of the Ste-Marguerite river has been carried out. We describe the different construction stages (calibration and validation of the mathematical model) of a free surface flow model. Our result are proposed has a benchhmark to test new solvers dedicated dedicated to flow conditions under adverse flow conditions (hydrology community). Result obtained show good agreement with experimental data.

Galerkin Finite Element Method give rise to central-difference type approximations of differential operators. ..... When trying to solve this numericaly with standard numerical scheme, one is faced with oscillations in the solution which destroy convergence process towards a solution (destroy the quuality of the solution). Wiggles are most likely to appear in convection dominated cases High Peclet number in the upper part, oscillations. A way to eliminate oscillations (due to the numnerical approximation) is to severely refine the mesh, such that the convection no longer dominates on an element level. We refine the mesh but still the presence of oscillations (in the variable Q).

Wiggle-free solutions could be obtained by use of "upwind" differencing on the convective term. Upwind differencing amounts to approximating the convective derivative according to the physics to be described. In this case (hyperbolic equation: wave propagation) the information propagate upstream to downstream ... with solution values at the upstream (influence and dependence zone).

To answer to this open question, we propose a more and efficient way to treat the upper part of the river with numerical scheme based on a flux extrapolation (way to remove this unphysical phenomenon is to use conservative schemes of ENO type (shock-capturing technic). We consider High-order upwind schemes .....

These schemes are best suited for this kind of flow, since it is described by hyperbolic equations ... zone of dependence and influence which is of paramount importance .. according to the physic phenomena (wave propagation).

Other difficulty are the numerical treatment of the pressure and bathymetry term. As Nujic has pointed

[1] from hydrosoft Energie Inc.

out, improper treatment of this term may lead to great inaccuracies if strong variations in the bottom topography are present. There are certain physical requirements, or compatibility conditions, which should be staisfied by any discrete model.

The adopted numerical algorithms are of paramount importance on the result.

The aim is a better understanding of flow on a very complicated bathymetry with high velocity (better understanding of the physical processus present in this kind of flow). The case of study is a very strong validation test because all the difficulties are present.

The purpose of this thesis is an investigation into the performance of a mathematical model based on conservative scheme.

The complexity of the phenomenon considered here is such that ... A 2D bidimensional .. compressible has been considered.

Résumé

L'utilisation de la simulation numérique dans plusieurs champ d'application est devenu un outil important. Cependant, l'efficacité de cette nouvelle approche est étroitement liée à la puissance des ordinateurs et fortement influencée par la validité des modèles qui sont utilisés pour la description des phénomènes et également par la précision des méthodes numériques.

Dans les écoulements à ciel ouvert la pratique commune est d'utiliser des modèles mathématiques principalement basé sur les équations de Shallow-water ou St-Venant pour simuler numériquement lesphénomènes permanent aussi bien que non-permanent. La non-linéarité des équations peut, cependant, rendre leur résolution plutôt difficile dans le sens qu'elles ne peuvent être résolues que par une approche numérique, donnant lieu à la possibilité de solutions discontinues lesquelles sont les contre-parties mathématique des phénomènes tels que le saut hydraulique et la propagation d'un front d'onde sur une surface initialement sèche. Les solutions de ce genre apparaissent habituellement quand on essaie de modéliser des écoulements permanents dans un canal à pente raide, écoulement permanent et non-permanent qui varient rapidement ou la simulation d'un bris de barrage. Dans tous les cas le traitement numérique adoptée est d'une importance capitale en ce qui concerne la qualité des résultats. Traditionnelement des méthodes aux différences finies sont utilisées pour obtenir la solution de ce système. Dûe à leur difficulté à traiter les gémétries complexes, les méthodes aux éléments finis leur sont préférés.Il est bien connu que toutes ces méthodes classiques présentent des oscillations autour des discontinuités lesquelles imposent une limite sur la précision et la convergence de la solution.

Récemment, une autre approche de modélisation appelé méthode aux volumes finis (FVM) a été proposée. Cette méthode a connu un succès considérable dans le domaine de la CFD (Computational Fluids Dynamic). Simplement on peut dire que la formulation aux volumes finis généralise les schémas aux différences finis sur un maillage arbitraire (non-structurée). La raison principale pour utiliser cette méthode est la capacité à modéliser les problèmes ayant des solutions discontinues en utilisant la théorie des équations hyperboliques, l'arsenal mathématique est déjà développé (solution du problème de Riemann). Approche naïve peut conduire à des résultats désatreux, même si ces méthods fonctionnent très bien pour des cas où les fonctions sont régulières. Dans lecas de St-Venant la présence d'une discontinuité dans la solution se fait à travers le saut hydraulique (passage d'un régime torrentiel à fluvial). Variable de la profondeur d'eau et donc de la vitesse deviennent discontinues.

Le but de cette thèse est une meilleure compréhension de la physique des écoulements à haute vitesse sur une bathymétrie non-uniforme. Ce type d'écoulement est caractérisé par les forces convectives. (Il est bien connu) que le terme convectif cause des difficultés pour obtenir un algorithme stable, généralement une évaluation "upwind" ou un schéma central avec une dissipation artificielle doit être ajoutée pour éviter les oscillations parasites. On considère un régime mixte, présence d'un saut hydraulique (choc stationnaire). Nous proposons d'utiliser des schémas aux volumes finis de haute résolution ENO ("Essentially Non-Oscillatory") pour répondre à ces questions. Ces schémas ont comme propriété de résoudre de façon précise les discontinuités et possèdent très peu de dispersion numérique. Ceci est un avantage dans le présent contexte, la conservation de la masse, un point crucial pour la présente application, est naturellement imposée parce que ces schémas sont conservatifs de manière inhérente. De plus les bonnes relations de continuités sont

imposées au choc. Le travail se fera en collaboration avec les compagnies: Hydrosoft Energie Inc., Genivar Inc., Elligno Inc. et le Centre de Recherche en Calcul Appliqué (CERCA).

Dans le cadre de cette thèse, des méthodes numériques récentes ont été appliquées à la modélisation de problèmes physiques de la dynamique des fluides. Le problème physique considéré est caractérisé par la présence de forts gradients .... La simulation numérique de tels phénomènes physiques a été réalisés grâce à des schémas dits "de capture de chocs", adaptés au traitement de discontinuités. Dans ce contexte, nous avons essayé d'évaluer la précision que l'on pouvait attendre de l'application de ces techniques à des cas de calculs ayant un intérêt physique réel. Plus spécifiquement des problèmes relevant de l'hydraulique ont été abordés.

Dans le cadre de l'hydraulique nous avons considéré le problème de la simulation numérique d'une rivière. Le but de telles simulations est la compréhension des processus physiques qui apparaissent pour les écoulements à haute vitesse sur une bathymétrie non-uniforme avec transition d'un régime torrentiel vers un régime fluvial (ressaut hydraulique qui est vu comme un choc stationnaire, vitesse nulle). Ce genre d'écoulement est caratérisé par les forces convectives et peut être décrit par des équations hyperboliques des lois de conservation. Ce type de problème trouve son intérêt dans l'industrie de l'hydro- électricité. À titre d'exemple, le cas du détournement d'une rivière, on veut être en mesure d'évaluer l'impact sur la variation du débit de celle-ci, ce qui peut avoir une influence sur l'exploitation et les coûts de production de l'électricité.

D'un point de vue de la mécanique des fluides la propagation d'un front d'onde est un phénomène extrêmement compliqué impliquant la dynamique du fluide avec une surface libre dans un mouvement turbulent intense sous l'accélération de la gravité. Quand on essaie de décrire mathématiquement cette situation, on doit résoudre les équations de Navier-Stokes 3-D à surface libre. Cependant, sous certaines approximations (St-Venant) on arrive à une représentation mathématique plus simple de la réalité physique et qui trouve de nombreuses applications dans le domaine de l'ingénierie. On obtient donc un ensemble d'équations couplées qui sont un équilibre entre les forces de la pesanteur, de frottement et d'inertie appliquée à la masse d'eau en mouvement. Ces équations décrivent des bilans entre les différents processus: diffusion, convection (transport). Selon que l'un de ces processus dominent sur les autres ceci change la nature physique de l'équation. Par exemple, et c'est le cas qui nous intéresse, quand la convection domine on a affaire à un phénomène de propagation.

Les équations de St-Venant peuvent être décrites mathématiquement par différents ensembles d'équations différentielles à dérivées partielles. Il est bien connu que la forme conservative est préférée à celle non-conservative si des changements brusques (ou discontinuités) dans la solution sont anticipés, ou si des régimes d'écoulements mixtes sont présents à l'intérieur du domaine de calcul. La formulation non-conservative introduit des termes de sources numériques (sources internes). Pour des écoulements continus ces termes sont du même ordre de grandeur que l'erreur de troncature et peuvent être négligeables [25]. Cependant, les expériences numérique et les comparaisons montrent que les formulations non-conservative sont moins précises que celles conservatives, particulièrement en présence de forts gradients.Pour un écoulement discontinu, tel qu'un écoulement transonique avec une onde de choc, ces termes de source numérique peuvent devenir important à travers la discontinuité et donne lieu à de larges erreurs. La forme non-conservative ne donnera pas les bonnes intensités pour le choc. Ainsi pour obtenir dans le calcul numérique les bonnes relations aux discontinuités (relations de Rankine-Hugoniot pour les équations de Euler) il a été démontré par Lax(1954) qu'il est nécessaire de discrétiser la forme conservative des équations [25].

Historiquement les méthodes de capture de choc peuvent être classifiées en deux catégories: principalement les méthodes de capture de choc moderne et classique. La stabilité du calcul en présence d'ondes de choc exige qu'une quantité de dissipation numérique soit ajoutée, en particulier pour éviter les oscillations numériques ("spurious oscillations") lesquelles imposent une limite sur la précision et empêche la convergence vers une solution. Dans le cas des méthodes de capture de choc classique les termes de dissipation numérique sont habituellement linéaire et la même quantité est appliquée à tous les points de grille. Ces

méthodes sont appelés "upwind" à cause des nombreuses variantes que l'on rencontre. Elles introduisent la direction de propagation de l'onde dans la discrétisation numérique. Les méthodes de capture de choc modernes vont plus loin dans l'introduction de la physique dans le schéma numérique. Ces schémas ont été initialement développés par Godunov et généralisés par Roe et Osher("Approximate Riemann Solver").

Ces schémas ont des propriétés intéressantes:

1. ils sont automatiquement conservatifs pour les équations homogènes (dans le cas de St-Venant on s'assure que la masse est conservée) parce qu'ils sont l'équivalents discrets de la forme conservative des équations. Ils permettent de bien capturer les chocs avec la bonne valeur pour la vitesse du choc,

2. ils sont monotones parce qu'ils traitent chaque onde non-linéaire selon sa nature (compression et rarefaction),

3. les propriétés physiques de l'écoulement sont incorporés dans le schéma numérique ("upwind built"), ainsi on évalue les dérivées selon la direction de propagation de l'onde.

De plus ces solvers contiennent assez d'information physique pour détecter les chocs et la transition entre les régimes d'écoulement par l'intermédiaire des valeurs propres de la matrice Jacobienne du flux F. Ces schémas ont prouvé qu'ils étaient stables et précis en présence d'ondes de choc.

Pour une rivière, les écoulements ne sont pas toujours regulier et continus, ils peuvent varier beaucoup en espace et en temps, tel que l'apparition d'une onde de choc dans le cas d'un bris de barrage, transition entre un régime torrentiel et fluvial (ressaut hydraulique). Les problèmes d'écoulements qui impliquent des changements abruptes en espace et en temps tombe dans la catégorie d'un problème de Riemann. Ainsi un schéma discret conservatif est nécessaire en solutionnant un problème de Riemann. Pour la solution approximer du problème de Riemann, l'approche utilisée habituellement est de généraliser le solver exact de Riemann pour le cas linéaire au cas non-linéaire. Les solvers approximés de Riemann inclus la méthode "flux-vector splitting" (FVS) [Steger et Warming (1981); Van Leer (1982)], la méthode "flux-difference splitting" (FDS) [Roe (1981); Glaister (1988)], et le schéma de Osher [Osher (1982); Spekreijse (1988)].

La caractéristique principale (attrayante) de ce modèle est qu'il calcule le flux de masse et de quantité de mouvement à travers chacun des côtés des éléments comme un problème de Riemann, lequel est résolu en utilisant un "Approximate Riemann Solver". Cette caractéristique fait que ce modèle est capable de manipuler les processus couvrant-découvrant, bris de barrage impliquant des écoulements discontinus, des écoulements sous-critique et super-critique. ———

Les principaux points de difficultés quand on considère les écoulements sur une topologie complexe (non-uniforme) sont les suivants:

- pour des régimes "steady-state" il est difficile de garder le débit constant, ceci est causé par deux raisons: premièrement, diffusion numérique, solution de la variable h qui représente la profondeur d'eau contient des variations abruptes, deuxièmement, incompatibilité entre le terme de pente et le terme de pression de la fonction flux $\mathbf{f}$ [38],

- traitement numérique du terme source de bathymétrie, traitement inadéquat de ce terme peut conduire à de larges erreurs si des variations brusques sont présentes dans le terme de bathymétrie [38]

- présence de régions sèches, dans certains cas on besoin d'ajouter une mince couche d'eau (concept de la profondeur minimum), modèle utilisé en éléments finis. Problèmes: 1) comment choisir la valeur pour h_{min}? 2)conservation de la masse n'est pas assurée [32]

- oscillations dans la solution numérique

- modélisation du terme de convection

Pour construire un modèle numérique qui soit valide pour un très grand nombre de situations physiques rencontrés, il est essentiel d'intégrer la représentation discrète de chaque terme avec un degré de précision et de stabilité acceptable.

Le mémoire de cette thèse est organisé en cinq chapitres où les différents aspects précédent introduits sont développés.

Le chapitre 1 fait un rappel sur l'approximation hydrodynamique. On fait un rappel sur les idées et les principes fondamentaux de la théorie mathématique des lois de conservation et des systèmes hyperboliques. Un problème de dynamique des fluides peut être formulé comme un ensemble de relations qui décrivent des processus physiques dans lesquels certaines quantités sont conservées. Par exemple, dans le cas de la dynamique des gaz on considère la conservation de la masse, de la quantité de mouvement et de l'énergie. Ces équations plus d'autres relations d'origines empiriques forment le cadre naturel dans lequel on peut modéliser la simulation de phénomènes physiques très différents. Dans ce travail, nous avons considéré la simulation d'une rivière, ce domaine est un exemple parmi tous les secteurs possibles de recherche où les phńomènes d'intérêt physique (coût de production, prdiction à long terme,...) ... reproduction en laboratoire et pour lesquels une approche du type numérique constitue un moyen d'investigation est crucial.

Le chapitre (?) introduit la discrétisation numérique utilisées. Par conséquent, la discrétisation spatiale est basée sur une approche volumes finis conservative de la formulation intégrale des équations. La motivation principale pour utiliser une discrétisation conservative est qu'elle permet de capturer correctement un choc (impose les bonnes relations de continuité Rankine-Hugoniot), dans notre cas au saut hydraulique qui est un choc stationnaire (vitesse nulle).

Le chapitre (?) présente les schémas de capture de choc de haute résolution. Une bonne méthode de capture de choc doit produire une approximation non-oscillante et ne pas trop être dissipative pour assurer un bon niveau de précision là où la solution cherchée est régulière. Il existe principalement deux classes de schémas de capture de choc pour la discrétisation du terme d'advection. La première classe consiste en des schémas dits "à variation totale décroissante" (TVD), où le flux numérique du deuxième ordre est limité par une correction du première ordre qui introduit le niveau de dissipation nécessaire à assurer la condition de stabilité. La seconde, celle utilisée dans cette thèse, est celle qui a été proposée par Godunov, dont on considère aussi les généralisations d'ordre élevé par des méthodes de reconstruction polynômiale.

La haute résolution du schéma est obtenue en utilisant une approximation ENO (Essentially Non-Oscillatory) pour les flux numériques. Les schémas ENO sont basés sur des stencils adaptatifs qui sélectionnent automatiquement les valeurs d'interpolation dans la région environnante où la fonction est régulière. On évite d'utiliser des données qui proviennent de la région où la fonction subit des variations brusques ou discontinuité. Ces schémas ont comme propriété de résoudre de façon précise les discontinuités et possèdent très peu de dispersion numérique. La conservation de la masse, un point crucial pour la présente application, est naturellement imposée parce que ces schémas sont conservatifs de amnière inhérente. Finalement , une discrétisation numérique "upwind" est utilisée pour modéliser le plus fidèlement possible le mécanisme physique de propagation ondulatoire contenu dans la partie convective, en accord avec la nature physique de l'écoulement. Dans ce contexte, nous voulons évaluer la précision que l'on peut attendre de l'application de ces techniques à des cas de calculs ayant un intérêt physique réel.

La discrétisation des termes visqueux est effectué par un schńa centré. La solution est intgrée en temps en utilisant un schńa de Runge-Kutta explicite au deuxième ordre. Le code implémente aussi certaines techniques standards pour accélérer la convergence comme des techniques multigrille, quand la solution cherchée est stationnaire.

Le travail efectué dans cette thèse a donné lieu aux publications suivantes:

- "A Test Field Calibration to Validate Shallow-Water Codes: the Case of the Ste-Marguerite River with *AquaDyn*" J. Bélanger, M. Carreau and A. Vincent, CERCA technical report, no. R00-6

- "On the Construction of Numerical Schemes Modelling Flood Wave Propagation over Complex Topography" J. Bélanger, E. McNeil and A. Vincent, Elligno-Genivar Technical Report, no. EG00-1

Chapter 1

State Of The Art

1.1 Introduction

Considerable progress has been made in Computational Fluid Dynamic (CFD) propagation over the last few years and nowhere has this progress been more evident than in area of flood wave propagation (open channel flow). ..... However the capability to handle complex topography and dry beds has been paid most attention.

This section discusses the principles and architecture of current .. systems and identifies the key issues affecting their future development. To illustrate the various points raised, the ... is described. This system is a modern design that gives state-of-the-art performance, and it is typical of the current generation of scientific software modelling open channel flows.

1.2 Fluid Mechanics Review

1.3 Numerical Approach

1.4 Current Issues

1.5 Conclusion

The role of computers in physics is growing in importance and there seems little doubt that this will accelerate. The vitality of computational physics stems from the inexorable advances in computer technology that will lead, by the end of the year, to supercomputers with sustained performance of teraflop (10^12) floating point operations per second. This teraflop supercomputer, and its associated gigaflop workstation on your desk, will enhance the role of simulation as a third approach to physics, augmenting the traditional experimental and theoretical methodologies.

Computer simulation is now an integral part of contemporary basic and applied science and is approaching a role equal in importance to the traditional experimental and theoretical approaches. Hence, the ability "to compute" is part of the essential repertoire of research scientists and industrial application. The definition of a computational approach involves several steps leading from an initial mathematical model to a final numerical solution. The first step, is the *level of approximation* to the physical to be solved, dependent on the accuracy required as much as on the computational power available.

One of the more conspicuous properties of nature is the great diversity of size or length scales in the

structure of the world. An ocean, for example, has currents that persist for thousands of kilometers and has tides of global extent; it also has waves that range in size from less than a centimeter to several meters; at much finer resolution seawater must be regarded as an aggregate of molecules whose characteristic scale of length is roughly 10^{-8} centimeter. From the smallest structure to the largest is a span of some 17 orders magnitude.

In general, events distinguished by a great disparity in size have a little influence on one another; they do not communicate, and so the phenomena associated with each scale can be treated independently. The interaction of two adjacent water molecules is much the same whether the molecules are in the Pacific Ocean or in a teapot. What is equally important, an ocean wave can be described quite accurately as a disturbance in a continuous fluid, ignoring entirely the molecular structure of the liquid. The success of almost all practical theories in physics depends on isolating some limited range of length scales.

Physicists propose various levels of description of our physical world, ranging from subatomic, atomic or molecular, microcospic and macroscopic up to the astronomical scale. The statistical description of a gas, motion of the individual atoms or molecules are taken into consideration and their behaviour is basicallyruled by Boltzmann equation. Then we can define temperature as a measure of the mean kinetic energy of the gas molecules; pressure as a result of the impulse of molecules on the walls of the body containing the gas; viscosity connected to the momentum exchange due to the thermal molecular motion, and so on. At this microscopic level of description the fundamental variables are molecule velocities, number of particles per volume and other variables defining the motion of the individual molecules, while pressure, temperature and viscosity are mean properties which are deduced from other variables. We can continue with other level of description and so on, that not our intention here. The mere existence of a fluid implies that a sufficiently high number of associated atoms form molecules in order to obtain a minimal amount of interaction between these molecules.

Fluid dynamics starts to exist as soon as the interaction between a sufficiently high number of particles affects and dominates, at least partly, the motion of each individual particle. Hence fluid dynamics is essentially the study of the interactive motion and behaviour of a large number of individual elements. ... the interaction between the particles becomes negligible if the mean-free path length attains a magnitude of the order of the length scale of the system considered. From this stage on and for higher values of the mean-free path length, the particles behave essentially as individual elements.

In conclusion, we can say that fluid mechanics is essentially the study of the behaviour of averaged quantities and properties of a large number of interacting elements.

The second step is the model used to describe the the flow. From the fluid mechanics point of view flood propagation is an extremely complicated phenomenon involving the dynamics of a fluid with a free boundary in intense turbulent motion under the acceleration of gravity. When trying to describe mathematically this situation, one is faced with solving a full three dimensional unsteady Navier-Stokes problem with a free boundary. Under certain assumptions (St-Venant approximation), one may resort to a simpler mathematical representation of the physical reality. This is certainly the case in many engineering applications.

In open-channel flow the common practice is to use mathematical model principally based on the Shallow-water or the St-Venant equation to simulate (numerically) steady as well as non-steady phenomenon. The non-linearity of the governing equations may, however, render their resolution rather complicated in the sense that they can only be solved numerically. In all of these cases the numerical treatment adopted is of a paramount importance on the precision of the numerical results.

The third step is the choice of the *discretization method* of the mathematical formulation and involves two components, the *space discretization* and the *equation discretization*. The space discretization consists of setting up a mesh or a grid by which the continuum of space is replaced by a finite number of points where the numerical values of the variables will have to be determined. It is intuitively obvious that the accuracy of a numerical approximation will be directly dependent on the size of the mesh, that is, the better

the discretized space approaches the continuum, the better the approximation of the numerical scheme. In other words, the error of the numerical simulation has to tend to zero when the mesh size tends to zero, and the rapidity of this variation will be characterized by the *order* of the numerical discretization of the equations.

Once the mesh has been defined the equations can be discretized, leading to the transformation of the differential or integral equations to discrete algebraic operations involving the values of the unknowns at the mesh points. The basis of all numerical methods consists of this transformation of the physical equations into an algebraic, linear or non-linear, system of equations.

The use of numerical simulation is becoming of a paramount importance in many engineering application. However, the efficiency of this new approach is related by the power available on computer and by the precision of the numerical model to describe the physical phenomena.

Until recently numerical methods for solving fluid-flow problem have been dominated by finite difference approximation. This technic is the oldest of the methods applied to obtain numerical solutions of the differential equations. The philosophy of finite difference solutions is to replace the partial derivatives appearing in the equations with algebraic difference quotients, yielding algebraic equations for the flow field variables at the specified grid points.

The finite element approach is one of the most popular method in numerical simulation in the last few decade. THe original "finite element" concept replaces the continuum by a number of subdomains (or elements) whose behavior is modelled adequatetly by a limited number of degrees of freedom and which is assembled by process well known in analysis of numerical methods. The capability to manage complex geometry using high order interpolation polynomial, make this class of approximation very attractable.

Traditionaly finite difference were used to obtain solution of this system, because of their difficulty to treat complex geometry, finite element method was preferred. Recently, another modelling approach called finite volume method (FVM) has been proposed. The finite volume takes the merit of both finite difference and finite element as follows: In some sense, the FVM can be considered as a finite difference method applied to the differential conservative form the conservation laws, written in arbitrary coordinates. Hence this method can be applied using any unstructured grid system as in FEM.

The FVM is based on the integral form of the conservation equations. A scheme in conservative form can be easily constructed. As the time and space approach to zero, the scheme is consistant with the differential equation of conservation laws. By discretization of the integral form of the conservation equations, we can ensure that the basic quantities, mass and momentum, will also remain conserved [25]. In the discretization of the Shallow-water equation in non-conservative form, an additional numerical source term is introduced. For continuous flow simulation, this additional numerical source termhas the same order of magnitude as the truncation error and it is usually negligible. However, for discontinuous flows, this term can become significantly large across discontinuity and can give rise to large errors [25]. In river bassins, the flows are not always smooth and continuous in that they could vary a lot in space, such as the flows at the transition between the main channel and the flood plain. Also, they could abruptly change in time, such as the occurence of shock wave when opening sluice gates or in a dam break. Flow problem involving abrupt changes of flows in time and space fall into the framework of a Riemann problem. Thus a conservative discrete scheme by solving a Riemann problem is needed. For the approximate solution of the Riemann problem the approach usually is to generalize the exact Riemann solver of the linear case to the non-linear case. The approximate Riemann solvers include the flux-vector splitting method (Steger and Warming) and the flux-difference splitting method (Roe, Glaister) and the Osher scheme.

Simulation of hydrologic systems using numerical methods has evolved within a decade from an interesting curiosity to an accepted and established engineering and scientific approach to the study of water resource problems.

The use of mathematical models in the simulation of diverse hydraulic phenomena has become essential

as a predictive tool in the evaluation of proposed engineering works. Until recently, the engineer was forced to revert to physical modelling in order to simulate, and thereby it was hoped predict, the impact of hydraulic work on the existing river system and its environment. Mathematical models that calculate the velocity field are therefore of fundamental importance in modern hydraulic engineering practice.

The need for reliable evaluation tools of flood risk potential is nowadays of paramont both in developed and developing countries. Mathematical modelling can be used for development planning and improvement, in order to avoid future high risk situations.

For applications in civil engineering, there are several complicating factors: turbulence, geometrical description, a lack of data to define parameters, problem size and numerical difficulties with the methods of calculation. Specially, the interaction between the main channel flow and the floodplain flow which requires a full 2D model that should also be able to simulate the drying and wetting processes in the floodplain, in the main channel (for the submergence of an island) and in the wetland. Ideally, the numerical model for river-bassin simulation should possesses the following features:

- ability to handle complex topography
- simulation of subcritical and supercritical flow
- simulation of steady and unsteady flow
- simulation of smooth flow and discointinuous flow
- ability to handle wetting and drying of floodplain

To construct a numerical model valid over a broad range of commonly encountered physical situations, it is essential to integrate the discrete representation of each term with an acceptable degree of accuracy and stability. Recent development in numerical analysis for hyperbolic equations have led to new methods which solve Shallow-water equations including complementary equations for hydraulic jumps.

.... Solutions of this kind usually appear when modelling steady flows in steeply sloping channels (rapidly varied steady). In this case the adopted numerical treatment is of paramount importance inb what concerns the quality of the results, often with a shock-capturing method of second order of accuracy being required.

.... finite difference and element It is well known that all these classical methods present oscillations around discontinuities, which degrade the accuracy of the numerical solution and are difficult to remove if a general method of resolution is sought.

During the last decade much effort has been devoted to the numerical solution of systems of conservation laws, mainly driven by the need for efficient Euler solvers in aerodynamics. This had led to a new class of methods that do not suffer from the known of classical ones [25] and to a generation of techniques able to improve the performance of conservative second-order classical schemes. It is the main purpose of this thesis to report on the implementation of these techniques.

In river modelling, supercritical flows and hyhdraulic jumps often occur, particularly in the of high flood flows. At present in such situations, either empirical formulae are used at the part of the river where supercritical flow appear, or large amounts of artificial viscosity are imposed to damp the resulting numerical oscillations. The inclusion of non-physical artificial viscosity significantly reduces the accuracy of the solution in areas with high velocity gradients. In some cases, even very high artificial viscosity cannot stabilize the model. As much of the numerical stability theory is based on linear equations, implicit schemes, which are believed to be 'unconditionnaly' stable, may becomes unstable in transcritical flow conditions.

For shock wave problems, which are the aerodynamic equivalent of a 'bore' in free surface hydraulics, computational fluid dynamics (CFD) has made enormous progress. Lax and Wendroff stressed the importance of models based on the conservation laws.

A Godunov scheme requires a Riemann solver to solve local Riemann problem at the cell interfaces. The Godunov scheme approximate the solution at each interface by a piecewise-constant extension of the] cell-averaged values assigned in each cell to the centroid. To asses the evolution of the piecewise discontinuous data, appeal is made, in gas dynamics, to the Riemann solution of the model shock tube problem with discontinuous piecewise constant initial data. Many approximate Riemann solvers have been developed which are more efficient than the exact Riemann solver originally proposed by Godunov.

Following such a progress in aerospace CFD, these methods have been gradually introduced into hydraulics. Indeed, classical tests for supercritical flows and discontinuous have been conducted using Roe type approximate Riemann solver for the Saint-Venant equations and successfully introduced Godunov-type methods to solve the dam-break problem. Other approximate solvers have been implemented for Shallow-water equations and these Godunov-type methods have produced very good results for dam-break problems. As a Godunov-type scheme requires the evaluation of the numerical fluxes at cell interfaces, a finite volume methods seems to be particularly and naturally appropriate for its implementation.

Mathematical simulation of hydraulic phenomena is becoming increasingly important in engineering practice since it offers the possibility of cheaply evaluating the response of hydraulic systems to a variety of practical situations. Among these, rapidly varied open channel flow possesses certain features that make it important to predict but most difficult to compute. Supercritical flows with hydraulic jumps and bore are ackward to represent even if a shock-capturing method is employed.

The estuarian and river flow modelling is higly complex and varying in nature. The relative importance of bottom friction, convective and local accelerations, gravitational and turbulent stresses may vary rapidly from one region to the other in a given flow. Moreover, the spatial boundaries may move significantly under the temporal influence of tidal waves, storm surges, rapid discharge variations, ebbing and flooding. During a given period (e.g. tidal cycle) a large part of the river or estuary bed may change from wet to dry and vice-versa. A numerical model should be able to correctly take into account such variations in order to obtain the realistic flow patterns essential for hydraulic and biological impact studies.

Real Case Application

The Ste-Marguerite River, Quebec an application is presented for flow simulations of a portion of the Ste-Marguerite river in Quebec. The model results have been compared with the data from a physical model study conducted by Groupe Lasalle in Montreal and also with field data from the Ste-Marguerite river collected by Hydro-Quebec environmental department.

The flow is caracterized by a torrential flow (hydraulic jump) and by fluvial flow ..... convective forces are dominating This is the principal difficulties noticed in a flow over a complex topography as the one considered in this thesis:

1. for a steady state regime it is difficult to keep the discharge rate constant, there are two factors responsible: firts numerical diffusion.... and incompatibility between the slope term and the pressure term (see [38]).

2. numerical treatment of the source term (bathymetry), inpropper treatment of this tyerm can lead to large errors if the variation

3. modelisation of the convection term

4. numerical oscillation

This thesis introduces two major techniques for the solution of completely supercritical, steady viscuous flows, the ... Indeed, all of computational fluid dynamics is more an art than a science - maybe this is why the field is so exciting and growing so fast in popularity.

1.6 Open Channel Flow

The depth-averaged St-Venant equations describe the motion of water with a free surface and are obtained by integration of the three- dimensional Navier-Stokes equations over the depth, with the assumption of hydrostatic vertical pressure distribution, *i.e.* negligible vertical accelerations. The need for efficient and accurate numerical schemes to solve these equations is at present of primary concern to computational hydraulics.

1.7 Numerical Approach

Practical example: For the physical phenomenon (description) i don't need to know the individual behaviour of each molecule. For example, if we want to describe the movement of air around aile d'avion, a molecular approach will insensed, besause this implies to calculate the movement of billions of molecules. We need a more glopbal approach, to define macroscopic quantities (another scale or level). Question: fluid dynamics start where, ... gas become a continuum. This implies we can define macroscopic property to describe the gas (global movement of the molecules and then define mean properties at macroscopic scale. We call that hydrodynamic approximation. To compute these properties we need algebraic equations (differential or integral equations which relate these properties to each other with appropriate boundary condition. These mathematical equations are deduced from the fundamental principles of physics (conservation laws).

Integral Form

The governing equations of a compressible flow can be formulated as a system of conservation laws, in which case u represents a set of variables involved in some fundamental principles of physics. For a flow, the system ofconservation laws express the conservation of mass, momentum and energy, so the conserved quantities are the density, the momentum per unit mass and the total energy per unit mass. There are other additional global conservation laws that can be deduced under special assumptions like the conservation of entropy for inviscid continuous flows, the conservation of kinetic energy for incompressible inviscid flows and vorticity for plane in compressible flow. It is well known that the solution of a conservation law may "break down" after a finite time, if its first derivative blows up, even if the initial solution is smooth. A conservation law expressed by an integral relation maybe satisfied by solutions not continuous or differentiable. Requiring only that the solution be measurable and bounded, a generalized solution can be easily defined. The main motivation for using the integral conservative formulation is to allow discontinuous solution and to automatically capture them numerically. The conservative formulation of equation (2.2) has a direct correspondant in numerical discretizations, leading to the so-called "conservative" schemes. Lax and Wendroff showed that conservative discretizations are the natural framework to correctly compute discontinuous solutions of conservation laws.

Chapter 2

St-Venant Model

2.1 Introduction

The St-Venant model has been shown to provide a satisfactory description of a wide range of physical flow with a free surface ranging from river and channel flows to estuarine circulation and floods due to dam or dike failure.

Traditionally, most phenomena or processes of interest have been of physical caracter. Fluid dynamics, a branch of continuum mechanics, has for centuries been a rich source of mathematical models, which take the form of Ordinary Differential Equations, Partial Differential Equations and Integral Equations. For compressible fluids, such as gas or a liquid at very high pressures, such mathematical models result from the formal statement of three basic physical principles, namely the principles of conservation of mass, momentum and energy.

Navier-Stokes equations

In the field of Computational Fluid Dynamics (CFD) the Navier-Stokes Equations (NSE) are know to fully describe the motion of incompressible viscous fluid. These equations describe balance between forces acting on control volume (hydrodynamic approximation). We have three types of] forces:

- **Body Forces** (F_g). These are forces that act on the entire control volume. We assume this is gravity only, so $F_g = \rho g$, ρ is the density and g is the gravitational force ($9.81 m/s^2$)
- **Pressure Forces** (F_p). These forces act inwards and normal to the control volume (surface). It is defined as the negative of the gradient of the pressure field of the control volume, *i.e.* $F_p = -\vec{\nabla} P$
- **Viscous Forces** (F_V) These are forces due to friction in he control volume and acts in all directions on all control volume. Given the fact that the fluid is Newtonian, *i.e.* a fluid where the stress is linearly proportional to the strain, the net viscous force (F_v) pe unit volume is defined as: $F_v = \mu \triangle V$, where $\mu = \frac{1}{\rho V L}$ with ρ the density, V the velocity and L is the dimension.

Now that we have all the forces acting in fluids, we will use Newton's law ($F = m\ a$) to describe the motion:

$$F_g + F_p + F_v = \rho A$$

where $\rho A = \rho\ g + -\vec{\nabla}\ P + \mu \triangle\ V$. Now assuming uniform density we can write the equation as:

$$A = g - \frac{1}{\rho}\vec{\nabla}\ P + \frac{\mu}{\rho}\triangle V$$

The conservation of the momentum is given by:

$$\frac{d\vec{u}}{dt} = -\frac{1}{\rho}\vec{\nabla}P + \nu\triangle\vec{u} + \vec{f} \tag{2.1}$$

where

$$\frac{d\vec{u}}{dt} = \frac{\partial\vec{u}}{\partial t} + (\vec{u}\cdot\vec{\nabla})\vec{u}$$

which is call "material derivative[1]". This equation conserves the momentum. In addition we need the mass to be conserved

$$\vec{\nabla}\cdot V = 0 \tag{2.2}$$

which express that the flux variation inside the volume is conserved.

These two equations together are refered as the NSE. Unfortunately the NSE is a set of highly non-linear Partial Differential Equations (PDEs) that's not easily solved.

2.2 St-Venant approximations

This is the hypothesis formulated by Barré Sainy-Venant in .... The equations for an open channel flow are given by the St-Venant. Hypothesis on which the St-Venant equations are based see [13] [42]

1. the flow is unidimensional, this mean that the velocity is uniform in the wetted cross-section and the transversal slope of the free-surface is zero (constant) La composante verticale de la vitesse est négligée comparativement aux composantes parallèles de la vitesse à la surface, la vitesse verticale peut être vue comme une constante sur la profondeur.

2. the curvature of the streamlines is small, so we can neglect the vertical acceleration (hydrostatic pressure) :

   $$\frac{dp}{dz} = -\rho g, \quad p - p_0 = (z-\xi)\rho g \tag{2.3}$$

   at the free-surface $z = \xi$ and $p = p_0$, where p_0 is a constant pressure (reference),

3. it is also supposed that the friction force can be approximated by the formula used in steady uniform flow (Manning formula),

4. bottom slope is very small, we can then replace the cosinus of the angle between the thalweg and the horizontal by unity (variations of the channel width along the axis are very smooth

5. water density is constant (incompressible flows) which can be expressed by material derivative (Lagrangian)

   $$\frac{D\rho}{Dt} = 0 \tag{2.4}$$

   where

   $$\frac{D}{Dt} = \frac{\partial}{\partial t} + u\frac{\partial}{\partial x} + v\frac{\partial}{\partial y} + w\frac{\partial}{\partial z} \tag{2.5}$$

In these conditions the flow can be described by two dependent variables, say A (wetted cross-section area) and Q (discharge). It is clear that we can define other group of dependent variables e.g. u (fluid velocity) and h (water depth). These two variables define the behavior of the fluid along the channel in terms of the independent variables: the distance (x) and the time (t).

[1] Take account

2.3 Approaches

Finite control volume approach Control Volume Approach: we consider a space domain (volume V) with a surface S. This volume can be fix or can move with a speed (it always contains the same amount of particles inside). Then we apply the law of conservation to this volume of control (fix or in movement), the integral form of the equations are obtained

Infinitesimal fluid element approach Infinitesimal Element δV: in this approach we consider an infinitesimal element of fluid δV (in a differential sens). It can be fix or move with a speed v (flow speed) along a streamline. We should note that δV contains a huge amont of molecules that we can consider the milieu as a continuum. We apply the conservation principles to the element of fluid to obtain the differential form

Molecular approach Molecular Approach: apply the laws of nature directly to each molecule then we define statistical average. The kinetic thoery, the equation of Boltzmann.

We apply these physical principles to the appropriate flow model. Then we obtain the mathematical equations which represent these principles.

2.4 General Formulation of Conservation Laws

In the different fields of physics (electromagnetism, nuclear, field theory ...) equations representing the law of nature are obtained from conservation principles (conservation laws). What do we mean by a conservation theorem? First, from a mathematical statement it mean that we can express the equations of physics as a time rate of some conserved quantities in term of flux density of these quantities (flow rate through some volume in space). The general of a conservation law is given by (takes on the form of a set of divergence conditions):

$$\frac{\partial T_{\mu\nu}}{\partial x_\nu} = 0 \tag{2.6}$$

where the $T_{\mu\nu}$ is a tensor representing, for example, in electromagnetism the components electric and electromagnetic fields.

.... writing the set of equations above as

$$\frac{\partial T_{\mu 0}}{\partial t} + \frac{\partial T_{\mu j}}{\partial x_j} = 0 \tag{2.7}$$

or

$$\partial T_{\mu 0} + \vec{\nabla} \cdot T_\mu = 0 \tag{2.8}$$

where T_μ are a set of space vector. In either form, Eqs.() or Eqs.() appear as equations of continuity, which say that the time rate of change of some density plus the divergence of some corresponding flux or current density vanishes. In turn, the equations of continuity imply the conservation of some integral quantities providing the field volume is finite *i.e.* the field can be contained within a volume beyond which the field quantities are zero. Define, in such case, integral R_μ by

$$R_\mu = \int_\Omega T_{\mu 0}\, d\Omega \tag{2.9}$$

where the volume integral extends beyond the region containing the field. Then, by Eqs.(),

$$\frac{dR_\mu}{dt} = \int \nabla \cdot \mathbf{T}_\mu = \int \mathbf{T}_\mu \cdot d\mathbf{A} = 0 \tag{2.10}$$

It is because of these conservation theorems, derived from Eq.(), that] the $T_{\mu\nu}$, are known in the parlance of modern physics as *conserved currents*.

EXAMPLE IN NUCLEAR PHYSICS:

It can be shown that the electric current density **j** is given by the equation of continuity for charge and current as follow:

$$\vec{\nabla} \cdot \vec{j} = -\frac{\partial \rho}{\partial t}. \tag{2.11}$$

Examples of conservation equations may be easily multiplied.

Now lets apply these principles in fluid mechanics. Consider a quantity U inside a volume Ω. The local variation U with respect to time inside the volume is given by the following expression

$$\frac{d}{dt}\int_{\Omega} U \; d\Omega \tag{2.12}$$

which is equal to the flux contribution going through the surface Γ of the volume

$$-\int_{\Gamma} \vec{F} \cdot \vec{d\Gamma} \tag{2.13}$$

where the surface element $d\Gamma$ pointing outward, the origin of the minus sign. To this we can add sources of the quantity U (surface sources Q_s volume sources inside the volume Q_v)

$$\int_{\Gamma} \vec{Q}_s \cdot \vec{d\Gamma} + \int_{\Omega} Q_V \; d\Omega \tag{2.14}$$

The general form for the conservation equation for a scalar quantity U is

$$\frac{\partial}{\partial t}\int_{\Omega} U \; d\Omega + \int_{\Gamma} \vec{F} \cdot \vec{d\Gamma} = \int_{\Omega} Q_v d\Omega + \int_{\Gamma} \vec{Q}_s \cdot \vec{d\Gamma} \tag{2.15}$$

This last integral equation can be re-written by using the Gauss theorem (we suppose that the fluxes are continuous functions ... the same apply to the surface sources). Then we obtain

$$\int_{\Omega} \frac{\partial U}{\partial t} \; d\Omega + \int_{\Gamma} \vec{\nabla} \cdot \vec{F} \; d\Omega = \int_{\Omega} Q_v \; d\Omega + \int_{\Omega} \; \vec{\nabla} \cdot \vec{Q}_s d\Omega \tag{2.16}$$

Since this expression is valid for an arbitrary volume, we obtain the differential form of the conservation law

$$\frac{\partial U}{\partial t} + \vec{\nabla} \cdot \vec{F} = Q_v + \vec{\nabla} \cdot \vec{Q}_s. \tag{2.17}$$

It is important to remark that in the absence of sources the variation of U inside the volume Ω depend only on the flux at surface.

Here u: $R \times R \to R^m$ is a m-dimensional vector of conserved quantities, or state variables. More properly, u_j is the density function for the j(th) state variable with the interpretation that $\int_{x_1}^{x_2} u_j(x,t)dx$ is the total quantity of this state variable in the volume Ω.

The functions u_j themselves, representing the spatial distribution of the state variables, at time t, will generally change as time evolves. The flux of the j(th) components is given by some function $f_j(u(x,t))$. The vector valued function $f(u)$ with j(th) component $f_j(u)$ is called **flux function** for the system of conservation laws, so $f : R^m \to R^m$. The derivation of the equation of Shallow-Water from physical principles will be illustrated below.

Conservative form of Shallow-water 1D The study of wave motion in shallow water leads to a system of conservation laws. To derive the one-dimensional equations, we consider fluid in a channel and

assume that the vertical velocity of the fluid is negligible and the horizontal velocity $v(x,t)$ is rouhly constant through any vertical cross section. This is true if we consider small amplitude waves in a fluid that is shallow relative to the wave lenght.

We now assume the fluid is incompressible, so the density ρ is constant. Instead the height $h(x,t)$ varies, and so the total mass $[x_1, x_2]$ at time t is

$$\text{total mass in } [x_1, x_2] = \int_{x_1}^{x_2} \rho h(x,t)\ dx \tag{2.18}$$

The momentum at each point is $\rho v(x,t)$ and is integrating vertically gives the mass flux to be $\rho v(x,t)h(x,t)$. More generally any quantity z which is advected with the flow has a contribution to the flux z of the form zv, where v is the velocity of the flow. The constant ρ drops out of the conservation of the mass equation, which then takes the familiar form

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(vh) = 0 \tag{2.19}$$

The conservation of momentum equation also takes the same form as in the Euler equations,

$$\frac{\partial}{\partial t}(\rho hv) + \frac{\partial}{\partial x}(\rho hv^2 + p) = 0 \tag{2.20}$$

but now the pressure p is determined for hydrostatic law, stating that the pressure at depth y is ρgy, where g is the gravitational constant. Integrating this vertically from $y = 0$ to $y = h(x,t)$ gives the total pressure felt at (x,t), the proper pressure term in the momentum flux

$$p = \frac{1}{2}\rho g h^2 \tag{2.21}$$

Using this in the previous equation and cancelling ρ gives

$$\frac{\partial}{\partial t}(\rho hv) + \frac{\partial}{\partial x}(\rho vh^2 + \frac{1}{2}gh^2) = 0 \tag{2.22}$$

This formula and (?) are conservative form of the Shallow-water waves equations in one dimension for rectangular cross-section.

Le flux convectif F_c, attaché à la quantité U dans un écoulement de vitesse $\vec{v}$, est la quantité U transportée par (ou avec) le mouvement et est donné par

$$\vec{F_c} = \vec{v}U. \tag{2.23}$$

le flux diffusif qui est la contribution dûe au mouvement moléculaire, agitation thermique que l'on exprime de façon générale par la loi de Fick

$$\vec{F_d} = -\kappa\ \rho\ \vec{\nabla} u \tag{2.24}$$

où u est la quantité de U par unité de masse *i.e.* $U = \rho\ u$, ρ est la masse spécifique du fluide et κ est la constante de diffusivité.

$$\frac{\partial(\rho\ u)}{\partial t} + \vec{\nabla} \cdot (\rho\ \vec{v}\ u) = \vec{\nabla} \cdot (\kappa\ \rho\ \vec{\nabla} u) + Q_v + \vec{\nabla} \cdot \vec{Q}_s. \tag{2.25}$$

Cette équation est la forme générale d'une équation de transport pour la quantité $U = \rho\ u$. Notons que les dimensions de la constante de diffusivité sont m^2/s pour n'importe quelle quantité U.

Maintenant si U est une quantité vectorielle la forme de l'équation pour la loi de conservation demeure la même seule le caractère des variables changent. La composante du flux convectif est donnée par le produit tensoriel

$$\overline{\overline{F}} = \vec{v} \otimes \vec{U} \tag{2.26}$$

où $\otimes$ signifie le produit tensoriel des vecteurs $\vec{v}$ et $\vec{U}$. Dans la notation tensorielle

$$F_{C_{ij}} = v_i \; U_j \tag{2.27}$$

In two dimension 2D. Les équations de continuité (4) nous suggère que T_i doit représenter un flux de densité vectoriel pour la ième composante de la densité du champ. Nous assignons une propriétété vectorielle à T_i parce qu'il peut y avoir, par exemple, un écoulement dans la direction y de la composante x de la densité de la variable (e.g. quantité de mouvement) et mesuré par T_{xy}. Une interprétation alternative à T_{ij} nous vient de la déformation d'un solide élastique. Il est bien connu que mise à part les forces de compression normales à la surface il y a aussi des forces de cisaillement, le long de l'élément de surface. La force totale peut être écrite en disant que la force $\mathrm{d}\vec{F}$ agissant sur un élément de surface $\mathrm{d}\vec{A}$ est exprimée en termes d'un tenseur de stress $\vec{T}$ tel que

$$d\vec{F} = \vec{T} \cdot d\vec{A}. \tag{2.28}$$

Alors la force totale dans la direction x, sur un élément de volume rectangulaire $dx \; dy \; dz$ a une contribution des forces sur les surfaces dans les plans yz donné par

$$[T_{11}(x + dx) - T_{11}(x)] \; dy \; dz = \frac{dT_{11}}{dx} \; dx \; dy \; dz, \tag{2.29}$$

mais il y a aussi une contribution des surfaces dans le plan xz;

$$[T_{12}(x + dx) - T_{12}(x)] \; dx \; dz = \frac{dT_{12}}{dy} \; dx \; dy \; dz, \tag{2.30}$$

et similairement pour le plan xy. Les équations de Newton ici correspondent au taux de changement par rapport au temps de la densité de quantité de mouvement dans la direction x, $-T_{10}$, est égal à la composante x de la force sir l'élément de volume unité:

$$-\frac{dT_{10}}{dt} = \frac{dT_{11}}{dx} + \frac{dT_{12}}{dy} + \frac{dT_{13}}{dz}, \tag{2.31}$$

laquelle est précisément la composante x de l'équation (4). Pour ce champ particulier T_{ij} représente les composantes d'un tenseur de stress à trois dimensions, d'où l'origine du nom "stress tensor" pour $T_{\mu\nu}$.

REMARQUE: la référence pour les notes ci-haut est le livre de Goldstein "Classical Mechanics", le chapitre 12 (section 12-3).

Le terme de divergence quand il est intégré sur le volume devient un terme de surface (théorème de Gauss)

$$\int_V \vec{\nabla} \cdot T_{\mu\nu} \; dV = \int_S T_{\mu\nu} \cdot \vec{dS} \tag{2.32}$$

Ainsi les contributions externes, par analogie avec la mécanique, sont des forces qui agissent sur la surface du volume. Pour une direction particulière (courant ou flux) de cette et des sources.

En électromagnétisme la quantité $T_{\mu\nu}$ est le tenseur électromagnétique (contient les composantes du champ électrique et magnétique). Voir le chapitre 12 du livre de Goldstein les sections 12-4, 12-5.

2.5 Conservative form

In this section we derive the equations of Saint-Venant for an arbitrary cross-section. .... to be completed. These equations are obtained from physical principles and by the Saint-Venant assumptions which has been formulated at the beginning of this chapter.

Since two variables are sufficient to describe the behavior of the fluid in a section, we need only two equations to describe unsteady flow. These two equations must represent two physics law, principally the mass and momentum conservation respectively. In the next section we obtain the integral relation of these two physical laws.

2.5.1 Integral Relations

PHYSICAL PRINCIPLE: **Mass can be neither created nor destroyed**

Let us apply this principle to the model of a fixed control volume.

Continuity Equation

Let us consider the control volume in the plane (x,t) between two sections $x = x_1$ and $x = x_2$ and in the time interval $t = t_1$ and $t = t_2$. we will obtain the mass and the momentum conservation equations for thios control volume from the Saint-Venant hypothesis. The net mass flow into the control volume through the entire surface is equal to the time integral of the difference between the discharge at x_1 $(\rho uA)_{x_1}$ and x_2 $(\rho uA)_{x_2}$

$$\int_{t_1}^{t_2} [(\rho uA)_{x_1} - (\rho uA)_{x_2}]dt \tag{2.33}$$

Note: the product $\rho u$is called *mass flux* i.e. the flow of mass per unit area per unit time[2]. This quantity must be equal to the variation of the mass inside (accumulated) between x_1 and x_2 in the time interval $t_1 - t_2$. The total mass indside the control volume $\oint \rho A dx$, the time rate of change of this mass inside the control volume is therefore

$$\int_{x_1}^{x_2} [(\rho A)_{t_2} - (\rho A)_{t_1}]dx \tag{2.34}$$

where $= \rho$ is the density and $u(x,t)$ flow velocity (uniform in the section) and $A(x,t)$ wetted cross-section. If $Q = uA$ (section discharge), the integral relation whhich express the continuity

$$\int_{x_1}^{x_2} [(A)_{t_2} - (A)_{t_1}]dx + \int_{t_1}^{t_2} [(Q)_{x_1} - (Q)_{x_2}]dt = 0 \tag{2.35}$$

Finally, the physical principle that mass is conserved (given at the beginning of this section) states that the net mass flow into the control volume must equal the rate of increase of mass inside the control volume. In terms of the integrals given above, a mathematical statement is simply the equation above. This equation is called the *continuity equation* it is the integral formulation of the conservation of mass principle to a fluid flow.

PHYSICAL PRINCIPLE:

The time rate of change of momentum of a body equals the net force exerted on it

Momentum equation

Written in a vector form, the above statement becomes

$$\frac{d}{dt} m\vec{v} = \vec{F} \qquad (1)$$

For constant mass, Eq.(? above) yields

$$\vec{F} = m\frac{d\vec{v}}{dt} = m\vec{a}$$

which is teh more familiar form of the Newton's second law, that force = mass × acceleration. However, the above principle with Eq.(1) is a general statement of Newton;'s second law than Eq.(? just above). In this section, we wish to put Newton's second law Eq.(first) on fluid mechanic terms by employing the same control volume utilized in the previous section and sketched in fig.().

The conservation of momentum in the direction of the ψ thwalweg require the equality between the variation of this quantity inside the control volume in the time interval $[t_1, t_2]$ and the sum of the net inflow

[2]Whenever you see a product of (density × velocity) in fluid machanics it can always be interpreted as mass flow per unit area perpendicular to the velocity vector

of momentum inside the volume plus the integral of external forces actintg in the direction ψ during the time interval $[t_1, t_2]$.

Now consider the left-hand side of Eq.(1). In terms of our fluid dynamic model, how is the time rate of change of momentum, $m\frac{d\vec{v}}{dt}$ expressed? To answer this question, again use some physical intuition. Look at the control volume in fig.(). Because it is fixed in space, mass flows into the control volume fromm the left (x_2) at the same time that other mass is streaming out toward the right. The mass flowing in brings with it a certain momentum at the same time, the mass flowing out also has momentum. With this picture in mind, let M_f represented the net rate of flow momentum across surface. The momentum is the product of the mass times the velocity and its flux for the section is the product of the mass discharge times the velocity:

$$\text{momentum flux} = \rho u A \times u = \rho u^2 A \tag{2.36}$$

Momentum inflow through section x_1 during an infinitesimal time interval dt will be given by

$$\left[\underbrace{\overbrace{\rho(uA)}^{\text{mass flux}} \times u}_{\text{momentum flux}} \times dt \right]_{x_1} \tag{2.37}$$

The net rate of flow of momentum going through the control volume at a given time is $(\rho u A^2)_{x_1} - (\rho u A^2)_{x_2}$. Therefore, the net inflow for the time interval $t_2 - t_1$

$$M_f = \int_{t_1}^{t_2} \left[(\rho u A^2)_{x_1} - (\rho u A^2)_{x_2} \right] \, dt \tag{2.38}$$

At this stage, it would be tempting to claims that M_f represents left-hand side of Eq.(1 Newton second law). However, consider an *unsteady* flow, where by definition the flow properties at any given point in flowfield are functions of time. Example, would be the flow over a body that is oscillating back and forth in time. If our control volume in fig.() were drawn in such an unsteady flow, then the momentum of teh fluid inside the control volume would fluctuating with time. Therefore M_f does not represent the whole contribution to the left-hand side of Eq.(1). There is, in addition, a time rate of change of momentum due to unsteady, transient effects in the flowfield inside Ω. Let ΔM represent thios fluctuation in momentum. On infinitesimal lenght dx the momentum at a given time t_1, will be

$$\left[\underbrace{(\rho u A)}_{\text{mass}} u \right]_{t_1} \tag{2.39}$$

Between the two sections, this quantity will be given

$$\int_{x_1}^{x_2} (\rho u A)_{t_1} dx \tag{2.40}$$

The increase of this quantity between t_1 and t_2 is

$$\Delta M = \int_{x_1}^{x_2} \left[(\rho u A)_{t_2} - (\rho u A)_{t_1} \right] \, dt \tag{2.41}$$

First, consider the forces on the control volume using some intuitive physical sense, we can visualize these forces as two types:

1. *Body forces* acting on the fluid inside Ω. These forces stem from "action at a distance", such as gravitational and electromagnetic force that may be exerted on the fluid inside Ω due to force fields acting through space.

2. *Surface force* acting on the boundary of the control volume. Surface forces in a fluid stem from two sources: pressure and shear stress distribution over the surface. Since we are dealing with inviscid flows here, the only surface force is therefore due to pressure.

Pressure Force

The pressure force F_{p1} is the difference between F'_{p1} and F''_{p1} applied respectively at x_1 and x_2. From the hydrostatic pressure[3] hypothesis, for a section x (cf. Fig.(?))

$$F'_{p1} = g \int_0^{h(x)} \rho \left[h(x) - \eta\right] \ \sigma(x, \eta) \ d\eta \tag{2.42}$$

where

η	= depth integration variable along y axis
$h(x,t)$	= water depth
$\sigma(x,t)$	= width for a fixed depth
$b(x,t)$	= width at free surface

The integral of F_{p1} between t_1 and t_2 will be given by:

$$\int_{t_1}^{t_2} F_{p1} dt = \int_{t_1}^{t_2} \left(F'_{p1} - F''_{p1}\right) dt = g \int_{t_1}^{t_2} \left[\left(gI_1\right)_{x_1} - \left(gI_1\right)_{x_2}\right] dt \tag{2.43}$$

where

$$I_1 = \int_0^{h(x)} \left[h(x) - \eta\right] \sigma(x, \eta) d\eta$$

If the channel width vary along dx, the longitudinal component of pressure force F_{p2} will increase, and this increase is represented by the increase of the wetted free surface $d\sigma d\eta$ for a given constant value of $h = h_0$, the increase multiply by the distance between its gravity center and the free surface $h(x) - \eta$:

$$\rho g \left[\underbrace{(\frac{\partial \sigma}{\partial x}) dx}_{d\sigma} \, d\eta\right]_{h=h_0} \qquad \left[h(x) - \eta\right] \tag{2.44}$$

This force must be integrate between $\eta = 0$ and $\eta = h(x)$ for a given section and from x_1 to x_2 to obtain the total force F_{p2} acting on the control volume at a given instant. The contour integral of the control volume for a time interval $t_2 - t_1$ is

$$\int_{t_1}^{t_2} \int_{x_1}^{x_2} \rho g \int_0^{h(x)} \left[h(x) - \eta\right] \left[\frac{\partial \sigma(x, \eta)}{\partial x}\right]_{h_0} d\eta \ dx \ dt \tag{2.45}$$

Define:

$$\int_{t_1}^{t_2} F_{p2} dt = g \int_{x_1}^{x_2} \rho I_2 dx dt \tag{2.46}$$

[3] $\frac{dp}{dz} = -\rho g \quad p = \rho g(h - z) + p_0; p_0$ pressure reference

where

$$I_2 = \int_0^{h(x)} (h(x) - \eta) \left[\frac{\partial \sigma(x,\eta)}{\partial x} \right]_{h_0} d\eta$$

NOTE: Eq.(above this one) is not valid for rapid changement of the section. Others forces acts and must take into account. Moreover, the streamline does not have anymore weak curvature (St-Venant hypothesis).

Gravity Force

This body force is equal to the product of its mass and force by unit mass, namely, $(\rho dv)f$. Hence this force is acting verticaly $P = \rho g A\ dx$. Its projection in the direction ψ of the thalweg will be: $f_g = Psin\alpha$. If we suppose that the bottom slope $S_0 = -\frac{\partial y_b}{\partial x} = tan\alpha$ is small, such as $sin\alpha \approx \tan\alpha$:

$$\int_{t_1}^{t_2} F_g dt = \int_{t_1}^{t_2} \int_{x_1}^{x_2} \rho g A S_0 dx\ dt \tag{2.47}$$

Friction Force

The resistance force due to friction is applied to the control volume for shear stress on the wall. The shear on a unity lenght of the channel is $\rho g A S_f$, where S_f is the 'friction slope' *i.e.* the gradient of energy line needed to resist the friction resistance in steady flow. The integral of the resistance force in time will be:

$$\int_{t_1}^{t_2} F_f dt = \int_{t_1}^{t_2} \int_{x_1}^{x_2} \rho g A S_f dx\ dt \tag{2.48}$$

Finally, the sum $M_f + \Delta M$ represents the total instataneous time rate of change of momentum of the fluid as it flows through the control volume. This is the fluid counterpart of teh left-hand side of (1), *i.e.*

$$\frac{d}{dt}(m\vec{v}) = M_f + \Delta M$$

Therefore, to repeat the physical principle stated at the beginning of this section, the time rate of change of momentum of the fluid that is flowing through the control volume at any instant is equal to the net force exerted on the fluid inside theh volume. In turn, these words can be directly translated into an equation. The conservation of momentum along the axis can be written

$$\Delta M + M_f = \int_{t_1}^{t_2} \Big[(F_{p1} + F_{p2}) cos\alpha - F_f + F_g \Big] dt \tag{2.49}$$

where $cos\alpha \approx 1$ (St-Venant Hypothesis) and constant density (ρ).

$$\begin{aligned} \int_{x_1}^{x_2} \Big[(uA)_{t_2} - (uA)_{t_1} \Big] dx &= \int_{t_1}^{t_2} \Big[(u^2 A)_{x_1} - (u^2 A)_{x_2} \Big] + g \int_{t_1}^{t_2} \Big((I_1)_{x_1} - (I_1)_{x_2} \Big) dt \\ &+ g \int_{t_1}^{t_2} \int_{x_1}^{x_2} I_2 dx\ dt + g \int_{t_1}^{t_2} \int_{x_1}^{x_2} A(S_0 - S_f) dx\ dt \end{aligned} \tag{2.50}$$

Integral form of momentum equation. It is the integral formulation of Newton's second law applied to inviscid fluid flows. Note that this equation does not include the effects of friction. If friction were to be included, it would appear as an additional force namely shear and normal viscous stresses integrated over the control surface.

The continuity equation and the momentum equation, despite their complicated looking integral forms, are powerful tools for analysis and understanding of fluid flows. For a study of incompressible flow, the continuity and momentum equations are sufficient tools to do the job. These equations govern the mechanical aspects of such flows.

For compressible flow, the principle of the conservation of energy must be considered in addition to the continuity and momentum equations. The energy equation is where the thermodynamics enters the game of compressible flow.

In conclusion, equation(?) is the integral form of the conservation law of momentum for unsteady 1D flow for a channel with arbitrary section. The second equation, also based on thge St-Venant hypothesis, is the law of mass conservation. These equations has been established without imposing any special conditions on the functions $A, Q, u, \ldots$. These variables cannot be differentiable. If one of these is discontinuous, the integral relations still valid if only if St-Venant hypothesis still valid.

This is the conservative form of the St-Venant equations:

$$\partial_t \vec{U} + \vec{\nabla} \cdot \vec{F}(\vec{U}) = \vec{H} \tag{2.51}$$

where $\vec{U}$ is the state variables vector, $\vec{F}$ flux of the state variables $\vec{H}$ represent source terms. If we write $\vec{F}$ in term cartesian components $\vec{E}, \vec{G}$ on peut écrire la dernière éqaution sous la forme suivante

$$\partial_t \vec{U} + \partial_x \vec{E} + \partial_y \vec{G} = \vec{H} \tag{2.52}$$

Les vecteurs $\vec{U}, \vec{E}, \vec{G}$ peuvent être exprimés en termes des variables de l'écoulement comme

$$\vec{U} = \begin{pmatrix} h \\ hu \\ hv \end{pmatrix} \qquad \vec{E} = \begin{pmatrix} hu \\ hu^2 + gh^2/2 \\ huv \end{pmatrix} \qquad \vec{G} = \begin{pmatrix} hv \\ huv \\ hv^2 + gh^2/2 \end{pmatrix} \tag{2.53}$$

where g is the gravitational acceleration. The source is given by

$$\vec{H} = \begin{pmatrix} 0 \\ gh(S_{0x} - S_{fx}) \\ gh(S_{0y} - S_{fy}) \end{pmatrix} \tag{2.54}$$

with S_{0x}, S_{0y} representent bed slope in both cartesisan direction, S_{fx}, S_{fy} friction terms which can computed from the Manning formula.

2.6 Summary

Conservative form The Saint-Venant equations can be written in a vectorial form as follow:

$$\frac{\partial}{\partial t}\left[\int_\Omega U \; d\Omega\right] + \int \vec{\nabla} \cdot \left[\vec{F} - \vec{G}\right] \; d\Omega = \int_\Omega S \; d\Omega \tag{2.55}$$

where

$$U = [h, uh, vh]^T = \left[h, \vec{V}h\right]^T \tag{2.56}$$

$$F = \left[\vec{V}, \vec{V} \otimes \vec{V}h + g\frac{h^2}{2} \cdot \vec{I}\right]^T = \begin{pmatrix} uh & vh \\ u^2h + g\frac{h^2}{2} & uvh \\ vuh & v^2h + g\frac{h^2}{2} \end{pmatrix} \tag{2.57}$$

which includes the convection tensor

$$\vec{V} \otimes \vec{V}h = \begin{pmatrix} u\,(uh) & u\,(vh) \\ v\,(uh) & v\,(vh) \end{pmatrix} \tag{2.58}$$

and the pressure term

$$g\frac{h^2}{2}\vec{I} = \begin{pmatrix} g\frac{h^2}{2} & 0 \\ 0 & g\frac{h^2}{2} \end{pmatrix} \tag{2.59}$$

G represents the turbulence stress effects

$$G = \left[0, K_D\ \vec{\nabla}\left(\vec{V}h\right)\right]^T = \begin{pmatrix} 0 & 0 \\ K_D\frac{\partial(uh)}{\partial x} & K_D\frac{\partial(uh)}{\partial y} \\ K_D\frac{\partial(vh)}{\partial x} & K_D\frac{\partial(vh)}{\partial y} \end{pmatrix} \tag{2.60}$$

and the source term

$$S = \left[0, -gh\left(\vec{\nabla}Z + C_f\vec{V}\right)\right]^T = \begin{pmatrix} 0 \\ -gh\ \left(\frac{\partial Z}{\partial x}\right) + C_f u \\ -gh\ \left(\frac{\partial Z}{\partial y}\right) + C_f u \end{pmatrix} \tag{2.61}$$

In the vector equation (?), the first component expresses the conservation matter, while the remaining two components express the conservation of momentum in the x and y directions respectively. Here $h(x, y, t)$ represents the instantaneous water depth while $Vh = [uh, vh]$ is the unit discharge vector with u and v being the respective velocity components. The source terms contains the bed slope, the Coriolis effect with parameter f, and friction terms due to shear stress at the bed, this being expressed with empirical laws.

Chapter 3

Numerical Methods of Conservation Laws

3.1 Introduction to computational fluid dynamics

As we have seen from the previous chapter, the cornerstone of theoretical fluid dynamics is a set of conservation equations which describe the physics of fluid motion; these equations speak words, such as:

- Mass is conserved
- $\mathbf{F} = m\mathbf{a}$ (Newton's second law)
- Energy is conserved

These equations also describe the variations of fluid pressure, temperature, density, velocity, etc., throughout space and time. In their most general form, they are integral equations or partial differential equations, and consequently are difficult to solve. Indeed, no general analytical solution to these equations has been found, nor is it likely to be found in the foreseeable future. For the two centuries since Bernouilli and Euler first formulated some of these equations in St Petersburg, Russia in 1730s, fluid dynamicists have been laboring to obtain analytical solutions for certain restricted and/or simplified problems.

In contrast, the modern engineer of today is operating in a new third dimension in fluid dynamics-*computational fluid dynamics*, which readily complements the previous dimensions of pure experiment and pure theory. Computational fluid dynamics, in principle, allows the practical solution of the exact governing equations for a myriad of applied engineering problems, and it is this aspect that is introduced in this chapter.

What is computational fluid dynamics? It is the art of replacing the governing nonlinear partial differentila equations with numbers, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flowfield of interest. The end product of computational fluid dynamics is indeed a collection of numbers, in contrast to a closed-form analytical solution. However, it can be argued in the long run the objective of most engineering analyses, using closed-form equations or otherwise, is a quantitative description of the probllem *i.e.* numbers.

La question qu'on est en droit de se poser, qu'est-ce qu'une méthode numérique? considérons une équation aux dérivées partielles notées symboliquement

$$L(\vec{u}, \vec{x}, t) = 0 \tag{3.1}$$

où $\vec{x}$, t sont les variables indépendantes d'espace te de temps. Dans un premier temps il faut faire une nuance entre les approches utilisées. Dans un deuxième temps on a les problèmes qui dépendent du temps (évolution

temporelle) et les problèmes stationnaires (qui ne dépendent pas du temps). Équation numérique

$$U_{k+1} = f(U_k) \tag{3.2}$$

où U_{k+1} est un vecteur qui contient les valeurs de $\vec{u}$ en un certains nombre de points x_i du domaine spatial à l'instant t_{k+1} en fonction de l'état U_k à l'instant t_k. On veut trouver un algorithme qui me permet d'obtenir U_k le vecteur solution et que cette solution soit la mieux représentative de la réalité physique temporelle $\vec{u}(x_i, t_k)$. Pour le cas stationnaire la solution du problème numérisé est celle d'un système, que l'on pose linéaire $AU = B$. Le problème auquel on fait face est celui de la dimension élevé (i.e. si la maille est de dimension N X N, alors le vecteur U contient N^2 éléments et la dimension de la matrice est de $N^2 \times N^2$. et comme en général N est très grand, il est impensable de stocker une telle matrice et de résoudre l'équation ci-haut par une méthode d'inversion. Ainsi pour solutionner de tel problème on fait appel aux méthodes itératives. On peut séparer (ou scinder le problème. Le stockage de la matrice de grande dimension est remplacépar la seule programmation des formules qui génèrent ses éléments et où loa solution est atteinte par la convergence d'une suite de calculs répétés. par exemple on peut chercher à scinder la matrice A sous la forme $A = M + N$ et à écrire suceesivement

$$MU = -NU + B; \qquad MU_{k+1} = -NU_k + B \tag{3.3}$$

ceci correspond à un algorithme itératif. Cet algorithme doit répondre à deux exigences: 1) faisabilité du calcul arithmétique de U_{k+1} (résolution de M) 2) convergence de U_k vers $A^{-1}B$ lorsque $\lim k \to \infty$.

Manière de scinder A et qui satisfait aux deux exigences ci-haut implique que l'on peut résoudre le problème. But: trouver une méthode optimale pour découper A et qui donne la convergence la plus rapide (nombre d'itératipons optimales). On veut réduire le temps de calcul (CPU time) et son coût. Pour cela on développe des algorithmes performants à convergence accélérée.

En conclusion, la résolution numérique des équations aux dérivées partielles, qu'elles fassent ou non intervenir le temps, conduit toujours à des calculs itératifs, c'est-à-dire à des problèmes "d'évolution numérique". L'évolution numérique a une signification temporelle que si l'équation d'origine est elle même une équation d'évolution temporelle et on comprend que les méthodes de résolution en soient tout à fait particulières. la nature de l'équation, stationnaire ou d'évolution, puis elliptique, parabolique ou hyperbolique, induit effectivement des méthodes de résolution très différentes.

REMARQUE très intéressante au sujet des méthodes numériques. Il existe deux classes, méthode d'approximation d'équations puis la méthode d'approximation de la solution. ce sont des méthodes qui conduisent à des algorithmes numériques destinés à être implantés sur ordinateur. Équation physique continue aux dérivées partielles:

1. méthode d'approximation d'équations, c'est-à-dire reformulation discrète approchée de l'équation d'origine puis résolution exacte de l'équation approchée,
2. méthode d'approximation de la solution, c'est-à-dire reformulation intégrale exacte de l'équation d'origine puis résolution approchée de l'équation exacte.

In summary, the following steps have to be defined in the process of setting up a numerical scheme:

1. *Selection* of a discretization method of the equations. This implies selection between finite difference, finite element or finite volume methods as well as selection of the order of accuracy of the spatial and, eventually, time discretization.
2. *Selection* of a resolution method for the system of ordinary differential equations in time, for the algebraic system of equations and for the iterative treatment of eventual non-linearities.
3. *Analysis* of the selected numerical algorithm. This step concerns the analysis of the 'qualities' of the scheme in terms of stability and convergence properties as well as investigation of the errors generated.

3.2 Finite Volume

3.2.1 Introduction

In this section i will go through a review of the conservative method, specifically, hyperbolic systems of equations and all the mathematical tools related to this field.

Finite Volume starts from an integral formulation of conservation laws which describe the physical fondamental processes in fluid dynamics. According to one‘s point of view, it can be considered as a finite difference method applied to the differential, conservative form of the conservation laws, written in arbitrary coordinates, or as variant of a weak formulation. The finite volume is nowadays the dominant technique of integration of the hydrodynamics equations of gas flows. These finite volume schemes approximates the integral form of the basic conservation laws. At each time step, they solve for an integral mean of the flows variables in each cell. The volume finite method is a discrete formulation of the integral conservation law. Hence, it gives a natural framework into which a conservative shock-capturing scheme can be developed and implemented.

3.2.2 Conservative Schemes

As mentioned before in chapter (?), the equation of the fluids mechanics can be expressed in a conservative form or a conservation law which express the conserved quantity as mass, momentum and energy. Differential form of a conservation law

$$\partial_t u + \vec{\nabla} \cdot \vec{f}(u) = 0. \tag{3.4}$$

where f is the convective flux. (also called divergence equation). If we write this last equation in its integral form

$$\int_{x_1}^{x_2} u(x,t_2)\,dx - \int_{x_1}^{x_2} u(x,t_1)\,dx = \int_{t_1}^{t_2} f(u(x_2,t))\,dt - \int_{t_1}^{t_2} f(u(x_1,t))\,dt \tag{3.5}$$

and divide the former by the quantity $1/h$ ($h \equiv dx$) and let

$$\overline{u} = \frac{1}{h}\int_{x_1}^{x_2} u(x,t)\,dx \tag{3.6}$$

The dependent variables are mean cell values on each volume which we solve on each time step.

$$\overline{u}(x,t_2) - \overline{u}(x,t_1) = \frac{1}{h}\left\{\int_{t_1}^{t_2} f(u(x_2,t))\,dt - \int_{t_1}^{t_2} f(u(x_1,t))\,dt\right\} \tag{3.7}$$

where u is the state variables and f the physical flux of the states variables. This equation express simply that the quantity u inside a volume depend only on the flux at the surface (no source inside the volume), it mean $\int_{-\infty}^{\infty} u(x)dt = \text{const}$. We have now an equation which express the time evolution of a mean cell value in terms of the flux. Integral form

$$\frac{\partial}{\partial t}\int_{\Omega} u\; d\Omega = \int_{\Gamma} \vec{F}\cdot\vec{n}\; d\Gamma \tag{3.8}$$

where Ω, Γ are respectively the domain and the surface of the domain, dΓ is the surface element, dΩ is thye element of volume and $\vec{n}$ is the normal vector which point outward. This relation express that the variation of the quantity u inside the volume Ω is equal to the contribution of the fluxes at the surface. There is no inside source.

The classification of the flow equations is connected to the mathematical concept of characteristics, which can be defined as a families of line (surface or hypersurface) along which certain properties remain constant or certain derivatives can become discontinuous. It is from the characteristic that we analyse the

behaviour of the physical system. Along these curves (in the x-t) travel certain physical properties (in the case of Shallow-water small perturbations). From this concept we can define zone of influence and depence of a differential equation (which is different for each type of differential equation: points which contribute at the solution and points which are affected by the solution). These last two notions are of a paramount importance if we want get result which agree with the physical reality.

The theory of characteristics yields a very elegant interpretation for the phenomenon of the appearing of discontinuities in the solution of non-linear conservation laws ans has greatly contributed to the evolution of the numerical techniques applied in the discretization of the inviscid flow equations. The solution $u(x,t)$ of a scalar law or any function of u remains constant along a family of curves in the domain of integration $[-\infty,+\infty]\times[0,T]$, called the 'characteristics' or characteristic curves. Along each characteristic curve a constant quantity is transported. Since the slopes of the characteristic curve are in general not constant, intersections can be present between different curves of the same family, transporting different values of the same quantity. Hence, at intersections, the solution should be multivalued, and a single-value solution is possible admitting the formation of a discontinuity.

We can write the differential form of the conservation law (divergence) as follow:

$$\frac{\partial u}{\partial t}+a(u)\frac{\partial u}{\partial x}=0 \tag{3.9}$$

with

$$a(u)=\frac{\partial f}{\partial x} \tag{3.10}$$

where $a(u)$ is a constant (linear case), t represents the time, and x represents the spatial variable. We give $u(x,t)$ at the initial time, which we always take to be 0 -that is, $u(0,x)$ is required to be equal to a given function $u_0(x)$ for all real numbers x -and we wish to determine the values of $u(x,t)$ for positive values of t. This is called the *initial value problem*. By inspection we observe that the solution

$$u(t,x)=u_0(x-at)$$

The formula tell us several things. First, the solution at any time t_0 is a copy of the original function, but shifted to the right, if a is positive, or left, if a is negative, by an amount $|a|t_0$. Another way to say this is that the solution at (t,x) depends on the value of $\xi=x-at$. The lines in the (x,t) plane on which $x-at$ is constant are called *characteristics*. The parameter a is the speed of propagation along the characteristic. Thus the solution of (?) can be regarded as a wave that propagates with speed a without change of shape. Secondly, whereas (1?) appears to make sense only if u is differentiable, (2?) require no differentiability of u_0. In general we allow discontinuous solution for hyperbolic problems.

$$\frac{du}{dt}=\frac{du}{d\xi}\frac{d\xi}{dt}=-a\frac{du}{d\xi}$$
$$\frac{du}{dx}=\frac{du}{d\xi}\frac{d\xi}{dx}=\frac{du}{d\xi}$$

if we replace in the original equation we find $du/dt=0$, differential equation for the characteristic. This equation specify that the information propagate along the curve with constant speed.

In linear case if a dicontinuity is present initialy, it will travel along a caracteristic, in non-linear case this is no more true. We will see later that we can generalize the linear case to the non-linear case by freezing some state.

Discontinuities may appear in the time evolution of the solution of a non-linear conservation law even if the initial solution is smooth. This fact provides a challenge to the development of numerical methods for the integration of conservation laws and systems of conservation laws.

It is from the characteristic that we can define the influence and dependence zone of the PDE and the nature (elliptic, parabolic and hyperbolic).

We can generalize the scalar conservation law introducing the definition of an hyperbolic system of conservation laws. Eq. (below) deals with a system of equations if we substitute the scalar conserved quantity u with a vector of conserved quantity U. Introducing the Jacobian $\vec{A}(U)$:

$$\vec{A}(u) = \frac{\partial \vec{F}}{\partial U} \tag{3.11}$$

Here $U = (u_1, \ldots, u_m)$ is the state vector and F(U), the flux, is a vector of m components. The equation above can be rewritten as a first-order system of quasi- linear equations:

$$\frac{\partial U}{\partial t} + \vec{A}(U) \cdot \vec{\nabla} U = 0 \tag{3.12}$$

In one dimension, $\vec{A}$ has only one component a. The monodimensional system is said to be hyperbolic if the Jacobian matrix A is diagonalizable and all its eigenvalues are real.

$$\lambda_1 \leq \lambda_2(u) \leq \ldots \leq \lambda_m(u) \tag{3.13}$$

If they are distinct, the system is said "strictly" hyperbolic.

In more than one dimension, the system is said hyperbolic if any linear combination with real coefficients of the components of the Jacobian $\vec{A}(U)$ yields a matrix which satisfies the same property as in the monodimensional case, *i.e. it is diagonalizable with real eigenvalues.*

When programming discontinuous solution, upwind scheme become very important. We use the field-by-field decomposition which consist of solving the system of equations along the characteristics, the sign of the eigenvalues (characteristic speed) give the information about the direction of propagation of the information.

To solve this system of equations we will use the characteristic variable (or space). We consider the linear case lets define:

$$V = RU, \qquad U = R^{-1}V \tag{3.14}$$

where R is the matrix of eigenvectors of A and v is the characteristic variable. Then the equation (?) becomes

$$R^{-1}\frac{\partial V}{\partial t} + AR^{-1}\vec{\nabla} \cdot V = 0 \tag{3.15}$$

if we multiply the latter by R we obtain the characteristic equation

$$\frac{\partial V}{\partial t} + RAR^{-1}\vec{\nabla} \cdot V = 0 \tag{3.16}$$

With $\Lambda = RAR^{-1}$ which its the diagonal matrix with the eigenvalues the diagonal elements (change basis).

$$\Lambda = \begin{pmatrix} \lambda_1 & 0 & \cdots & 0 \\ 0 & \lambda_2 & \cdots & \vdots \\ \vdots & & \ddots & 0 \\ 0 & \cdots & 0 & \lambda_p \end{pmatrix} \tag{3.17}$$

A partial differential equation is in a sense easier to manipulate than the corresponding integral equation in order to derive numerical discretizations. Unfortunately, it does not hold at discontinuities. When a discontinuity appears, a classical solution, which is differentiable everywhere, is not possible, hence, the conservation law in differential form is no more valid. A possible approach is to supplement the differential

equations by providing additional "jump relations", that must be satisfied across discontinuities. These supplementary conditions are just limiting forms of the integral relations. In fact, the integral formulation is still valid, even if discontinuities are evolving in the solution.

Continuity relation and shock speed:

$$\int_\Gamma \phi(f_2 - f_1)\ dt - (u_2 - u_1)\ \ dx = 0 \tag{3.18}$$

or equivalently

$$\frac{dx}{dt} = \frac{f_2 - f_1}{u_2 - u_1} \tag{3.19}$$

The discontinuous function u satisfying condition (?above) is a weak solution of the integral conservation law. Eq. (?above) defines the jump conditions that must be satisfied by any function u to be a solution of the integral conservation laws. This relation is generally known under the name of Rankine-Hugoniot jump conditions.

The Riemann problem is a particular initial value problem (IVP) which consist of a conservation law or a system of conservation laws with a discontinuous initial solution of the type:

$$u(x,0) = \begin{cases} u^l & : \quad x \leq x_i \\ u^r & : \quad x \geq x_i \end{cases}$$

The initial solution is characterized by the two constant states u^l and u^r respectively on the left (l) and on the right (r) of the position $x = 0$, where a finite discontinuity is located. In the case of Euler equations there is always a solution and the algorithm which solves the problem is called a Riemann solver.

On obtient donc m relations de saut ("jump relation"). Voir le livre de Leveque au chapitre 6 (section 6.5).

$$[u] = (\beta_p - \alpha_p) r_p \tag{3.20}$$

where r_p is the eigenvector of the p(th) eigenvalue.

These relations satisfy the jump conditions of Rankine-Hugoniot since

$$\begin{aligned} f(u) &= Au, \qquad [f] = A[u] && (3.21)\\ &= (\beta_p - \alpha_p) A r_p && (3.22)\\ &= \lambda_p [u] && (3.23) \end{aligned}$$

Solving a Riemann problem is to find a way to decompose the jump $u_R - u_L$ as a sum of jump

$$u_R - u_L = (\beta_1 - \alpha_1) r_1 + \ldots + (\beta_m - \alpha_m) r_m \tag{3.24}$$

Because of non-linearity of the equations considered, discontinuity can be present in the solution and the numerical scheme must be able to treat this correctly. Therefore, the spatial discretization is based on finite volume method of the integral equation. The main reason to use a conservative scheme we make that we capture the shock correctly: its position, speed if it is not stationary,.. A capturing-shock technique must reproduce a non-oscillatory solution and do not introduce dissipation, precision of the solution where it is smooth.

When trying to solve a non-linear problem we face new difficulties (especially when discontinuities are present because of the hyperbolic nature of the equations). The numerical method may converge to a solution which is not a weak solution of the problem under consideration. We want to make sure that the numerical method produce the correct solution. One possibility is to work with the integral form (as mentioned before it

is hard to discretize). Another way is to impose a condition to satisfy on the numerical scheme (conservative form).

One way to derive numerical methods in conservation form is to use standard finite difference discretizations but to start with the conservative form of the PDE.

We integrate the differential equation on a volume $(x_{j-1/2}, x_{x_j+1/2}) \times (t_n, t_{n+1})$. If U_j^n (discrete values obtained from a discretization or by initial condition) is an approximation of the mean value $\overline{u}_j^n$ (defined as mean integral), we have a natural formulation for the numerical scheme. We know that the weak solution $u(x,t)$ satisfy the integral form of the conservation law. Lets $U_j(t)$ be an approximation of the mean value of u for the cell j at time t (value centre cell $dx = h$)

$$U_j(t) \approx \overline{u}_j(t) \equiv \frac{1}{h} \int_{x_{j+1/2}}^{x_{j-1/2}} u(x,t)\ dx \tag{3.25}$$

From the theory of conservation law, the integral form for a system of m equations is:

$$\frac{d}{dt} \int_{x_1}^{x_2} u(x,t)\ dx = f(u(x_1,t)) - f(u(x_2,t)) \tag{3.26}$$

The last equation give us an equation for evolution of the mean value

$$\overline{u}_j'(t) = -\frac{1}{h}\Big[f(u(x_{i+1/2},t) - f(u(x_{i-1/2}))\Big] \qquad (3) \tag{3.27}$$

Numerical $F(U_j, U_{j+1})$ are view as mean flux across the interface $x_{j+1/2}$ time on a time interval $[t_n, t_{n+1}]$:

$$F(U_j, U_{j+1}) \sim \frac{1}{dt} \int_{t_n}^{t_{n+1}} f(u(x_{j+1/2}, t))\ dt \tag{3.28}$$

De même pour les flux, ceux-ci seront approximer par $\mathcal{F}(U^n, j)$, we obtain a discrete system of ordinary differential equations for the $U_j(t)$

$$U_j'(t) = -\frac{1}{h}\Big[\mathcal{F}(U(t), j) - \mathcal{F}(U(t), j-1)\Big]. \tag{3.29}$$

It is important to note that this represent a system of equations since each flux $\mathcal{F}$ depend on two or more $U_i(t)$.

For a simple case, $p = 0$ and $q = 1$ then the flux function depend only on two variables and becomes

$$U_{n+1}^n = U_j^n - \frac{dt}{dx}\Big[F(U_j^n, U_{j+1}^n) - F(U_{j-1}^n, U_j^n)\Big]. \tag{3.30}$$

which we can write more generally:

$$U_j^{n+1} = U_j^n - \frac{dt}{dx}\Big[F(U_{j-p}^n, U_{j-p+1}^n, \ldots, U_{j+q}^n) - F(U_{j-p-1}^n, U_{j-p}^n, \ldots, U_{j+q-1}^n)\Big] \qquad (1) \tag{3.31}$$

for a function F with $p + q + 1$ arguments. F is call a numerical flux.

3.2.3 Upwind Schemes

We would like to introduce more physical property into the numerical scheme and the effect of non-linear effect. Since the hyperbolic equation correspond the physical phenomenon of wave propagation (non-linear wave). The hope is that taking into account in a more or less direct way the non-linear propagation properties

of the waves can prevent the appearence of unwanted numerical effects like oscillations near discontinuities. We will distinguish two different levels at which the physical properties of the problem can be introduced in the discretization.

At the first level, only information about the sign of the propagation speed of the characteristic waves is taken into account. Historically, the origin of upwind methods may be taken back to the work made by Courant, Isaacson and Reeves, who proposed to solve certain equations along the characteristics going back from any point of the grid at the time step t_{n+1} to the time step t_n (CIR method). The crucial point of upwind differencing is in the introduction of some knowledge about the physical properties of the flow equations into the discretization.

Then the first-order upwind scheme can be written as:

$$u_j^{n+1} = u_j^n - \frac{\Delta t}{2\Delta x}\left(f_{j+1} - f_{j-1}\right) + \frac{\Delta t}{2\Delta x}\left[|A_{j+\frac{1}{2}}|(u_{j+1} - u_j) - |A_{j-\frac{1}{2}}|(u_j - u_{j-1})\right] \tag{3.32}$$

where $A = \frac{\partial f}{\partial u}$ is the Jacobian matrix which is diagonalized by the matrix of the left and right eigenvectors R^{-1} an R. Hence, $\Lambda = R^{-1}AR$ an $|A|$ is defined as $|A| = R|A|R^{-1}$. In the non-linear case, for a general system of conservation laws, the Jacobian matrix A and the matrix—A— have non-constant terms, which can generally depend on the solution. Usually, the approach to a numerical discretization consists in locally freezing them about an average estimate of the state variables, like the Roe's average for Euler equations. Different ways of taking in consideration the eigenvalues of the Jacobian matrix and the propagating waves have been widely explored in the last decade; we mention for completeness the *flux-vector splitting*, where flux terms are split and discretized directionaly to the sign of the associated propagation speeds.

Upwind differencing helps the numerical discretization to be faithfull to the physical mechanism of propagation of waves signals underlying the convective part of the equations. The upwinding can be directly introduced in the flux estimation via a Riemann solver which correctly considers the non-linear interactions among the different families of waves forming the solution.

Upwind-differencing attemps to discretize hyperbolic partial differential equations by using differencing biased in the direction determined by the sign of the characteristic speeds. On the other hand, symmetric or central schemes do not consider any information about the wave propagation in the discretization.

At the second level, the physical properties of the equations are more deeply introduced in the discretization, augmenting the complexity of the scheme and the computational effort. This approach was originally proposed by Godunov in (1959) and all its generalisations and extensions are sometimes referred as *flux-difference splitting* methods or *Godunov-type* methods. Godunov in his pioneering work suggested to consider the propagation of the basic wave solution of the Euler equations by solving local Riemann problems in time, instead of following characteristics backward in time (like in CIR method).

3.2.4 High Order Schemes

The Godunov scheme approximate the solution at each interface by a piecewise-constant extension of the cell-averaged values assigned in each cell to the centroid. Being the spatial error of the same order of the cell size Δx, this strategy produce a discretization which is only first-order accurate in space.

The discontinuous piecewise constant distribution of those integral mean values correspond naturally to a class of scheme pionnered by Godunov. To assess the evolution of the piecewise discontinuous data, appeal is made, in gas dynamics, to the Riemann solution of the model shock tube problem with discontinuous piecewise constant initial data.

The Godunov's scheme may be established in three main steps:

- a piecewise-constant function approximates the solution inside each cell at $t_n = n\Delta t$

- the solution of the local Riemann problem is provided by the application of a Riemann solver
- the update in Godunov-type schemes is done on the cell-averaged conservative variables and requires the estimation of the numerical fluxes at cell-interfaces and a sucessive integration in time over a time step. The state variables at time $t_{n+1} = (n+1)\Delta t$ are obtained by averaging the local solution inside each cell

From an historical point of view, shock-capturing can be classified into two general categories: namely, classical and modern shock capturing methods. A stable calculation in presence of shock waves requires a certain amount of numerical dissipation, in order to avoid spurious numerical oscillations, which could limit accuracy and prevent convergence towards steady solutions. In the case of classical shock-capturing methods, numerical dissipation terms are usually linear and the same amount is applied at all grid points. Classical shock-capturing methods only exhibit accurate results in the case of smooth and weak-shock solution, but when strong shock waves are present in the solution, non-linear instabilities and oscillations can arise near discontinuities. Modern shock-capturing methods have, instead, a non-linear numerical dissipation,with an automatic feedback mechanism which adjust the amount of dissipation in any cell of the mesh, in accord to the actual solution. These schemes have proven to be stable and accurate even for problems containing strong shock waves.

Modern shock-capturing methods are generally based on "upwind- differencing", in opposite to classical symmetric or central discretization.

Finite-Volume Technic Taking an inertial frame of reference and dividing the domain of integration into finite contiguous cells, whose volume $V(t)$ can vary in time, the general form of a conservation law is

$$\int\limits_{V(t_2)} u\ dV - \int\limits_{V(t_1)} u\ dV + \oint\limits_{t_1}^{t_2} \vec{n} \cdot \vec{F}\ dSdt = \int\limits_{t_1}^{t_2} \int\limits_{V} PdVdt \tag{3.33}$$

$\vec{n}\ dS$ is a vector element of surface area with outward normal $\vec{n}$, u is a conservative variable per unit volume, $\vec{F}$ is the flux of u per unit area per unit time, and P is the rate of production of u per unit volume per unit time. If u and P are scalars $\vec{F}$ is a vector while if u and P are vectors $\vec{F}$ is a tensor. If we assume that all variables are continuous in time, equation (2.1) takes the form:

$$\frac{d}{dt}\left[\int\limits_{V} u\ dV\right] + \oint \vec{n} \cdot \vec{F} dS = \int\limits_{V} PdV \tag{3.34}$$

Eq. (?) must be satisfied both globally taking for V the full domain of integration and locally taking for V a computational cell. It is important to remark that the time variation of u inside the volume V depends essentially on the surface-values of the fluxes.

The computational domain is partitoned into a finite number of control volume $[x_{i-1/2}, x_{i+1/2}]$ around a nodal position x_i. The position of the right face $x_{i+1/2}$ is defined as $(x_i + x_{i+1})/2$. The differential system is integrated over each control volume to produce the discrete equivalent of the conservation law:

$$U_i^{n+1} = U_i^n + \frac{dt}{dx}\Big[(F_{i-1/2} - F_{i+1/2}) + Q_i^n\Big] \tag{3.35}$$

with Q is the source term. This integration technique forms the basis of what is known as the finite volume method. The specific difference between various finite volume schemes are the way in which they approximate the interface convective flux $F_{i+1/2} = F(U(x_{i+1/2}, t))$.

3.3 Polynomial Reconstruction Methods

The design of high-order accurate numerical schemes for conservation laws and hyperbolic systems is concerned with the following problem of the approximation of functions:

The order of this polynomial gives the formal spatial order of accuracy of the resulting numerical method. That is, the spatial order of accuracy whenever the solution is smooth.

Shock capturing schemes

Note: Deux classes de schémas de capture de choc pour la discrétisation du terme d'advection $(\vec{v} \cdot \vec{\nabla})\vec{v}$ responsable de la fromation de discontinuités. La première classe consiste en des schémas dits "à variation totale décroissante" (TVD). La seconde classe est celle qui a été proposée par Godunov, dont on considère aussi les généralisations d'ordre élevé par des méthodes de reconstruction polynômiale (ENO).

When discontinuity are present in the solution, appropriate numerical techniques must be adopted in order to perform computation. These techniques are generally known under the name of "shock-capturing methods". The numerical schemes presented in this chapter are concerned with Godunov-type scheme and its high-order generalizations by interpolation and reconstruction techniques. The main motivation to using a conservative discretization is that it is able to correctly capture the discontinuity in the solution. The ideal shock-capturing discretization should have some desirable features like providing a good resolution where the solution is smooth even if discontinuities are present elsewhere, being able to correctly locate the discontinuities, producing a sharp and non-oscillatory approximation of the jump, ensuring some entropy condition to select the weak solution corresponding to the physical solution.

A good resolution in smooth regions is obtained by increasing the order of the approximation. Neverthless, this approach can produce spurious numerical oscillations near discontinuities. The appearence of numerical oscillations can be avoided by a limiting function or by a special polynomial reconstruction technique (ENO).

The analogue of the shock tube model in Shallow-water theorie is the dam-break problem. The Riemann solvers in the context of the Shallow water wave equations designated the class of schemes, Godunov, Roe, Osher and Van Leer, that may viewed as providing exact or approximate solution of the local dambreak problem. These schemes have several desirable prperties:

A. they are automatically conservatives for homogenuous equations (mass) because they use discrete law of the divergent or 'conservative' form of the basic equations. This also provides good shock capturing capabilities with correct speed of shock

B. they are monotone because they deal with each nonlinear wave depending of his nature, shock or rarefaction, and consequently they are oscillations free

C. they have an inherent 'upwind' property built in: in the case of supercritical flows they result in the coreect direction of propagation of flow properties from the upstream state to the downstream state

These Riemann solvers have an upwind character since for the single linear advection they reduce to the first order upwind scheme. Moreover, they contains enough 'physical information' to detect shocks and transition between flow regimes through the eigenvalues of the Jacobian flux matrix F.

Chapter 4

Conservative Numerical Schemes

4.1 Introduction

The computational fluid dynamic techniques discussed in this chapter have two aspects in common:

1. It involve the calculation of the flow fields properties at discrete points in the flow. For example, consider an $x - y$ coordinate space which is divided into a rectangular grid (see Fig.?). The solid circles denote grid points at which flow properties are either known or to be calculated. The points are indexed by the letters i in the x direction and j in the y direction. For example, the point directly in the middle of the grid is denoted by (i, j), the point immediatly to its right is $(i + 1, j)$, and so forth.
2. In the theoretical limit of an infinite number of grid points (*i.e.* Δx and Δy the solution are *i.e. exact*. Since all pratical calculations obviously utilize a finite number of grid points, such numerical solutions are subject to *truncation error*.

Depending on the nature and the physical magnitude of the phenomena to be modelled (convection, pressure, diffusion, bathymetry, Coriolis and friction) may be of significant magnitude when compared to the others. To construct a numerical model valid over a broad range of commonly encountered physical situations, it is essential to integrate the discrete representation of each term with an acceptable degree of accuracy and stability. The principal difficulties to be overcome in this system arise from the non-linear convection and bed friction terms.

4.2 Time discretization: semi-discrete methods

Nous pouvons maintenant discrétiser en temps. Par exemple prenons un schéma Euler avant d'ordre $\mathcal{O}(1)$

$$U^{n+1} = U^n - \frac{k}{h}\Big[\mathcal{F}(U^n, j) - \mathcal{F}(U^n, j-1)\Big], \tag{4.1}$$

qui est la forme habituelle d'une méthode conservative (h est le pas en espace et k le pas en temps). L'ordre de précision temporelle est contenue dans la discrétisation de la dérivé en temps et la discrétisation spatiale dans l'évaluation des flux, cette dernière dépendra implicitement de l'ensemble des valeurs discrètes U_j^n. L'ordre de précision total est donné par $\mathcal{O}(h^q + k^l)$. Ainsi pour augmenter la précision spatiale cela implique une plus grande précision des U_j^n. Ce sera l'objet de la section des schémas ENO.

Formally, the equations discretized in space give a set of ODE in time which can be written as:

$$\frac{d\overline{U}}{dt} = L(\overline{U}) \tag{4.2}$$

where $L(\overline{U})$ is the discretization of the spatial flux terms, given by one of the above described scheme. The conserved cell- averaged quantities $\overline{U}$ are advanced in time from the level n to the level $n+1$ with the TVD Runge-Kutta time-discretization schemes intorduced by Shu and Osher (1988) [45].

2^{nd}order

$$\begin{aligned} U^{(0)} &= U^{(n)} \\ U^{(1)} &= U^{(0)} + \Delta t L(U^{(0)}) \\ U^{(2)} &= \frac{1}{2}\left(U^{(0)} + U^{(1)} + \frac{\Delta t}{2} L(U^{(1)})\right) \qquad (4.3) \\ U^{(n+1)} &= U^{(2)} \qquad (4.4) \end{aligned}$$

These schemes are TVD in the sense that it can be shown they are non- increasing the total variation of the spatial part under a suitable CFL condition, see Shu and Osher (1988) [45]. Notice also that both the cases need only two storages level for the $U^{(i)}$ and one for $L(U^{(i)})$, since $U^{(2)}$ can overwrite $U^{(1)}$.

The local time-stepping technique is introduced to accelerate convergence towards a stationary solution. For inviscid calculations a locallly varying time step is defined as:

$$\Delta t \leq CFL\ \Delta t_c \qquad (4.5)$$

where Δt_c is the limit due to the convective terms and the CFL coefficient represents the usual Courant-Friederichs-Levy number.

In the case of vicous calculation, there is both a convective and a diffusive contribution to the local time step, which is now defined as:

$$\Delta t = CFL\left(\frac{\Delta t_c \Delta t_d}{\Delta t_c + \Delta t_d}\right) \qquad (4.6)$$

where Δt_d is the limit due to the diffusive terms.

4.3 High-Order Reconstruction Methods

4.3.1 Flux Extrapolation based on Upwind Scheme

To evaluate the flux at the interface we can do that by different approaches (reconstruction of the variables or by the flux). In this we consider the second approach, all the schemes used in this thesis ... flux extrapolation with different method of reconstruction.

The basic formula for fluxes extrapolation are the following: (backward apply to the positive part of the flux)

$$F^{+b}_{i+1/2} = f^+_i + \frac{\epsilon}{4}\Big[(1-\kappa)(f^+_i - f^+_{i-1}) + (1+\kappa)(f^+_{i+1} - f^+_i)\Big] \qquad (4.7)$$

(forward apply to the negative part of the flux)

$$F^{-f}_{i+1/2} = f^-_{i+1} - \frac{\epsilon}{4}\Big[(1+\kappa)(f^-_{i+1} - f^-_i) + (1-\kappa)(f^-_{i+2} - f^-_{i+1})\Big] \qquad (4.8)$$

The parameter κ define the weight of the flux difference and the second order of the numerical flux is obtained

$$f^{*(2)}_{i+1/2} = F^{+b}_{i+1/2} + F^{-f}_{i+1/2} \qquad (4.9)$$

4.3.2 ENO Reconstruction

Let us now describe the recursive ENO algotithm for determining the optimal stencil for the smoothest reconstruction of f. All the information concerning the smoothness of f can be extracted from a table of divided difference of f. Employing a standard notation in numerical analysis, the k^{th} divided difference of f can be defined recursively as:

$$f[x_i] = f(x_i) \tag{4.10}$$

$$w[x_i, \ldots, x_{i+k}] = \frac{w[x_{i+1}, \ldots, x_{i+k}] - w[x_i, \ldots, w_{i+k-1}]}{x_{i+k} - x_i} \tag{4.11}$$

It can be shown that if the function f is smooth in the interval $x_0, x_k)$ but it is discontinuous in $(x_1, x_{k+1}$, then, for a Δx small enough

$$|w[x_0, \ldots, x_k]| < |w[x_1, \ldots, x_{k+1}]| \tag{4.12}$$

Hence, we can compare the relative smoothness of the function f in two intervals defined by an equal number of adjacent segments by comparing their corresponding divided differences. This fact gives us a powerful tool to select automatically the best stencil for the smoothest interpolation. Since all the points of the interpolating stencil must be contigous, to specify a stencil we need only one of the two extreme points, usually the left-most one, and the number of points in the stencil. The smoothest possible stencil can be built in an iterative way applying the following algorithm:

- since any stencil must include the interval (x_i, x_{i+1}) where the function is reconstructing, the first point in the stencil will be x_i. We can represent this choice by indicating the relative index $j_1(i) = i$
- then we add one point at a time on the left or on the right choosing the one which will give rise to the smoothest interval for the interpolating function:

$$j_{k+1}(i) = \begin{cases} j_k(i) - 1 & if \quad |w[]| < |w[]| \\ j_k(i) & if \quad \text{otherwise} \end{cases}$$

Primitive Variables

The reconstruction via primitive variables provides a very elegant solution to the problem of approximating nodal values of a function once given its cell-averages. The basic idea is quite simple: given a set of cell-averaged values $\{\overline{v}\}$, the fundamental theorem of calculus allows to compute the exact values of the primitive functions at the cell-interfaces by an arbitrary additive constant. The primitives values can be now interpolated in some way, providing a polynomial reconstruction of the primitive function, and the derivative of this polynomial gives the desired nodal reconstruction. Let us formalize the procedure: assume that $\overline{v}_j$ be a set of cell-averages of a piecewise-smooth function $v(x)$, which is the derivative of another (smooth) function $w(x)$. At a fixed time t, the primitive[1]

We can now define the primitive of u(x,t). function $w(x)$ is defined by

$$w(x) = \int_{x_1/2}^{x} u(\xi) d\xi \tag{4.13}$$

The lower limit $x_{1/2}$ is arbitrary, any fixed point could be used. Changing the lower limit only shifts $w(x)$ by a constant, and the property of w that we will ultimately use is that

$$v = \frac{dw}{dx} \tag{4.14}$$

[1]fondamental calculus theorem: if $f(x) = \frac{d}{dx}\ g(x)$, then $\int_a^b f(x)\ dx = \int_a^b \frac{d}{dx}\ g(x)\ dx = g(b) - g(a)$. f(x) is a continuous function in the interval [a,b]

which is unaffected by a constant shift. Equation (?above) allows us to obtain pointwise values of u if we have a good approximation of w. First, it is important to notice that knowing the mean values of u this gives us the exact values of w at $x_{i+1/2}$. Lets

$$W_j = w(x_{i+1/2}) = \int_{x_{1/2}}^{x_{i+1/2}} u(\xi, t)\ d\xi \qquad (10) \tag{4.15}$$

this is dx times the mean values of u for j(cells) then

$$W_j = h \sum_{i=1}^{j(cells)} \overline{u}_i(t). \qquad (11) \tag{4.16}$$

The discrete pointwise-values at any cell-interface position $x_{j+1/2}$ can be computed by applying the following relation:

4.4 One-Dimensional Case

Here we desribe the algoritm of the three different schemes used for the simulation. These are suited .. hydraulic jump and high velocity regime over a complex bathymetry. The numerical technique considered here is based on a structured cell- centered Finite Volume discretization. Dans cette thèse nous considérons une classe de schémas numérique qui généralisent les schémas de Godunov et son extension à l'ordre deux à n'importe quel ordre de précision. Ces schémas peuvent être écrits sous forme conservative standard

$$v_j^{n+1} = v_j^n - \lambda(F_{i+1/2} - F_{i-1/2}) \equiv \left(\overline{E}(\tau) \cdot v^n\right)_j \tag{4.17}$$

Ici $\overline{E}(\tau)$ est l'opérateur solution numérique et $F_{i+1/2}$ est le flux numérique, une fonction de 2k variables

$$F_{i+1/2} = F(v_{j-k+1}^n, \ldots, v_{j+k}^n) \tag{4.18}$$

lequel est consistent avec le vrai flux (flux physique) f(u) dans (1) dans le sens que $F(u, u, \ldots, u) = f(u)$.

4.4.1 Algorithm 1: a simplified version of ENO schemes

This scheme has been developped by Harten, Lax and Van Leer in the .. The scheme presented here belongs to the flux-vector splitting upwind family discussed in the previous section. Use a local adaptive stencil to obtain information automatically from regions of smoothness when the solution develops discontinuities. As a result, approximations using these methods can obtain uniformly high-order accuracy right up to discontinuities, while keeping a sharp, essentially non-oscillatory shock transition

$$\mathbf{f}_{j+1/2} = \mathbf{f}_{j+1/2}^+ + \mathbf{f}_{j+1/2}^- \tag{4.19}$$

This is a simplified version of the ENO scheme proposed by Shu and Osher (1988). The numerical flux is splitted up and each of the fluxes $f_{j+1/2}^+$ and $f_{j+1/2}^-$ is approximated up to the required order of accuracy using ENO moving stencil idea. Second-order accurate numerical fluxes are defined by:

$$\begin{aligned} f_{j+1/2}^+ &= f_j^+ + 0.5\delta f_j^+ \\ f_{j+1/2}^- &= f_{j+1}^- - 0.5\delta f_{j+1}^- \end{aligned} \tag{4.20}$$

where

$$\begin{aligned} f_j^+ &= 0.5(f_j + \alpha u_j), \\ f_j^- &= 0.5(f_j - \alpha u_j) \end{aligned}$$

where α represents some positive coefficients, and it is required that

$$\alpha \geq \max|\lambda_i|, \qquad i = 1, \ldots, m \tag{4.21}$$

4.4.2 Algorithm 2: a simplified version Roe's Approximate Solver

This scheme has been proposed by Shu and Osher .... The numerical scheme is based on the first order Roe's scheme. Roe's approach replaces the Jacobian matrix in the divergent form by a constant Jacobian matrix (linearize about some constant state). The scheme read

$$f_{j+1/2} = 0.5\Big[F^R + F^L - |A_{j+1/2}|(U^R - U^L)\Big] \tag{4.22}$$

where $F^R = f(U^R)$ and $F^L = f(U^L)$. The matrix $A_{j+1/2}$ represents Roe's average matrix for the cell face $x_{j+1/2}$ and it satisfies $\Delta\, f_{j+1/2} = A_{j+1/2}\, \Delta u_{j+1/2}$, called the Rankine-Hugoniot condition.

In Nujic approach, the Roe's average matrix is replaced by a constant coefficient. To obtain a higher order of accuracy, the intermediate states U^L and U^R are obtained by using MUSCL type of extrapolation in an analog way as equations (9) and (10), using *minmod* function (limiter). The quantities δU_j and δU_{j+1} is approximated up to the required accuracy using a moving stencil ENO type (using a 3 points). The quantity δU_j is introduced to limit U_j to avoid the introduction of spurious oscillations in the numerical solution. Second-order accurate numerical fluxes are obtained for $f_{j+1/2}$ in the Lax-Friedrichs type scheme using

$$f_{j+1/2} = \frac{1}{2}(F^L + F^R) - \frac{\alpha}{2}(U^R - U^L) \tag{4.23}$$

The coefficient α is defined as

$$\alpha \geq \max_j|\lambda_j|, \qquad i = 1, 2, \ldots, N \tag{4.24}$$

where N is the number of computational nodes in the problem. This is no more than satisfying the Courant condition. The Lax-Friedrichs implementation of the ENO scheme is accurate, robust and very simple to implement.

4.4.3 Algorithm 3: HLL Approximate Riemann Solver

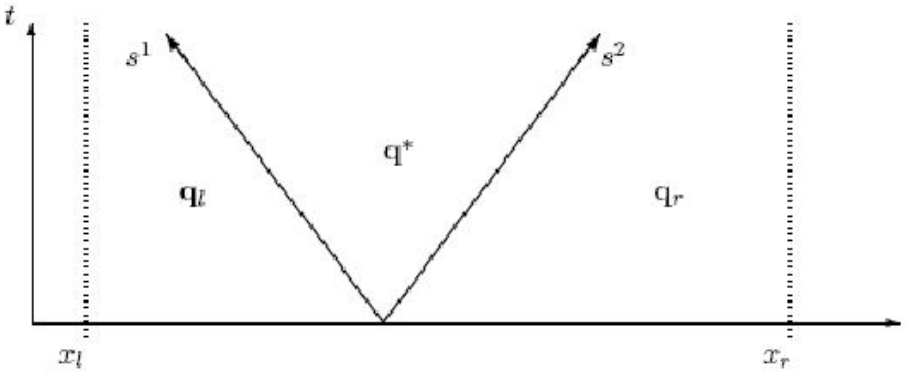

Figure 4.1: Constructing a solution using conservation and estimated speed

This scheme is based on estimating the speeds that information or waves propagate away from a Riemann problem. A linear solution with two discontinuities is then constructed[2], using estimates for the speeds of the

[2]Two speeds are used in the original HLL method even for equations with more than two characteristic families

propagating discontinuities[3]. The first-order HLL scheme is based on the scheme described by Harten et *al.* (1983). The intercell flux, $f_{j+1/2}$ is evaluated by solving the approximate Riemann problem which involves only two shocks, although two rarefaction fans could be considered, separating three states; left-right, and intermediate state. For the Shallow-Water equations Toro(1992) presented a suitable HLL-type flux based on the suggested approximations of Harten. The intercell flux is chosen from

$$F^{HLL}(U^L, U^R) = \begin{cases} F^L & \text{if} \quad 0 \leq S_L \\ F^* & \text{if} \quad S_L \leq 0 \leq S_R \\ F^R & \text{if} \quad S_R \geq 0 \end{cases}$$

For the shallow-water wave equations, the intermediate states u^* and h^* are given by,

$$\sqrt{gh^*} = \frac{1}{2}(\sqrt{gh_L} + \sqrt{gh_R}) - \frac{1}{4}(u_R - u_L) \tag{4.25}$$

$$u^* = \frac{1}{2}(u_L + u_R) + \sqrt{gh_L} - \sqrt{gh_R} \tag{4.26}$$

and are used to estimate the shock speeds.

$$\begin{aligned} S_L &= \min\{u_L - \sqrt{gh_L}, u^* - \sqrt{gh^*}\} \\ S_R &= \min\{u_R + \sqrt{gh_R}, u^* + \sqrt{gh^*}\} \end{aligned} \tag{4.27}$$

which are estimates of the largest and smallest propagation speed. Although this may overestimate the true wave speed, Fraccarollo and Toro (1995) suggest that this enhance the stability of the scheme. These wave speeds are used to solve for the flux in the intermediate state

$$\mathbf{F}^* = \frac{S_R F_L - S_L F_R + S_L S_R (U_R - U_L)}{S_R - S_L} \tag{4.28}$$

by satisfying the integral form of the conservation law over a control volume. Note that the above given wave speeds are obtained under an assumption of wet bed, *i.e.* a non-zero flow depth h, on both side of the computational domain.

[3]Contrast this to a method such as Roe solver, where a constant estimate to the Jacobian matrix is constructed first, and the eigenvalues of this estimate subsequently affect the approximate Riemann solution

Chapter 5

Numerical Results

5.1 Presentation of the Real Case

A fundamental aspect when dealing with numerical schemes is to be able to check their prediction against suitable test problems, preferably ones for which an exact solution is available. This is not the case for the example we present in this thesis. The procedure to follow is then first to make a *calibration* on a well documented regime and secondly to ensurethe *validation* on another regime.

For the real life problem, the upstream inflow discharge Q is approximatively constant with value of $2500\frac{m^3}{s}$. The purpose of the simulation is to obtain flow field information depending on the discharge... In order to carry out the computations, besides the bathymetry data illustrated by means of isocontours in Fig.?, two more parameters have to be specified: the downstream surface elevation H_d and the Manning coefficient N. Because these variables are not necessarily known, two preliminary calibrations have been conducted.

The first step consisted of the calibration of the Manning coefficient N. This has been done by conducting a simulation with a known (measured) downstream water surface level $H_d = 74.0m$ and a high upstream discharge rate $2500m^3/s$. .. where surface level data is available, noted at location R4 to R9, was used as reference points. Some computer runs were executed and the Manning coefficient was monitored. Finally it was found that the following values of the Manning leads to a predicted (steady) water surface level H_{R9} of $74.55m$ which reasonably matches the measured value of $74.37m$. Fig.? gives, by means of the velocity field, a qualitative idea of the computed flow phenomenon for this set of parameters.

With the estimated value of the Manning, a second simulation was conducted to calibrate the downstream water surface level for a low discharge rate of $1100m^3/s$. The value $H_d = 72.3m$ was found to give a surface level of $72.2m$ at location R9, which is very close to the measured level of ....

With this information at hand, it is possible to predict solutions for different discharge. For a given discharge Q between 1100 and 2500 m^3/s, the downstream surface elevation is calculated using the two corresponding downstream elevations together with a linear interpolation.

5.2 Hydraulic Application

5.2.1 Introduction

Many class of shock-capturing exist for the discretization of the advection term $(\vec{v} \cdot \vec{\nabla})\vec{v}$. First one of the most method used in hydraulic is TVD ("Total Variation Diminishing") where the numerical flux is second order is limited by a first order correction which introduce enough dissipation for the stability condition.

The second class is one proposed by Godunov with its generalization of higher order called ENO (Essentially Non- Oscillatory).

The code presented in this thesis implements three different Riemann solvers: one-dimensional solver in the formulation of ENO fluxes, two approximate the Roe's solvers and the HLL one.

The numerical discretizations which couple a non-oscillatory polynomial approximations and a Riemann solver for the inviscid part of the equations are generally called "Godunov-type" schemes. Finally, the viscous terms are discretized applying a standard 2^{nd} order central differencing scheme.

The cell-centered solution is advanced in time by an explicit 2^{nd} order multistage Runge-Kutta scheme. The solver implements also some standard techniques to accelerate the convergence when a steady state solution is rewuired.

The class of the schemes considered offer a great reliability by avoiding numerical oscillations near sharp gradient, and a large variety of fluid flow problems has been solved numerically using them. Among all the numerical solutions of the cases reported in literature, we selected three test cases for the validation of the solver. Basically the main criterium of our choices was to investigate the capability of the numerical discretization in capturing strong shock patterns in steady flows. The result are reported in the first part of the next chapter and are in good agreement with the litterature.

5.2.2 The 1D Solver

A number of numerical schemes, actually four, for solving the one-dimensional shallow-water wave equations applied to problems containing discontinuities in the solution have been examined. Results from several computations show that the overall performance of these algorithms can be considered as very good and allows for accurate open-channel flow computations involving hydraulic jump. In order to study the performance of the numerical scheme, simulations have been performed on a test case characterized by a supercritical flow (dam-break problem). A CFL number of 0.6 was chosen in order to compare the first and second order scheme introduced in section 6, respectively. Figures 8-11 present the results for the 1D dam break with $h_L/h_R = 2.$, show ability of the model for simulating dam-break type flows. In the modeling, 100 computational cells are used, *i.e.* the spatial step is set to 0.1 m. The domain that we are considering is the following:

$$\Omega = \{(x,t) \in \mathbf{R}^2 \,|\, 0 < x < 1, \quad 0 < t < T\}$$

The numerical results compare very well with the exact solution, and show that the schemes are stable and that a little oscillations are introduced. HLL scheme show that the method can cope better with complex supercritical flows than other classical schemes. The algorithm is capable of capturing sharp discontinuities without generating spurious oscillations. With the exception of naive first-order schemes, all schemes produce reasonable results. First-order central scheme (e.g. Lax-Friedrichs) have a symmetry property about the eigenvalue (change sign) of the Jacobian, hence unable to distinguish between upstream and downstream influence (Hirsch 1988) which suffer of oscillations. Sometimes these oscillations can be minimized by using sufficiently small step-sizes.

5.2.3 The 2D Solver

5.3 Discussion and Open Question

Galerkin Finite Element method coupled with a Newton-Raphson solver to solve numerically the Saint-Venant equations on triangular grids. This numerical scheme achieves a satisfactory stability for simulating wetting and drying phenomena in the presence of complex flow and steep topographical gradients.

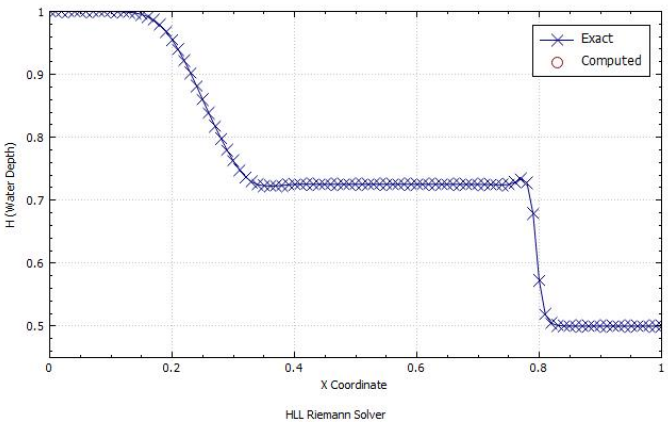

Figure 5.1: HLL Solver

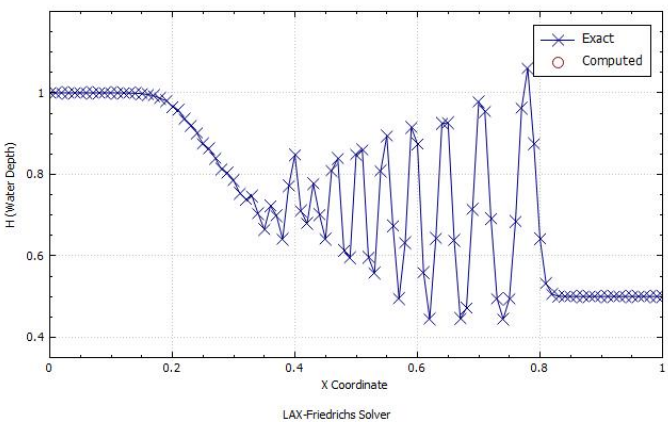

Figure 5.2: Lax-Friedrichs Solver

Table 5.1: Table 1: Mean errors between observed (Fig. 5) and calculated (Fig. 6) water level H (m). R9 is the only point for which we know the exact bathymetry.

Point	H_{obs}	H_{cal}	*Error*
R4	76.92	76.50	.42
R5	77.57	78.39	.82
R6	77.33	78.20	.87
R7	77.16	78.16	1.
R8	76.33	77.20	.87
R9	74.30	74.25	.05

In the torrential part of the river, where the water advection is dominant, spatial numerical oscillations (for the water level in particular) are observed. This is a known problem with Galerkin Finite Element Method which can be controlled by decreasing the Peclet number. Refining the mesh there eliminates the numerical oscillation but increases the computation time and memory requirements. Alternatively, an upwinding technique (Hirsch 1988) or an SUPG (Hugues and Brooks 1982) with a fully conservative scheme would also reduce the numerical oscillation without the need to modify the mesh.

The example of a $2km$ section of the Ste-Marguerite River in Canada could be used as a *benchmark* to test new numerical solvers dedicated to flow conditions under adverse flow conditions.

Despite the fact that the Galerkin scheme is not shock-capturing and thus not naturally adapted to the simulation of torrential regimes, when coupled to a Newton-Rapson solver, the Galerkin scheme is stable over a broad range of various flow regimes.

Chapter 6

Object-Oriented Numerics: A New Approach

6.1 Introduction

Many fields of science rely on various types of mathematical models, typically used to describe dynamic process in nature or for representation and analysis of information gathered from measured daa sets. In most application dealing with such models, computers are necessary tools in order to convert he researcher's intuition and experiences, via critical hypotheses and complicated equations, into numbers indicating success or failure. The need for doing numerical computations has strongly influenced the development .... to be completed. Today we are routinely solving problems that would be impossible to attack with he help of pen, paper and brain alone. As the performance of computers increases, we will continue to stretch the limits of what is accepted to be computationaly feasible entreprises. This thrilling development also puts extreme demands on quality management and robustness at all levels of the problem solving process. In particular, we need flexible but rigorous techniques and guidelines for software development.

Traditionaly, numerical software is based on use of procedural languages like Fortran and C. In large applications, the involved procedures are wrapped in librairies, possibly also linking to other external librairies. These principles for software development have remained more or less static over the last decades, partly because of inherent limitations of simple procedural languages. Over the last few years, there has been an increasing interest in applying the paradigms of *Object-Oriented programming* (OOP) in numerical software development[....]. This approach seems attractive due to well-defined mechanisms for moduular design, re-use of code and for creating flexible applications that can be easily extended with respect to problems classes as well as solution methods. In short, OOP encourages computer implementations of mathematical abstraction. This statement is indeed supported by practical experience made in Elligno Virtual Physics Studio and ..., respectively utilizing OOP for software development in he fields of partial differential equations and approximation of scattered data. Throughout this presentation we will use the term *Obeject-Oriented Numerics* (OON) to denote the use of object-oriented programming techniques in numerical applications. Hopefully, this chapter will reveal that OOP requires a new way of thinking about programming. At first sight, object-oriented implementations may often be more abstract and more difficult to understand than conventional codes. Usually, Fortran programmers need several months of practice before they will manage to improve the quality of their applications using OON. It is therefore natural to ask the question: Who needs object-oriented numerics? Modern scientific computing imposes two important demands on their implementation; those regarding efficiency and complexity. Efficiency typically consists of CPU-time and human time, the latter reflecting the work associated with implementation, testing and maintenance. The goal of OON is to keep the CPU-time constant while reducing the human time drastically. In addition, OON addresses complicated problems. The complexity can be on both the algotrithmic and the application side. The build-

ing blocks of OON support a step-wise software development from simple problems to more complicated problems in a unified framework. The key to this goal is the reuse of already tested software components, which is a fundamental issue in OOP. Interfacing to other software packages is also made simpler by an object-oriented design. Finally, we will mention a very important requirement of software used for research, namely flexibility in the choice of problems and methods. The real advantage of OOP will in many cases be the possibility of building flexible codes for scientific experimentation. To summarize, comprehensive and complicated physical problems or numerical algorithms are candidates for Object-Oriented implementations.

Although Fortran and C can be used to implement object-oriented designs [...], the period of development is significantly reduced when applying a language with native support for OOP. Restricting our attention to such programming languages in general, it is observed that the lack of computyational efficiency has proved to be the most important argument against object-oriented numerics. This aspect has indeed prevented serious use of OOP for numerical work until the advent of C++. This language can be seen has an extension to C, and with careful use it provides a viable alternative to Fortran and C. However, it should be emphasized that the use of certain language constructs that may be encouraged in C++ textbooks influenced by computer science, prove to be disastrous in applications that require a high level of numerical efficiency. Based on the different, and sometimes conflicting, design criteria in the fields of computer science and scientific computing, we should keep in mind that "object-oriented numerics" and "object-oriented programming" are overlapping but not identical terms. However, we emphasize that this chapter is not an introduction to OOP or C++ in general.

Abstraction and top-down design

The design process is often recognized as the most important phase of OOP. In general this process consists of identifying and grouping local entities, i.e. making useful abstractions of the problem at hand. In the context of numerics, the logical entities are often readily available in the form of mathematical abstractions. Thus, large parts of the design are already inherent in the underlying mathematical structures. In effect, numerical problems are often well suited for formulation in an object-oriented framework. This is not only true for the design of overall system structures, also at the algorithmic level concepts from OOP lend themselves naturally to the design.

Remarks

— Advantage of using of OOP approach (may be in Why OOP section?)

The concepts offered by object-oriented languages for data-hiding, inheritance and virtual functions permit a close relationship between the implementation and the underlying mathematical abstractions. Using these tools to organize and implement numerical methods as part of class hierarchies, we can derive very flexible building blocks for development of large-scale applications. A particularly attactive and user-friendly feature of such software is the ability to flexibly combine different numerical algorithms and data structures without disturbing the principal mathematical steps in the calling code with uninteresting implementational details. The resulting code is naturally divided into relatively small modules corresponding to the entities in the class hierarchies. This stuctured approach suggests that each module should be tested separately, thus promoting a high-level of robustness. Compared to traditional procedure oriented programming in Fortran or C, object-oriented programming simplifies the development of the code, increases the robustness and reliability as well as decreases the maintenance efforts dramatically. Within this group of programming languages, C++ stands out as the only competitive alternative for numerical applications due to efficiency considerations. Even using this tool, special care should be taken to avoid constructs that result in loss of efficiency or waste storage space. The main rule proves to be that algoritms expected to carry out heavy computations should be implemented as member functions of the relevant classes using low-level C functionality. The sophisticated features of C++ are the used at higher abstraction levels for implementation of application interfaces and software administration, typically by organizing calls to the low-level functions.

Object-Oriented Programming (OOP) has proven to be useful programming paradigm for complex pro-

grams, especially those modelling "real world problems." The scientific community has been slower to adopt this paradigm, but even here OOP is beginning to draw a following and even more curious interest. There are number of reasons for this reticence in the scientific community. One reason is that many scientists who write modest-sized programs for their own needs, are comfortable using Fortran 77 and C, and see no reason to change. Others with more complex programs written in Fortran have a great deal invested in their legacy codes and do not want to switch to new programming language because of the threat to this investment. Adopting OOP means not only learning a whole programming style, but learning a new and unfamiliar language as well. The dominant OO language in the scientific community, C++, is very complex and requires a substantial investment of time to learn how to use effectively. In using C++, there are also concerns about reported poor performance, lack of language and compiler standardization, and lack of standard class librairies for scientific computing. Although many of these concerns are being addressed by the C++ community, the scientific programmer may not know how to evaluate the current situation. Finally, many people have no clear idea of how their scientific productivity will improve by using OOP.

Fortran90 is modern programming language with many new features which appear to be useful for OOP.

6.2 Object-Oriented Concept

New approach for designing software ... to be completed

There are several themes underlying object-oriented technology. First of them is the concept of *abstraction* which consists of focussing on the essential, inherent aspects of an entity and ignoring its accidental properties. Subdivide the problems in concept to be reusable in other application. The second concept is *encapsulation* consists of separating the external aspects of an object, which are accessible to other objects, from the internal implementation detail of the object, which are hidden for other objects. encapsulation prevents a program from becoming so interdependent that a small change has massive ripple effects. The third concept is *inheritance* promote sharing at several different levels. Inheritance of both data structure and behavior allows common structure to be shared among several similar subclasses without redundancy. The sharing of code using inheritance is one the main advantages of Object- Oriented languages.

Object-oriented development not only allows information to be shared within an application, but also offers the prospect of reusing designs and code on future projects. One of the major justification for object-oriented technology, this development approach provides the tools, such as abstraction, encapsulation, and inheritance, to build librairies of reusable components.

6.2.1 Why OOP Approach

6.2.2 Basic of OOP

If your are not familiar with the Object-Oriented technology, you first need to understand the underlying concepts before you begin to read this chapter. You need to understand what an object is, what a class is, how objects and classes are related, and how objects communicate by using messages. The first few sections describe the concepts behind object-oriented programming. The last section shows how these concepts translate into code.

- **What is a class?** A class is a blueprint or prototype that defines the variables and the methods common to all objects of a certain kind. Classes are the fundamental building blocks in C++. A class usually represents a single concept or physical concept.
- **What is an object?** An
- **What is inheritance?**

- **What is an Interface?** An interface is a contract in the form of a collection of method and constant declarations. When a class implements an interface, it promises to implement all of the methods declared in that interface.

Relationship

- 'A-Kind-Of' relationship
- 'Is-A' relationship
- 'Part-Of' relationship
- 'Has-A' relationship

6.2.3 OOP and Scientific Programming

Numerical Solver is written in Fortran90. The C++ language is very powerful, flexible and complex. It is a language which is constantly evolving with new ideas. It is relatively poor in standard librairies and intrinsics, although that may improve with the adoption of the Standard Template Library.

Fortran90 is able to express many of the important concepts of C++, such asbstract data types, encapsulation, function overloading, and classes directly. Concept such as inheritance are not directly supported, but can be emulated.

6.2.4 Design Pattern Concept

6.2.5 UML Notation and Diagram

The Elligno Virtual Physics Studio is based on Rational's Unified Modelling Language (UML) notation. The Elligno Virtual Studio uses some minor extension (*i.e.*, extra notations) to the UML but, for the most part, the notation is UML. The UML notation follows the object-oriented methodology for organizing data.

Class Notation

Central to the UML notation is the concept of a class data. A class is an abstract, user-defined description of a type of data. It identifies the attributes of the data and the operations that can be performed on instances (*i.e.*, objects) of the data. This modelling technique allows description of a system using the same terminology as the corresponding real-world objects and their associated characteristics. In Elligno Virtual Studio, the data representation model contains the classes necessary for the representation

6.2.6 Current and Future

My point of View on the future of scientific Software.

6.3 Elligno Virtual Physics Studio

In this section we present the software developed in this thesis. It use the most modern concept in software development.

6.4 Open Architecture Approach

The requirement for open computing architectures is driving the next generation of constructive software development. Such architectures provide for ease of integration of different software components. The software components exist as librairies that are accessed by the user at the simulation build phase. In our case, typical models could include various numerical schemes for treating the mathematical terms in flow equations (convective, pressure, diffusive etc...).

6.4.1 Application Program Interface: API

6.4.2 Type Hierarchy

6.4.3 Math API

Elligno Virtual Physics Studio includes a linear algebra and miscellaneous math library that can be leveraged in your application software. Basic types, vector, and matrix manipulation functions are the heart of the math API (numerical simulation). These types must be compliant with the Numerical Recipes (programming scientific). These recipes will be used thoroughly in a numerical simulation.

Basic Types

Aggregate Types

Vector Types

6.4.4 Framework Concept

Refrence: article "LaSRS++ an Object-Oriented Framework for Real-Time Simulation of Aircraft" chemise Articles Software 2001

Frameworks represent a collection of classes that provide a set of services for a particular domain; a framework exports a number of individual classes and mechanisms which clients can use or adapt. In this we present an overview of an Object-Oriented (OO) framework. The abstractions used and the benefits of object technology in this environment will be discussed.

Key OO Concepts Utilized In Framework

Four basic OO concepts utilized in the Elligno Virtual Physics Studio framework are:

1. Abstraction
2. Encapsulation/Containment
3. Inheritance
4. Polymorphism

6.4.5 Plug-In Concept

Plug-In offer immense flexibility in customizing Elligno tools and applications. Plug-Ins can be loaded by an executable that was linked with them directly, or they can be loaded or unloaded dynamically at any time. They allow existing applications to be extended with your code. They also provide a convenient method for third parties to create extensions to Elligno Tools Kit.

Appendix A

Object-Oriented Technology

A.1 Introduction to Object-Oriented Concepts

The building block of OOP support a step-wise software development from simple problems to more complicated problems by identifying concepts of the problem at hand. The task is to develop computational "objects" that represent fundamental abstractions of elements in a computational model. The key to this goal is the reuse in other applications of already tested components, which is a fundamental issue in OO. This result is an implementation that is manageable, extensible, and easily modified.

Then what is OO software? The most important concept is the *object*, which represents a relevant abstraction either from the real world or from the implementation domain. The conceptual similarity between objects in the software model and real-world objects is a key to explain the success of OO software construction. This is enhanced by program constructs which model relationship between objects, for instance that one object is an *aggregate* [1] of several other objects.

To achieve this, OO uses three main themes: encapsulation, inheritance and polymorphism. The former, means that software is organized into objects that store both data and operations on data. By encapsulating the data and operations together isolates the classes and promotes the reuse of code. It prevents a program from being so interdependent that a small change has a massive ripple effects. Changes to a class affect only the class in question. It allows the details of the implementation of the object to be hidden, and thus easily modified. Code reuse is further enhanced by inheritance, that consists of putting abstractions in a hierarchy share behavior through attributes and operators that are common to several subclasses into superclasses, which is implemented once for all. Interdependencies between the classes are explicitly laid out in the class interfaces. Finally, polymorphism allows the same operation to behave differently in different classes and thus allows objects of one class to be used in place of those of another related class.

The code uses inheritance to group similar data structures and procedures into families that share some elements and that are, in appropriate contexts, interchangeable.

The object-Oriented philosophy consists of identifying and grouping local entities i.e. making useful abstractions of the problem at hand. The Object-Oriented paradigm provides four fundamental concepts: *objects*, *classes*, *inheritance*, and *polymorphism*. Software is organized into *objects*, that store both its data and operators on the data. This permits developers to abstract out the essential properties of an object, those that will be used by the others objects. This abstraction allows the details of the implementation of the object to be hidden, and thus easily modified. Objects are instances described by a class definition. *Classes* are related by *inheritance*. A *subclass* inherits behavior through the attributes and operators of the superclass. *Polymorphism*, allows the same operation to behave differently in different classes and thus

[1] see Terminology section for the definition

allows objects of one class to be used in place of those of another related class.

A.2 Definition

Definition A.2.1 (Virtual)**.** *A virtual function is a member function of a class, whose functionality can be over-ridden in its derived classes. It is one that is declared as virtual in the base class using the virtual keyword. The virtual nature is inherited in the subsequent derived classes and the virtual keyword need not be re-stated there. The whole function body can be replaced with a new set of implementation in the derived class.*

Definition A.2.2 (Polymorphism)**.** *???*

Definition A.2.3 (Singleton Design Pattern)**.** *Guarantee only one instance of the class. The implementation as the following: first declare a static attribute to the class (mySingleton). Default and copy constructor are declared protected (doesnt allow creation by external), only one instance of this class is allowed. Declare a method static that create the singleton instance if it has not been created, otherwise return the instance.*

Definition A.2.4 (Iterator)**.** *An STL iterator is a generalization of a pointer. Iterators provide a clean, memory-safe method of traversing STL data structures. Iterators have the added advantage over pointers of being type generic. An STL iterator can be used on most of the STL's data structures, and most STL algorithms (sorting, searching, and so on) are designed to work with iterators. Also, memory access using iterators is protected, so you can't move past the end of a list and so on, further helping to reduce errors in code.*

Definition A.2.5 (Pointer)**.** *For a type T, T* is the pointer to T. A variable of type T*, can hold the address or location in memory of an object of type T.*

Definition A.2.6 (Reference)**.** *is an alternative name or an alias for an object. Since reference is an alternative name for an object, a reference must be initialized and can not be changed to refer to another object. The man use of references is in specifying arguments and return values for functions and overloaded operators.*

Definition A.2.7 (Smart Pointer)**.** *smart pointer are simple C++ classes which maintains reference counts in an object when the handle is created, copied or destroyed. The last handle to reference an object will destroy the object and release resources. We control de-allocation of the shared data structures by maintaining reference counts to the dataflow data. we maintains these reference counts automatically through the use of handles or smart pointer.*

Definition A.2.8 (Binding)**.** *Binding refers to the act of associating an object or a class with its member. If we can call a method fn() on an object O of a class C, we say that the object O is binded with the method fn(). This happens at compile time and is known as static or compile - time binding. The calls to the virtual member functions are resolved during run-time. This mechanism is known as dynamic binding. The most prominent reason why a virtual function will be used is to have a different functionality in the derived class. The difference between a non-virtual member function and a virtual member function is, the non-virtual member functions are resolved at compile time.*

Smart Pointer In the case of a scalarField we use handles ... because grid is shared by instance of a scalarField if the scalarField is destroy we dont want to destroy the grid, because it may be used by another scalarField.

Appendix B

Terminology

A number of common terms, that are frequently used in hydraulics that are necessary for comprehension are defined:

gravity wave the free surface of a liquid in equilibrum in a gravitational field is a plane. If, under the action of some external perturbation, the surface is moved from its equilibrum position at some point, motion will occur in the liquid. This motion will be propagated over the whole surface in the form of waves, which are called it gravity waves, since they are due to the action of the gravitational field. Gravity waves appear mainly on the surface of the liquid; they affect the interior also, but less and less at greater and greater depths. If the wave lenght is greater than the water depth ($\lambda \gg h$) where λ is the wave lenght and h water depth, these waves are called celerity waves. These waves appear in open channel flow and the velocity of such waves is given by: $v = \sqrt{gh}$

density the density $\rho(kg - sec^2/m^4)$ of a fluid or solid is the mass that it possesses per unit volume

viscosity viscosity is the proprerty of a fluid that resists relative motion and deformation in the fluid and causes inetrnal shear. Therefore, viscosity is property exhibited only under dynamic conditions. Acording to Newton, the shear τ at a point within a fluid is proportional to the velocity gradient du/dx at that point or

$$\tau = \mu \frac{\partial u}{\partial x} \tag{B.1}$$

wher μ in $kg - sec/m^2$, is the dynamic viscosity. When divided by the density ρ, it is the kinematic viscosity $\mu/\rho = \nu$ in m^2/sec. Under ordinary conditions of pressure, viscosity varies only with temperature. The viscosity of a liquid decreases with increasing temperature, the reverse is true for gases.

discharge the discharge Q is the volume of a fluid or solid passing a cross section of a stream per unit time

cross-sectional area the cross-sectional area A is the area of a cross section of the flow normal to the direction of flow

wetted perimeter the wetted perimeter P is the length of wetted cross section normal to the direction of flow

hydraulic radius the hydraulic radius R is the ratio of the cross-sectional area to wetted perimeter, $R = A/P$

water depth the depth of the flow h is the vertical distance from the bed of a stream to the water surface

energy line the energy line E' is defined as the total head with respect to the reference level

$$E' = H + \frac{U^2 + V^2}{2g} \tag{B.2}$$

where

$$\begin{aligned} H &= \text{water level} \\ \frac{U^2 + V^2}{2g} &= \text{velocity head} \end{aligned} \tag{B.3}$$

slope of the energy grade line the energy grade line is a graphical representation with respect to a selected datum, of the total head or energy possessed by the fluid. For open channel the energy gradient is located a distance $U^2/2g$ above the free water surface. The slope of the energy grade line is designated by the symbol S_f. The slope of energy line is calculated using the Manning formula:

$$S = \frac{N^2 \sqrt{U^2 + V^2}}{h^{4/3}} \tag{B.4}$$

where

$$\begin{aligned} N &= \text{Manning coefficient} \\ \sqrt{U^2 + V^2} &= \text{modulus of water velocity} \\ h &= \text{water depth} \end{aligned} \tag{B.5}$$

Froude number The Froude number is

$$F_r = \frac{U}{\sqrt{gL}} \tag{B.6}$$

where g is the gravitational acceleration, U is the mean velocity and L a characteristic length. The Froude number relates the inertia forces to the gravitational effects and is important wherever the gravity effect is dominating, such as with water waves, flow in open channels, sedimentation in lakes and reservoirs, salt-water intrusions, and the mixing of air masses of specific weights. For open channel flow $L = h$.

Reynolds number the Reynolds number is

$$Re = \frac{\rho U L}{\mu} = \frac{U L}{\nu} \tag{B.7}$$

where U is the velocity, L is the length and ν is the kinematic viscosity. The Reynolds number relates the inertia forces to the viscous forces and is usually involved wherever viscosity is important, such as in slow movement of fluid in small passages or around an small objects.

Appendix C

Classification of Open Channel Flow

Open channel flow can be classified into many types and described in various way. The following classification considers change in velocity with respect to the time and space.

1. **steady flow** The flow is steady if the velocity ata point does not vary in magnitude or direction with time. The mathematical expression of this statement is $\frac{\partial u}{\partial t} = 0$ where ∂u represents the variation of the velocity at a given point with time, ∂t.
2. **unsteady flow** the flow is unsteady if the velocity at a point varies with time, $\frac{\partial u}{\partial t} \neq 0$.

In summary open-channel flow is classified as:

- Steady flow
    1. Uniform flow (rare in alluvial channels)
    2. Varied flow
        a. Gradually varied flow
        b. Rapidly varied flow
- Unsteady flow
    1. Unsteady uniform flow
    2. Unsteady flow
        a. Gradually varied unsteady flow
        b. Rapidly varied unsteady flow

Figure ? illustrates steady flow phenomena common in engineering practice. Upstream of the dam the backwater curve illustrates gradually varied subcritical flow. Gradually varied, supercritical flow occurs downstream of the spillway. Supercritical uniform flow may occur in the canal downstream of the spillway if the canal is long enough and the hydraulic jump illustrates rapidly varied flow.

Again referring to figure ? the two solitary waves illustrate rapidly-varied, unsteady flow. When a gate suddenly closes in a uniform flow, solitary waves are generated (see Fig. ?). The variation of the discharge, depth of flow, etc., are gradually varied flow phenomena that may occur during a flood. Figure ? illustrate this phenomena.

- STATE OF FLOW Open channel flow is governed to a large degree by viscosity, gravity and inertial forces.

A. **effect of viscosity** The flow may be laminar, turbulent, or transitional depending on viscosity relative to inertia.

(a) Laminar flow: the flow is laminar if the viscous forces are so strong relative to the inertia forces that the viscosity palys a significant part in determining flow behavior. The Reynolds number is the ratio of inertial to viscous forces and

$$Re = \frac{UL}{\nu} \tag{C.1}$$

where U is the mean velocity of flow, L is a characteristic length and ν is the kinematic viscosity of the fluid.

(b) Turbulent flow: the flow is turbulent if the viscous forces are weak relative to the inertial forces. With turbulent flow the fluid particles move in irregular paths which are neither smooth nor fixed but still represent the forward motion of the fluid.
For the analysis of turbulent flow the shear stress is defined by the relation

$$\tau = \eta \frac{du^2}{dy} \tag{C.2}$$

where η is a turbulent mixing coefficient, corresponding to the dynamic viscosity, μ, in laminar flow and is, therefore, often called 'dynamic', 'virtual', or 'eddy' viscosity. The eddy viscosity, η, is not a property of the fluid like μ, but depends on the velocity u.

(c) Transitional flow: between laminar and turbulent flow there is a mixed, or transitional state. The value of the Reynolds number is usually between 500 and 2500.
NOTE: with free surface flow the limits defining laminar and turbulent flow are slightly different.

B. **effect of gravity** the flow may be subcritical (tranquil), supercritical (rapid), or critical depending on the effect of gravity relative to inertia.

(a) Subcritical flow or Tranquil flow: in this state, the role played by gravity forces is pronounced, so that the flow has a low velocity. The Froude number defined by the equation below relates the inertial forces to the gravity forces and is

$$F_r = \frac{U}{\sqrt{gL}} \tag{C.3}$$

where L is a characteristic length of the flow. For subcritical flow $U < \sqrt{gL}$ and $F_r < 1$. In the mechanics of the water waves, the critical velocity $\sqrt{gL}$ is identified as the celerity of a shallow water waves; in this case L is equal to the depth of the flow. Such a wave may be created by disturbances or obstacles in the channel that causes a displacement of water above or below the mean water surface level. Because this wave celerity is greater than the mean velocity of the flow, the Shallow water wave can move upstream. More specifically if a rock is thrown into a subcritical flow the wave generated by the disturbance will move upstream because the surface waves have a celerity greater than the velocity of the flow i the channel.

(b) Supercritical flow or Rapid flow: in this state the inertial forces are dominant and the flow has a high velocity. THe Froude number is greater than unity. Under this condition, the Shallow water waves are smaller than the velocity of the flow and they can only move downstream.

(c) Critical flow: the flow is critical when the Froude number is equal to unity. Under this condition, a shallow water wave remains approximately stationary. The accompanying depth is called critical depth.

A hydrodynamic analysis of a body of water consists in avaluating the velocity, pressure and level of the warter under various conditions. The main varibles to be analysed are listed in table ?

u, v, w	Velocity components as a function of $(x, y, z)(m/s)$
$x, y, z; t$	System of spatiotemporal reference coordinates $(m; s)$
$p, \rho(x, y, z, t)$	Pressure and density $(N/m^2; kg/m^3)$
$h(x, y, t)$	Instantanous depth (m)

Among these, $u, v, w, p$and$h$ are the dependent variables that constitute the unknowns of the problem.

The physical characteristics of the fluid are:

$$\begin{aligned} \nu &= \text{kinematic viscosity}(m^2/s) \\ \mu &= \text{dynamic viscosity}(kg/m.s) \\ \mu = \rho\nu & \end{aligned}$$

The various forces acting on open channel flow are:

- gravitational
- inertial
- viscous

Two of the parameters required for classifying flow involve a comparison of the relative vallues of the forces identified above. Thus, we define dimensioless numbers, the value of which wil reflect the relative importance of the forces involved. The numbers normally used are the Reynolds and Froude numbers. The Reynolds number is expressed as follows:

$$Re = \frac{\text{inertial forces}}{\text{viscous forces}} = \frac{Uh}{\nu} \tag{C.4}$$

An increasing Reynolds number implies a decreasing influence of viscous forces on the flow. Under natural conditionss, a flow is termed turbulent when the inertial forces dominate over viscous forces.

The Froude number is a dimensionless number of great importance in open-channel flow. It is expressed as follows:

$$F_r = \frac{\text{inertial forces}}{\text{gravitational forces}} = \frac{U}{\sqrt{gh}} \tag{C.5}$$

where:

$$\begin{aligned} g &= \text{acceleration of gravity} \\ h &= \text{a characteristic dimension, such as the water depth} \end{aligned}$$

This number plays the same role as the Mach number (Ma) used in aerodynamics for compressible-fluid flow:

$$\frac{Ma = U}{\sqrt{K_c/\rho}} \tag{C.6}$$

where:

$$\begin{aligned} K_c &= \text{compressibility, in} N/m^2 \\ \rho &= \text{density, in} kg/m^3 \end{aligned}$$

In addition, $\sqrt{gh}$ represents the propagation speed or celerity c of a long wave in Shallow water, just as $\sqrt{K_c/\rho}$ represents the speed of sound in air.

When the Froude number is less than 1 ($F_r < 1$), *i.e.* when the fluid velocity is less than the celerity of a surface wave, the flow is termed **subcritical** or **fluvial**. Under these conditions, small waves may propagate upstream.

When the Froude number is higher than 1 ($F_r > 1$), *i.e.* when the fluid velocity exceeds the celerity of the gravity wave, the flow is described as being **supercritical** or **torrential**. Under these conditions, an upstream disturbance propagates rapidly downstream and may cause shocks and instabilities. The downstream conditions will not have any perceptible effect on the upstream flow behavior.

The transition from a torrential to a fluvial flow involves the formation of a hydraulic jump but transition from a fluvial to a torrential flow will occur without any discontinuity.

Another parameter affecting the flow is roughness, or irregularities of the wall surface. Resistance to flow is a function of the nature of the wall surface and the bottom or bed configuration. Generally, a parameter κ is used to represent the average size (in mm or cm) of the surface asperities.

The Manning coefficient N is an empirical measure of the effect of roughness. In contrast, κ is solely a characteristic length representing elements of the wall surface that is subsequently used to classify various types of walls.

Finally, a flow regime may be classified according to its variability as a function of time. Steady flow may result of unchanging boundary conditions (river, canal, reservoir, etc.). Unsteady flow occurs when the imposed boundary conditions (tides, draining of a reservoir, etc) vary with respect to time.

In summary, the flow regime may be classified according to the various elements of the environment and the fluid to be modeled, as shown in the table below.

Appendix D

Numerical Method

D.1 Introduction

In the following section, we review various techniques proposed in the litterature to solve the St-Venant equations numerically. The two ..... important tools for the space and time discretization of differential operators are the following:

1. Finite difference
2. Finite element

These methods are powerful and play a major role in problem solutions. In this section we attempt to present the fundamental insight to these methods. We begin by dicussing the concept of discrete pointwise approximation of functions.

I would like to mention that this cover only a small part of the whole theory. It is not our intention here to enter into these mathematical aspects and the reader will find these detailed developments in the references mentioned.

D.2 Finite Difference

Until recently, numerical methods for solving fluid-flow problem have been dominated by finite-difference approximations. The idea of finite difference methods is actually quite simple, since it corresponds to an estimation of a derivative by the ratio of two differences according to the definition of the derivative. This approach is based on Taylor series expansion of the differential operator. The derivative is approximated by divided difference. The basic idea of the method is the following.

Let $f(x)$ be a function defined in the range $a \leq x \leq b$. The interval [a,b] is discretized by considering the set $x_0 = a, x_1 \ldots, x_i, \ldots, x_{N+1} = b$, and the discrete representation of $f(x)$ is the set $f(a), f(x_1), \ldots, f(x_i), \ldots, f(b)$. Generally, the value $f(x_i)$ is denoted by f_i. When $f(x)$ is known as a solution of some mathematical problem, say the the solution of a differential equation, the values $f(x_i)$ are not known exactly but are the result of some approximation and, in this case, f_i is a discrete approximation of $f(x)$ and we note $f_i \cong f(x_i)$.

The quantity $(x_{i+1} - x_i)$ is the mesh size and we shall assume, for the sake of simplicity, that this mesh size is constant: $\Delta x = (b-a)/(N+1)$ and $x_i = a + i\Delta x, i = 0, \ldots, N+1$.

Consider the definition of a derivative

$$u_x \equiv \left(\frac{\partial u}{\partial x}\right) = \lim_{\Delta x \to 0} \frac{u(x + \Delta x) - u(x)}{\Delta x} \tag{D.1}$$

If Δx is small but finite the expression on the right-hand side is an approximation to the exact value of u_x. The approximation will be improved by reducing Δx, but for any finite value of Δx an error (the *truncation* error) is introduced which goes to zero for Δx tending to zero. The power of Δx with which this error tends to zero is called *the order of the difference approximation*, and can be obtained from a Taylor series development of $u(x + \Delta x)$ around point x. Actually, the whole concept of finite difference approximations is based on the properties of Taylor expansions. Developing $u(x + \Delta x)$ we obtain

To obtain an estimate of the error of the derivative we proceed by a Taylor expansion

$$u(x + \Delta x) = u(x) + \Delta x \; u_x(x) + \frac{\Delta x^2}{2!} \; u_{xx}(x) + \frac{\Delta x^3}{3!} \; u_{xxx}(x) + \ldots \tag{D.2}$$

and therefore, to the highest order in Δx,

$$\frac{u(x + \Delta x) - u(x)}{\Delta x} = u_x(x) + \frac{\Delta x}{2} u_x x(x) + \ldots \tag{D.3}$$

This approximation for $u_x(x)$ is said to be first order in Δx, and we write

$$\frac{u(x + \Delta x) - u(x)}{\Delta x} = u_x(x) + \mathcal{O}(\Delta x) \tag{D.4}$$

indicating that the truncation error $\mathcal{O}(\Delta x)$ goes to zero like the first power in Δx.

The mth derivative of $f(x)$ at point x_i is approximated in the form

$$\frac{d^m f(x_i)}{dx^m} \cong \sum_{j=-J_1}^{j=J_2} \alpha_j f_{i+j} \tag{D.5}$$

where the α_j's are determined by means of a Taylor expansion of f_{i+j} around x_i and J_1, J_2 are integers depending on the order m of the considered derivative and also on the degree of accuracy of the approximation. If for $m = 1$ we consider an approximation using three values of f_i *i.e.*, $J_1 + J_2 = 2$, and we take $J_1 = J_2 = 1$, we can write a general expression

$$\frac{df(x_i)}{dx} \cong \frac{(1 - \alpha) \; f_{i+1} + 2\alpha f_i - (1 + \alpha) \; f_{i-1}}{2\Delta x} \tag{D.6}$$

where α is an arbitrary constant. The error of the approximation is

$$-\frac{\alpha \Delta x}{2} \frac{d^2 f}{dx^2} \; - \frac{\Delta x^2}{6} \frac{d^3 f}{dx^3} + \mathcal{O}(\Delta x^3) \tag{D.7}$$

The type of finite difference which is used to replace the partial derivatives can be selected from a number of different forms, depending on the desired accuracy of the solution, convergence behavior, stability, and convenience. However, the most common forms in current use are *forward, backward and central differences*, all of which stem from Taylor's series.

A very large number of finite difference approximations can be obtained for the derivatives of functions. By specifying the value of α we obtain the standard differences:

Centered: $\alpha = 0$

$$\frac{df(x_i)}{dx} \cong \frac{f_{i+1} - f_{i-1}}{2\Delta x} \equiv \Delta_x^0 f_i, \qquad \text{error} = \mathcal{O}(\Delta x^2) \tag{D.8}$$

Backward: $\alpha = 1$

$$\frac{df(x_i)}{dx} \cong \frac{f_i - f_{i-1}}{\Delta x} \equiv \Delta_x^- f_i, \quad \text{error} = \mathcal{O}(\Delta x) \tag{D.9}$$

Forward: $\alpha = -1$

$$\frac{df(x_i)}{dx} \cong \frac{f_{i+1} - f_i}{\Delta x} \equiv \Delta_x^+ f_i, \quad \text{error} = \mathcal{O}(\Delta x) \tag{D.10}$$

Now, if we choose $J_1 = 2$ and $J_2 = 0$ and employ a Taylor series to find the α's we obtain the second-order accurate approximation.

$$\frac{df(x_i)}{dx} \cong \frac{1}{2\Delta x}(3f_i - 4f_{i-1} + f_{i+2}) \tag{D.11}$$

Similarly, if $J_1 = 0, J_2 = 2$, we obtain the second-order accurate approximation

$$\frac{df(x_i)}{dx} \cong \frac{1}{2\Delta x}(-3f_i + 4f_{i+1} - f_{i+2}) \tag{D.12}$$

A fourth-order accurate result is obtained if $J_1 = J_2 = 2$, *i.e.*,

$$\frac{df(x_i)}{dx} \cong \frac{-f_{i+2} + 8f_{i+1} - 8f_{i-1} + f_{i-2}}{12\Delta x} \tag{D.13}$$

In the same way, we can define approximations for all the derivatives. For example, the classical difference approximation of the second derivative is

$$\frac{d^2 f(x_i)}{dx^2} \cong \frac{f_{i+1} - 2f_i + f_{i-1}}{\Delta x^2} \equiv \Delta_{xx} f_i \tag{D.14}$$

which is accurate to $\mathcal{O}(\Delta x^2)$. The use of the difference operators $\Delta_x^0, \Delta_x^+, \Delta_{xx} \ldots$ is very useful, and it is easy to verify the following relationship:

$$\begin{aligned} \frac{1}{2}(\Delta_x^+ + \Delta_x^-) &= \Delta_x^0 \\ \Delta_x^+ - \Delta_x^- &= \Delta x \Delta_{xx} \\ \Delta_x^+ \Delta_x^- &= \Delta_{xx} \end{aligned} \tag{D.15}$$

However, note that $\Delta_x \Delta_x \neq \Delta_{xx}$ because

$$\Delta_x \Delta_x f_i = \frac{f_{i+2} - 2f_i + f_{i-2}}{4\Delta x^2} \tag{D.16}$$

When these difference quotients are used to replace the partial differentials in an equation, then a *difference equation* results. For example, consider the continuity equation given below

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{V}) = 0 \tag{D.17}$$

For steady, two-dimensional flow, equation (?) becomes

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho v)}{\partial t} = 0 \tag{D.18}$$

Defining $F = \rho u$ and $G = \rho v$, equation (?) is written as

$$\frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0 \tag{D.19}$$

Replacing the x derivative with a forward difference [Eq.()], and the y derivative with a central difference [Eq.()], we have

$$\frac{F_{i+1,j} - F_{i,j}}{\Delta x} = \frac{G_{i,j+1} - G_{i,j-1}}{2\Delta y} \tag{D.20}$$

or

$$F_{i+1,j} = F_{i,j} + \frac{\Delta x}{2\Delta y} G_{i,j+1} - G_{i,j-1} \tag{D.21}$$

Equation (?), or equation (?), is the *difference equation* that replaces the original partial differential equation, namely Eq. (?). Equation (?) is an approximation for Eq. (?); Eq. (?) contains a truncation error which is a combination of the truncation errors from the difference quotients in (5.14) and (5.16).

The interval (0,1) is discretized with $x_i = i\Delta x, i = 0, \ldots, N+1$, with $\Delta x = 1/(N+1)$. If we denote by f_i the numerical approximation of $f(x_i)$ at point x_i, a finite difference approximation of (?) is .....

The use of centered differences leads to an error of second order with respect to the mesh size Δx. Equation (?) produces a linear algebraic system whose solution yields the N values of f_i. In the present case the associated matrix is tridiagonal, so it is very efficient to use a direct method of solution. The general method of factorization leads to a simple solution algorithm described .....

A distinction between various finite-difference solutions is that of explicit versus implicit approaches. .....

The advantage of explicit methods is that they are relatively simple to set up and program. In explicit methods the matrix of the unknown variables at the new time is a diagonal matrix , while the right-hand side of the system is dependent only of the flow variables at the previous times. This leads therefore to a trivial matrix inversion and hence to a solution with a minimal number of arithmetic operations for each time step. The disadvantage is that the spatial increments Δx and δy are limited due to stability constrains associated with explicit methods. For a given Δx, δy is constrained to be less than a certain value dictated by numerical stability considerations. In turn, if Δx is constrained to be small, the computer time required to calculate the flow over a prescribed downstream distance can be large.

The advantage of implicit methods is that stability can be maintained over much larger values of Δx, hence using considerably fewer steps to make calculations over a prescribed downstream distance. In implicit methods the matrix to be inverted is not diagonal, since more than one set of variables are unknown at the same time level. In most cases, however, the structure of the matrix will be rather simple, such as block pentadiagonal, block tridiagonal or block bidiagonal, allowing simple algorithms for the solution of the system at each time step, although the number of operations required will be higher when compared with explicit methods. A disadvantage of implicit methods is that they are more complicated to set up and program in comparison to explicit methods. Moreover, massive matrix manipulations are usually required at each spatial step to solve the simultaneous algebraic equations, hence the computer time per step is larger for the implicit approach.

D.3 Numerical Analysis

Coming in the next version. References are Strickwerda and Le Pourhiet.

Any numerical scheme must satisfy these three requirements if we want our approximation be suitable to the solution we want to approximate. These notions are really important and i will cover only the most important points. The reader can find details in the references listed in the bibliography. Before we admit that the approximate solution of the numerical scheme is valid, one must satisfy the following criteria:

1. consistency of the scheme
2. stability of the numerical scheme
3. convergence towards the exact solution

I will follow the approach given in [46] which is the Fourier analysis.

For a function $u(x)$ defined on the real line R, its Fourier transform $\hat{u}(x)$ is defined by

$$\hat{u}(\omega) = \frac{1}{sqrt2\pi} \int_{-\infty}^{\infty} e^{-i\omega x} u(x) \, \mathrm{d}x \tag{D.22}$$

The Fourier transform of u is a function of the real variable ω and is uniquely defined by u. The function $\hat{u}u$ is an alternative representation of the function u. Information about certain properties of u can be inferred from the properties of $\hat{u}$. The Fourier inversion formula, given by

$$u(x) = \frac{1}{\sqrt{2\pi}} \int_{-\infty}^{\infty} e^{i\omega x} \hat{u}(\omega) \, \mathrm{d}\omega \tag{D.23}$$

shows how u can be recovered from $\hat{u}$. The Fourier inversion formula expresses the fuunction u as a superposition of waves, given by $e^{i\omega x}$, with different amplitudes $\hat{u}(\omega)$.

In a similar fashion, if v is a grid function defined for all integers m, its Fourier transform is given by

$$\hat{v}(\psi) = \frac{1}{\sqrt{2\pi}} \sum_{m=-\infty}^{\infty} e^{-in\xi} v_m \tag{D.24}$$

for $\psi \in [-\pi, \pi]$, and $\hat{v}(-\pi) = \hat{v}(\pi)$. The Fourier inversion formula is given by

$$v_m = \frac{1}{\sqrt{2\pi}} \int_{-\pi}^{\pi} e^{im\xi} \hat{v}(\xi) \mathrm{d}\psi \tag{D.25}$$

D.4 Finite Element

The finite element concept

The finite element method is concerned with the solution of mathematical or physical problems which are usually defined in a continuous domain either by local differential equations or by equivalent global statements. To render the problem amenable to numerical treatment, the infinite degrees of freedom of the system are *discretized* or replaced by a finite number of unknown parameters, as indeed is the practice in other processes of approximation.

The original "finite element" concept replaces the continuum by a number of subdomains (or elements) whose behaviour is modelled adequately by a limited number of degrees of freedom and which are *assembled* by processes well known in the analysis of discrete systems.

The finite element method is one of the most popular method in numerical simulation in the last few decade. The capability to manage complex geometry ... using high order interpolation polynomial. The method has been developped in the field of mechanical engineer based on law to be completed.

The space domain can be discretized by subdivision of the continuum into elements of arbitrary shape and size. Since any polygonal structure with rectilinear or curved sides can finally be reduced to triangular and quadrilateral figures the latter are the basis for space subdivision. The only restriction is that the elements may not overlap and have cover the complete computational domain. Within each element a certain number of points are defined, which can be positioned along the straight (or curved) sides or inside the element. These nodes will be the points where the numerical value of the unknown functions, and eventually their derivatives, will have to be determined. The total number of unknowns at the nodes, function values and eventually their derivatives are called the *degrees of freedom* of the numerical problem, or *nodal values*.

The finite element method can be characterized by three features:

1. the domain of the problem is represented by a collection of simple domains, called *finite elements*. The collection of finite elements is called *finite element mesh*

2. over each finite element, the physical process is approximated by functions of desired type (polynomials or otherwise), and a algebraic equations relating physical quantities at selective points, called *nodes*, of the element are developped

3. the element equations are assembled using continuity and/or 'balance' of physical quantities

Thus we define the finite element process as any approximation process in which

- the behaviour of the whole system is *approximated* to by a finite number,n, of parameters, $a_i = i = 1, \ldots, n$
- the n-equations governing the behaviour of the whole system, *i.e.*

$$F_j(a_i) = 0, \qquad j = 1, \ldots, n \tag{D.26}$$

can be assembled by the simple process of addition of terms contributed from all subdomains (or elements) which divide the system into physically identifiable entities (without overlap or exclusion). Thus,

$$F_j = \sum F_j^e \tag{D.27}$$

where F_j^e is the element contribution to the quantity under consideration.

What are the procedures by which a finite element approximation can be made? We will concentrate on any problem which can be defined mathematically either by a (set) of differential equations. Let us consider the time-independent equation

$$\mathcal{D}(\mathbf{u}) = f, \qquad \text{in } \Omega \tag{D.28}$$

where Ω is a bounded domain and the differential operator (non-linear[1]) $\mathcal{D}$ acts on the unknown function u to generate the known function f. The function u is not only required to satisfy the operator equation (above), it is also required to satisfy the boundary conditions associated with operator equation.

together with their associated boundary conditions on boundaries (Γ) of the domain

$$\mathbf{B}(\mathbf{u}) = 0 \tag{D.29}$$

In both statements u represents either the single unknown function or a set of unknown functions.

Steps involved in Finite Element Analysis

1. **Discretization and interpolation functions**

   To obtain a finite element approximation to the general problem defined by equations (?) to (?) we proceed as follows: –the unknown function is expanded in a finite set of assumed, known trial functions ϕ_j and unknown parameters c_j, *i.e.*

   In the weighted-residual method, the solution u is approximated by the expression

$$u_N = \phi_1(x) + \sum_{j=2}^{M} c_j \phi_j(x) \tag{D.30}$$

[1]for an operator A to be linear in its arguments, it must satisfy the relation $A(\alpha u + \beta v) = \alpha A(u) + \beta A(u)$ for any scalars α and β and dependent variables u and v. When an operator does not satisfy this condition, it is said to be non-linear

Generally in the finite element method, the term $\phi_1(x)$ is not written explicitly but is incorporated in the series through a specification on the c_j, Eq. (above) becomes

$$u_N = \sum_{j=1}^{M} c_j \phi_j(x) \tag{D.31}$$

It should be recalled

$$u(x) \approx U^e = \sum_{j=1}^{n} u_j^e \ \Psi_j^e(x) \tag{D.32}$$

where $\Psi_j^e(x)$ are interpolation polynomials.

$$\Psi_i^e(x_j) = \delta_{ij} \sum_{i=1}^{n} \Psi_i^e(x) = 1 \tag{D.33}$$

Let a residual $\mathcal{R}$ be defined by

$$\mathcal{R}(x) = \mathcal{D}(u_N) - f \tag{D.34}$$

Direct substitution of such approximation into the governing equations does not always result, for an arbitrary choice of the data of the problem, in a necessary and sufficient number of equations for the undetermined coefficients c_j. We can write the equations

$$\mathcal{R}(x) = \mathcal{D}\left[\sum_{j=1}^{M} c_j \phi_j(x)\right] - f \tag{D.35}$$

If the trial solution were exact solution, the residual would vanish.

Therefore, a procedure whereby a necessary and sufficient number of equations can be obtained is needed. One such procedure is provided by a weighted-integral form of the governing differential equation.

2. **Approximation Integrals**

This is the most essential step of the finite element approximation since it requires the definition of an **integral** formulation of the physical problem equivalent to the field equations to be solved. Two possibilities are open for that purpose: either a *variational principle* can be found expressing the physical problem as the extremum of a functional or a an integral is obtained from the differential system through a *weak* formulation, also called the *method of weighted residuals*.

The integrand is automatically zero when U is replaced by its approximation. Mathematically, (?) is statement that the error in the differential equation (due to the approximation of the solution) is zero in the weighted-integral sense.

A. **Variational principles**

If the problem is stated in terms of a stationary functional Π then the formulation is most direct. We can write the approximate form of the functional as

$$\Pi = \hat{\Pi} = \Pi(\hat{\phi}) \tag{D.36}$$

and for stationarity we have a set of equations

$$\mathbf{F}_j \equiv \frac{\partial \Pi}{\partial \mathbf{a}_j} = 0 \tag{D.37}$$

which by definition of Π is already cast in an integral form.

A variational principle requiring stationary (max.,min., or "saddle") of some scalar functional Π

$$\Pi = \int_\Omega G(\phi)\ d\Omega + \int_\Gamma g(\phi)\ d\Gamma \tag{D.38}$$

This basis of forming a finite element approximation has been and remains most popular, *providing* a physically meaningful variational principle exists and can be readily identified. Although many physical models can be expressed through a variational equation it is well known that it is not always possible to find a straightforward variational principle for all physical problems (for instance, for the Navier-Stokes equations). Therefore the weak formulation, or method of weighted residuals, is the most technique which allows us to define in all cases an equivalent integral formulation. Actually, in situations where discontinuous solutions are possible (such as shock waves in transonic flows) the integral formulation is the only one which is physically meaningful, since the derivatives of the discontinuous flow variables are not defined. The important question of how to proceed from the differential equation directly in cases where a variational principle does not exist or cannot be identified remains. The answer to it lies in the reformulation by use of weighting function.

B. **Weighted Integral Formulation**

Recall that one can always write the weighted-integral form of a differential equation, whether the equation is linear or non-linear (in the dependent variables). The weak form can be developed if the equations are second-order or higher, even if they are non-linear. The method of weighted residuals can be used to approximate the weighted-integral form of any equation.

The method of weighted residuals can be described in its generality by considering the operator equation. Since the latter form does not include any of the specified boundary condition of the problem, the approximation functions should be selected such that the approximate solution satisfies both natural and essential boundary conditions[2].

It should be recalled that the sole purpose of developping a weighted- integral statement of a differential equation is to have the means to obtain N linearly independent algebraic relations among coefficients c_j of the approximation.

The approximation must be cast in a form of n equations *which are defined as integrals over* Ω *and* Γ *i.e.*

$$\mathbf{F}_i = \int_\Omega \mathbf{E}(\hat{\phi})\ d\Omega + \int_\Gamma \mathbf{e}(\hat{\phi})\ d\Gamma, \qquad j = 1, \ldots, n \tag{D.39}$$

Immediatly, we note that the basic definitions of the finite element process previously given apply, as *for integrable functions*

$$\int_\Omega (\ )\ d\Omega \equiv \sum \int_{\Omega^e} (\ )\ d\Omega \tag{D.40}$$

and

$$\int_\Gamma (\ )\ d\Gamma \equiv \sum \int_{\Gamma^e} (\ )\ d\Gamma \tag{D.41}$$

in which Ω^e, Γ^e represent "element subdomains". The problem of how the integrals of approximations are formed is thus the first, crucial, question of casting a problem in a finite element form.

Weighted integral statements: It is obviously possible to replace the governing equations by an integral statement in all respects equivalent, *i.e.* multiply the equations by the weight functions. The weight functions can be selected independently of the approximation functions, but are required to be linearly independent (so that the resulting algebraic equations are linearly independent).

[2] what are natural and essential boundary conditions?

In the method of weighted residuals, an attempt is made to force this residual to zero, in an average sense, through selections of the constants $c_j (j = 1, \ldots, M)$. The c_j are calculated by satisfying the constraints which arise when setting the weighted integrals of the residual to zero, *i.e.*

$$\int_{\Omega} \mathcal{R}(x) w_i(x)\ dx \qquad (i = 1, \ldots, M) \tag{D.42}$$

Equation above can now be written as[3]

$$\langle \mathcal{R}(x), w_i \rangle = 0 \qquad (i = 1, \ldots, M) \tag{D.43}$$

From the last equation, M equations are obtained which, in conjunction with the boundary conditions, can be solved for M values of c_j. There are several weighted residual methods and each is distinguished by choice of weighting functions w_i.
Therefore,

$$\int_{\Omega} \mathbf{W}^T \mathcal{D}(\phi)\ d\Omega + \overline{\mathbf{W}}^T \mathbf{B}(\phi)\ d\Gamma \tag{D.44}$$

in which $\mathbf{W}$ and $\overline{\mathbf{W}}$ are completely arbitrary, "weighting" functions. Immediately an approximation is possible in an integral form by choosing specific functions W_j and $\overline{W}^T$ and writing

$$\mathbf{F}_j = \int_{\Omega} \mathbf{W}_j^T \mathcal{D}(\hat{\phi})\ d\Omega + \overline{\mathbf{W}}_j^T \mathbf{B}(\hat{\phi})\ d\Gamma \tag{D.45}$$

This process is known as the weighted residual method if $\mathcal{D}(\hat{\phi})$ and $\mathbf{B}(\hat{\phi})$ are recognized as *residuals* by which the approximation misses the zero value required.

NOTE: the weighted residual form given by equation (?) arises in a form which can be obtained from such equations by the use of *integration by parts*. Such integration reduces the continuity requirement imposed on both functions W and N by 'integrability'. This relaxation of requirements is known mathematically as a 'weak formulation' of the problem.

3. **Numerical Solver**

Finite element equation (for each element)

$$\sum_{j=1}^{n} K_{ij}^e u_j^e = q^e \tag{D.46}$$

4. **Galerkin Method**

Classical procedures of Galerkin's method, collocation etc. are immediately recognized. The Galerkin process, in which the weighting function and the trial function are identical ($W_j \equiv N_j$), is the most popular form used in finite element analysis.

Galerkin's method is formulated by selecting the basis functions (also known as coordinate functions and bases) $\phi_j(x)$ as weighting functions. Thus the weighted residual equations become

$$\langle \mathcal{R}(x), \phi_i(x) \rangle = 0 \qquad (i = 1, \ldots, M) \tag{D.47}$$

There is a second interpretation of the Galerkin method that may provide additional insight into the important role played by the coordinate functions. As in the most general case of Ritz procedure, the $\phi_j(x)$ are formally required to satisfy the boundary conditions imposed on the governing equation. Galerkin's method is used to numerically solve the St-Venant equations.

[3]The inner product (or scalar product) of two functions v and w is written as $\langle v, w \rangle \equiv \int_{\Omega} v \cdot w\ d\Omega$

D.5 Discretized Equations

D.5.1 Steady state equation

In this section we describe the processus for discretizing the St- Venant equations following the steps in chapter ?.

1. step 1

   Multiply the entire (each equation) equation by the weignt functions, and integrate over the domain of the problem.

$$\begin{aligned}
\delta w &= \int\int_A \Big(\delta U\left(B_1(U,V,H)\right) - \nu\delta U(\partial_{xx}U + \partial_{yy}U)\Big)\, dA \\
&+ \int\int_A \Big(\delta V(B_2(U,V,H)) - \nu\delta V(\partial_{xx}V + \partial_{yy}V)\Big)\, dA \\
&+ \int\int_A \delta H(B_3(U,V,H))\, dA \\
&= 0
\end{aligned} \tag{D.48}$$

   where $\delta U, \delta V, \delta H$ represent the weighting function and:

$$\begin{aligned}
B_1(U;V;H) &= U\partial_x U + V\partial_y U + g\partial_x H + \frac{gU\sqrt{U^2+V^2}}{C^2 h}, \\
B_2(U;V;H) &= U\partial_x V + V\partial_y V + g\partial_y H + \frac{gV\sqrt{U^2+V^2}}{C^2 h}, \\
B_3(U;V;H) &= \partial_x(hU) + \partial_y(hV)
\end{aligned}$$

   with the notation used

$$\partial_x \equiv \frac{\partial}{\partial x}; \qquad \partial_y \equiv \frac{\partial}{\partial y}$$
$$\partial_{xx} \equiv \frac{\partial^2}{\partial x^2} \qquad \partial_{yy} \equiv \frac{\partial^2}{\partial y^2}$$

   We shall call the statement (1) the *weighted-integral* or *weighted-residual* equivalent to the original equation.

2. step 2

3. step 3

Appendix E

Non-Linear Solver

In this Appendix we present non-linear solver to compute the non- linear part (convection term of the St-Venant equations. These solvers are used in Finite element Method and Finite Volume Method.

E.1 Newton Raphson Technic

One of the most popular used to solve non-linear problem is Newton- Raphson technic. For a non-linear problem we can define a Newton linearization in iterative way. The non-linear system $K(U)\ U = F$, Newton iteration U^{n+1} is (Taylor serie expansion)

$$K(U^{n+1}) = K(U^n + \Delta U) = K(U^n) + \frac{\partial K}{\partial U}\ \Delta U^n = F \tag{E.1}$$

The terms of higher order in ΔU has been neglected. In indicial notation

$$KT_{ij} = K_{ij} + \frac{dK_{il}}{dU_j} U_l \tag{E.2}$$

The Jacobian with respect to U, $\frac{dK_{il}}{dU_j}$, is called tangential matrix. It is this equation which define iterative process

$$KT\ \Delta U^n = -R^n \tag{E.3}$$

We have a linear at each iteration. Then we have to inverse KT to obtain ΔU^n.

The system of equations D-? is resolved iteratively using the following algorithm:

$$[K_t]^i \{\Delta U\}^{i+1} = \{F\} - [K]^i \{U\}^i = \{R\}^i \tag{E.4}$$

with

$$[K_t]^i = \frac{\partial \{R\}}{\partial \langle U \rangle}_{U=U^i} \tag{E.5}$$

therefore:

$$\{\Delta U\}^{i+1} = \left[K_t^i\right]^{-1} \{R\}^i \tag{E.6}$$

and using a convergence factoe:

$$\{U\}^{i+1} = \{U\}^i + \alpha \{\Delta U\}^{i+1} \tag{E.7}$$

where R is the vector of residues, K the matrix of these equations, U is the unknown and i the iteration number.

The calculation comprises the following steps:

- initialization of vector U^0 at iteration zero
- calculation of R^i using (a)
- calculation of K^i using (b)
- calculation of ΔU^{i+1} using (c)
- calculation of U^{i+1} using (d)
- verification of convergence
- adjustment of the convergence coefficient, if necessary
- repeated operation until the desired accuracy is reached

The following criterion is used to evaluate the desired accuracy

$$\frac{\max\{U^{i+1} - U^i\}}{\max\{U^{i+1}\}} < \epsilon \tag{E.8}$$

which offers the advantage that the values of U may be very small.

E.2 Godunov Algorithm

The reconstruction is done by the application of the opérator R at the states variables $\overline{v}^n$ (mean values) which represent the solution at time t_n. The operator $R(\cdot;\overline{v}^n)$ is a polynomial of high order which approximate the values at node of the exact solution.

The reconstruction procedure for two adjacent cell at the interface produce two constant states (left) and (right), here we recognize the Riemann problem. The Riemann solver solve in time this initial discontinuous solution around the interface, then we define the operator E_τ (time evolution interval τ). The Riemann solver compute the solution as a superposition of elementary waves which propagate along the characteristics and interact in a non-linear way $u_l - u_r = \sum_p (\beta_p - \alpha_p)\ r_p$ as mentioned before.

Finaly, the solution at time t_{n+1} is averaged to obtain $\overline{v}^{n+1}$. This mean value is represented by the application of the operator 'cell-average' $A_{\Delta x}$ where Δx is the space step.

Formaly the Godunov scheme and its extension at higher order can be written as follow:

$$\overline{v}^{n+1} = A_{\Delta x} \cdot E_{\Delta t} \cdot R(\cdot;\overline{v}^n) \tag{E.9}$$

where the application of these three operator is done in the following sequence: R recontruction operator, $E_{\Delta t}$ time evolution operator ($u(x,t) = E_{\Delta t} u(x,0)$), and $A_{\Delta x}$ average on the volume, update the solution of the mean value from time t_n to final time t_{n+1}. All schemes that can be put in the form of a product operator as the one just mentioned are generalisation of the Godunov scheme. Particularly, if R is a constant piecewise reconstruction (Godunov),

$$R(x,\overline{v}^n) = \overline{v}_j^n \qquad\qquad x \in\ [x_{j-1/2}, x_{j+1/2}] \tag{E.10}$$

if R is a linear piecewise reconstruction

$$R(x,\overline{v}^n) = \overline{v}_j^n + s_j^n (x - x_j) \qquad x \in [x_{j-1/2}, x_{j+1/2}] \tag{E.11}$$

où s_j^n est un estimé de la pente à l'intérieur de la cellule

$$s_j^n = \frac{\partial v}{\partial x}(x,t) + \mathcal{O}(h). \tag{E.12}$$

This algorithm give a numerical discretization which is upwind built and non-linear, because as mentioned earlier, the Riemann solver compute the Riemann problem solution through the interaction of elementary waves (shock, rarefaction, shock discontinuity ...) which propagate along the local characteristics. The main desavantage with this approach is the complexity introduced in the difference scheme by the application of Riemann sover and the cost in term of the number of operation and the CPU time [37]. According to [53] the basic scheme (constant piecewise) Godunov at first order, cost twice as much for each cell by time step than the MacCormack scheme at second order with an artificial viscosity. But when discontinuity are present in the flow these schemes has demonstrate that the computation of the solution based on the Riemann solvers give better result than others schemes.

Appendix F

Classification of Partial Differential Equations

In this appendix we introduce some of the fundamental concept when searching for a solution of the partial differential equations. The of these is the concept of charateristic. As we will see in the next section, the charateristic contains all the information about the property of the flow. It is from these characteristic that we can determine the influence and dependence zone of the equations. This is a crucial point we want to determine if the flow is supercritical or subcritical, it has impact on the choice of the numerical scheme. Then we are ready to classified the equation according to the nature of the flow. As we have seen in previous chapter, the equations of fluid mechanics are balance between the different forces acting on the fluid in movement. Depending on the nature and the physical magnitude of the phenomena to be modelled (convection, pressure diffusion, bathymetry, Coriolis and friction) may be of significant magnitude when compared to the others. Mathematical nature of the equations will depend on the physical nature, we can then classified these equations in three types. Once the classification done, ... numerical analysis. But before we discretize, we need to know if our approximation ... must satisfy conditions of consistency, stability and convergence.

F.1 Theory of Characteristics

Philosophy of the characteristics.

F.2 Zone of influence and dependence

Following the classification of partial differential equations, there is the most important concept: influence and dependence zone. Each differential equation has is own zone of influence and dependence. We will see that this notion is crucial ... choosing the suitable numerical scheme to describe the physical phenomena under consideration.

F.3 Classification of Equations

Partial differential equations are classified as parabolic, hyperbolic or elliptic based on the properties of the equations. It is not surprising, therefore, that the selection of a numerical scheme for solving a given problem is generally based onthe type of equations encountered. The method of characteristics, for example, is often

applied to equations of hyperbolic type while parabolic equations can often be solved effectively with finite difference methods.

The classification of partial differential equations is formally developed utilizing the theory of characteristics. This development provides considerable insight into the theoretical foundation of the classification. The essential elements of the resulting cataloging procedure can be summarized by considering a second-order partial differential equation in the function u for the two independent variables x and y such as

$$a\frac{\partial^2 u}{\partial x^2} + b\frac{\partial^2 u}{\partial x \partial y} + c\frac{\partial^2 u}{\partial y^2} + d(x, y, u, \frac{\partial u}{\partial x}, \frac{\partial u}{\partial y}) = 0 \tag{F.1}$$

which may be either linear or quasi-linear. Equation above is classified as hyperbolic, parabolic, or elliptic in a region B when the discriminant $b^2 - 4ac$ is positive, zero or negative, respectively. Because the coefficients a, b, and c are, in general, functions of the independent variables, the classification of an equation may change at different locations in the region B in which the equation is defined.

Hyperbolic Equations: $b^2 - 4ac > 0$ **(propagation dominated)**

A hyperbolic partial differential equation defined in region B is characterized by having a positive discriminant everywhere within B. Equations of this type require both initial and boundary conditions (see Fig. ?). The initial conditions are the values of the functions u and its firts time derivative defined at some time t_0. The boundary conditions may consist of the value of the function (Dirichlet type), its normal derivative (Neumann type), or a combination of the function and its normal derivative (mixed type) on the region of definition. An important equation of this type is the one-dimensional wave equation which describes the longitudinal velocity u of water in a channel with a wave being generated at one end

$$a^2\frac{\partial^2 u}{\partial x^2} = \frac{\partial^2 u}{\partial t^2} \tag{F.2}$$

where a is the celerity of the wave.

Parabolic Equations: $b^2 - 4ac = 0$ **(damped diffusion)**

A parabolic partial differential equation is characterized by a zero discriminant at all points within the region B over which the equation is defined. Initial and boundary values are required for a properly posed problem (see Fig. ?). The initial value consists of the function u defined at some time t_0 and the boundary conditions are either the value of the function, its normal derivative, or a linear combination of the function and its normal derivative on the boundary.

An important equation of parabolic type is the one-dimensional heat flow equation

$$K\frac{\partial^2 u}{\partial x^2} = \frac{\partial u}{\partial t} \tag{F.3}$$

where $K = \kappa/\rho C$ is the specific heat, ρ the density of the material, and κ the thermal conductivity.

Elliptic equations: $b^2 - 4ac < 0$ **(diffusion dominated)**

Elliptic partial differential equations are characterized by a negative discriminant everywhere within the region over which the equations are defined. In contrast to parabolic and hyperbolic equations, which are propagated in an open domain, elliptic equations require boundary conditions specified over a closed boundary of the region B. Either the function$u$, its normal derivative, or a combination of the function and its normal derivative must be specified to assure a unique solution.

An important elliptic equation is Laplace's equation which describe the steady state temperature distribution in two space dimensions as

$$\left(\frac{\partial^2 u}{\partial x^2}\right) + \left(\frac{\partial^2 u}{\partial y^2}\right) = 0 \tag{F.4}$$

The equivalent expression with a source term is called the Poisson equation and ahs the form

$$\left(\frac{\partial^2 u}{\partial x^2}\right) + \left(\frac{\partial^2 u}{\partial y^2}\right) = f(x, y) \tag{F.5}$$

This distinction is of paramount importance, since the numerical discretization and solution methods will have to take into account the differences between phenomena as distinctive and afr apart in their physical behaviour as diffusion and propagation. The former property is essentially independent of the flow direction acting in all directions and in the whole space domain, while the latter is essentially direction dominated and acts in specific regions of space defined by the wave-propagation directions.

Their mathematical properties are directly connected to the physical properties of the flow.

Flow configuration is the outcome of a balance between the effects of convective fluxes, diffusive fluxes and the external and internal sources. Therefore each of these contributions will influence the mathematical nature of the equations, particularly the competition between elliptic, parabolic and hyperbolic character of the system of equations.

Appendix G

Non-Conservative Form

G.1 Eulerian notation and Lagrangian derivatives

Let us consider an orthonormal reference frame which can be at rest (that is Galilean) or rotating with a solid-body rotation $\vec{\Omega}$ with respect to a "fixed" frame. A "fluid particle", of size large in comparision to the molecular scales and small in comparison to the Kolmogorov dissipation scale and located in $\vec{x}$ at time t, will have a velocity $\vec{u}(\vec{x},t)$ with respect to the reference frame. The components of the velocity will be $u_i(\vec{x},t)$. Let $\rho(\vec{x},t)$ be the density of the fluid element passing by $\vec{x}$ at time t. This notation corresponds to the Eulerian formulation. Let $A(\vec{x},t)$ be any quantity associated with the motion of the fluid. When the fluid particle considered above moves, it produces a variation of A, and the derivative of A following the fluid motion in the reference frame will be denoted DA/Dt. The operator D/Dt is the Lagrangian derivative ("Lagragian" means here "following the motion", and is not to be confused with Lagrange approaches in analytical mechanics). Let consider some quantity at a point 1 at time $t_1, f(x_1,y_1,z_1,t_1)$ and another point 2 at time $t_2, f(x_2,y_2,z_2,t_2)$. Taylor expansion around point 1.

$$f_2 = f_1 + \Big(\frac{\partial f}{\partial x}\Big)_{x_1}(x_2-x_1) + \Big(\frac{\partial f}{\partial y}\Big)_{y_1}(y_2-y_1) + \Big(\frac{\partial f}{\partial z}\Big)_{z_1}(z_2-z_1) + \Big(\frac{\partial f}{\partial t}\Big)_{t_1}(t_2-t_1) \tag{G.1}$$

Higher terms have been neglected. Rate of variation of the quantity along the trajectory with the coordinate system attached to the particle. Divide this equation by t_2-t_1 which is the time taken to go from point 1 to point 2, we obtain

$$\frac{f_2-f_1}{t_2-t_1} = \frac{(x_2-x_1)}{(t_2-t_1)}\Big(\frac{\partial f}{\partial x}\Big)_{x_1} + \frac{(y_2-y_1)}{(t_2-t_1)}\Big(\frac{\partial f}{\partial y}\Big)_{y_1} + \frac{(z_2-z_1)}{(t_2-t_1)}\Big(\frac{\partial f}{\partial z}\Big)_{z_1} + \Big(\frac{\partial f}{\partial t}\Big)_{t_1} \tag{G.2}$$

Let define a new quantity which is the variation rate

$$\lim_{t_2\to t_1}\Big(\frac{f_2-f_1}{t_2-t_1}\Big) \equiv \frac{Df}{Dt} \tag{G.3}$$

Repeat... same for all other terms

$$\lim_{t_2\to t_1}\Big(\frac{x_2-x_1}{t_2-t_1}\Big) \equiv u \quad \lim_{t_2\to t_1}\Big(\frac{y_2-y_1}{t_2-t_1}\Big) \equiv v \quad \lim_{t_2\to t_1}\Big(\frac{z_2-z_1}{t_2-t_1}\Big) \equiv w \tag{G.4}$$

u,v,w which represent the velocity components of the coordinate system. We can write the last expression as

$$\frac{Df}{Dt} = \Big(\frac{\partial f}{\partial t}\Big)_{t_1} + u\,\Big(\frac{\partial f}{\partial x}\Big)_{x_1} + v\,\Big(\frac{\partial f}{\partial y}\Big)_{y_1} + w\,\Big(\frac{\partial f}{\partial z}\Big)_{z_1} \tag{G.5}$$

Then the material derivative or Lagrangian can be define as

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \vec{v} \cdot \vec{\nabla} \tag{G.6}$$

Signification of each term: first of all the term $\partial/\partial t$ is the local variation of the quantity du of the variation at this point. The second term (convective) variation because i move from point 1 to point 2 where the propriety are different. The following analogy can be made: let say i am in a river and i want to measure the water temperature. I can stand at one point in the river and let the water go through myself and feel the change of temperature at this point (local, $\partial/\partial t$) or go with the current and feel the change of temperature along the trajectory (convective, D/Dt). These last two principles are valid in the case compressible fluid (*the density is constant*).

G.2 Navier-Stokes equations

Infinitesimal element approach

Now let δV be the volume of the small fluid particle. It can be shown, for instance by the change of coordinates $\vec{x} \to \vec{x}_0$, where $\vec{x}_0$ is the original position at some initial time t_0 of the fluid particle located in $\vec{x}$ at time $t \geq t_0$, that the divergence of the velocity is given by

$$\vec{\nabla} \cdot \vec{u} = \frac{1}{\delta v} \frac{D\ \delta V}{Dt} \tag{G.7}$$

The continuity equation

This equation is the mass conservation: let $\delta m = \rho \delta V$ be the mass of the fluid particle. It is conserved following the fluid motion, since average exchanges of mass with the surrounding fluid, which are due to molecular diffusion across the boundary $\partial(\delta V)$ of δV, will be zero for macroscopic time scales large in comparison to the molecular time scales. Hence, the logarithmic Lagrangian derivative of δm wil also be zero, and one obtains

$$\frac{1}{\rho} \frac{D\rho}{Dt} + \frac{1}{\delta V} \frac{D\ \delta V}{Dt} = 0 \tag{G.8}$$

or equivalently because of (?)

$$\frac{1}{\rho} \frac{D\rho}{Dt} + \vec{\nabla} \cdot \vec{u} = 0 \tag{G.9}$$

which is the continuity equation. The particular case of incompressibility (conservation of volumes following the fluid motion) reduces to

$$\vec{\nabla} \cdot \vec{u} = 0, \text{or} \frac{D\rho}{Dt} = 0 \tag{G.10}$$

The conservation of momentum

The second law of motion is obtained by applying to the fluid particle the fundamental principle of Newtonian mechanics, namely

$$\delta m \frac{D\vec{u}}{Dt} = [\text{body forces}] + [\text{surface forces}] \tag{G.11}$$

The body forces applied to the fluid particle are gravity $\delta m \vec{g}$, the Coriolis force (if any) $-2\delta m \vec{\Omega} \wedge \vec{u}$, and possible other external forces (like the Lorentz force in the case of an electrically-conducting flow).We recall that $\vec{\Omega}$ is the rotation of the earth.

As shown in Batchelor (1967) the fact that the surface forces applied to the fluid particle have to be proportional to δV in order to balance the two order terms of (?) implies the existence of a strain tensor σ_{ij} such that the force exerted on a small surface $d\Sigma$ oriented by a normal unit vector $\vec{n}$ is

$$df_i = \sigma_{ij} n_j d\Sigma \tag{G.12}$$

In a Newtonian fluid, the strain tensor is assumed to be linear with respect to the deformation tensor

$$e_{ij} = \frac{1}{2}\Big(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\Big) \tag{G.13}$$

and isotropic. Hence, it is found (Batchelor (1967)) that

$$\sigma_{ij} = -p\delta_{ij} + \mu\Big[\Big(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\Big) - \frac{2}{3}\vec{\nabla}\cdot\vec{u}\delta_{ij}\Big] \tag{G.14}$$

where δ_{ij} is the Kronecker tensor and μ the dynamic viscosity. Following Batchelor(1967), we have defined the pressure p with the aid of the trace of the strain tensor ($p = -(1/3)\sigma_{ii}$, with summation upon the i). The dynamic viscosity μ may vary with the physical properties of the fluid. Therefore after integration of the surface forces over the surface of the fluid particle

$$\frac{Du_i}{Dt} = \Big(\vec{g} - 2\ \vec{\Omega}\wedge\vec{u}\Big)_i + \frac{1}{\rho}\frac{\partial\sigma_{ij}}{\partial x_j} \tag{G.15}$$

or equivalently

$$\frac{Du_i}{Dt} = \Big(\vec{g} - 2\ \vec{\Omega}\wedge\vec{u}\Big)_i - \frac{1}{\rho}\frac{\partial p}{\partial x_j} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\ \mu\left[\left(\frac{\partial u_i}{\partial x_j}\right) + \left(\frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\vec{\nabla}\cdot\vec{u}\ \delta_{ij}\right] \tag{G.16}$$

which is the momentum equation for a compressible fluid. The flows we will consider in the present thesis will have a constant uniform dynamic viscosity μ, so that (?) yields in this case

$$\frac{D\vec{u}}{Dt} = \vec{g} - 2\vec{\Omega}\wedge\vec{u} - \frac{1}{\rho}\vec{\nabla}p + \nu\left[\nabla^2\vec{u} + \frac{1}{3}\vec{\nabla}\ \left(\vec{\nabla}\cdot\vec{u}\right)\right] \tag{G.17}$$

with

$$\frac{D\vec{u}}{Dt} = \frac{\partial\vec{u}}{\partial t} + \vec{u}\cdot\vec{\nabla}\vec{u} \tag{G.18}$$

where $\nu = \mu/\rho$ is called the kinematic viscosity. The term proport ional to ν in the r.h.s. of eq. (?) characterizes the diffusion of momentum due to rthe molecular exchanges between the fluid particle and the fluid surrounding it. When the velocity is nondivergent, which will very often be the case in the present thesis, the dissipative term deduces to $\nu\nabla^2\vec{u}$. Notice that if other forces exist, they have to be added to the r.h.s. of (?).

The system of equations (?) and (?), supplemented by an energy equation is called the Navier-Stokes equations. The momentum equation (?) without the viscous and Coriolis terms is called Euler equations, and was derived by Leonhard Euler in the middle of the 18th century.

G.3 Shallow-Water Equations

These equations are also known as the "shallow-water equations". We consider the Navier-Stokes equations locally on a sphere for a fluid of constant uniform density ρ_0, that is

$$\frac{D\vec{u}}{Dt} = -\frac{1}{\rho_0}\vec{\nabla}p + \vec{g} - f\vec{z}\wedge\vec{u} + \nu\nabla^2\vec{u} \tag{G.19}$$

$$\vec{\nabla}\cdot\vec{u} = 0 \tag{G.20}$$

$\vec{g}$ and $\vec{z}$ being parallel. The fluid is assumed to have a free surface of mean elevation H and to lie above a topography of height $\tau(x, y)$. $h(x, y, t)$ is the depth of the fluid layer and $\eta(x, y, t)$ the elevation of the free surface with respect to H (see figure ?). Hence we have

$$h(x, y, t) + \tau(x, y) = H + \eta(x, y, t) \tag{G.21}$$

The pressure at the free surface is uniform and equal to p_0. The assumption of shallowness actually means that one assumes the pressure is hydrostatically distributed along the vertical, that is,

$$p(x, y, z, t) = p_0 + \rho_0 g(h + \tau - z) \tag{G.22}$$

and that the horizontal velocity filed $\vec{u}_H = (u, v, 0)$ depends only on the horizontal space variables x and y and on the time. The vertical velocity $w(x, y, z, t)$ still depends on the vertical coordinate, in order to allow vertical variations of the free surface. With these assumptions, and integrating the continuity equation along the vertical, one obtains the Barré de Saint-Venant equations

$$\frac{D_H u}{Dt} = -g\frac{\partial \eta}{\partial x} + fv + \nu \nabla_H^2 u \tag{G.23}$$

$$\frac{D_H v}{Dt} = -g\frac{\partial \eta}{\partial y} - fu + \nu \nabla_H^2 v \tag{G.24}$$

$$\frac{D_H h}{Dt} = -h\vec{\nabla}_{H.}\vec{u} \tag{G.25}$$

is the derivative following the horizontal motion, and ∇_H^2 and $(\vec{\nabla}_{H.})$ stand respectively for the horizontal Laplacian and divergence operators.

Non-conservative form Momentum equation is given by the following expressions:

- x component

$$\partial_t U + (\vec{U} \cdot \vec{\nabla})U - fV + g\partial_x H + \frac{gU\sqrt{U^2 + V^2}}{C^2 h} - \vec{\nabla} \cdot (\nu_t \vec{\nabla} U) = 0 \tag{G.26}$$

- y component

$$\partial_t V + (\vec{U} \cdot \vec{\nabla})V + fU + g\partial_y H + \frac{gV\sqrt{U^2 + V^2}}{C^2 h} - \vec{\nabla} \cdot (\nu_t \vec{\nabla} V) = 0 \tag{G.27}$$

The continuity equation:

$$\partial_t h + \vec{\nabla} \cdot (h\vec{U}) = 0 \tag{G.28}$$

wher ν_t kinematic viscosity (*i.e. moléculaire + turbulente*), C Chezy coefficient which is expressed in term of the Manning paremeter and the water depth.

$$C = \frac{1}{n} h^{1/6} \tag{G.29}$$

Notation used:

$$\partial_t \equiv \frac{\partial}{\partial t} \qquad \partial_x \equiv \frac{\partial}{\partial x} \qquad \partial_y \equiv \frac{\partial}{\partial y}$$

Here are some explication about the different terms of the equation of momentum:

1. $\frac{gU\sqrt{U^2+V^2}}{C^2 h}$ friction term, Manning which represent a grade line energy
2. $g\nabla_x H$ pressure gradient obtained from Shallow-Water approximation.
3. $\vec{\nabla} \cdot (\nu_t \vec{\nabla} U)$ diffusion term

The others parameters are g: gravitational acceleration, f : Coriolis coefficient ($f = 2\Omega sin\phi$), where ϕ lattitude angle, h: water depth (H-Z) with H water level and Z the bathymetry.

Bibliography

[1] Ahlander K. *An Object-Oriented Approach To Construct PDE Solvers* Uppsala University, Sweden (1996)

[2] Belanger J. *Validating Shock-Capturing Scheme ...* Technical Report ???, Elligno Inc. (2007)

[3] Belanger J. and *al. Real-Case ...* Technical Report ???, Elligno Inc. (2000)

[4] Akhilesh Kuma Jha, Juichiro akiyama Masaru Ura, *First and second order flux difference splitting schemes for da m-break problem* Journal of Hydraulic Engineering, vol. 121, pp. 877-884 (1995)

[5] Alcrudo F., Garcia Navarro P., *A high resolution Godunov type scheme in finite volume for the 2-D Shallow-water equations* International Journal for Numerical Methods in Fluids," vol. 16, pp. 489-505 (1993)

[6] Alcrudo F., Garcia Navarro P., Saviron, J.M., *Flux difference splitting for the 1-D open channel flow equations* International Journal for Numerical Methods in fluids, vol. 14, pp. 1009-1018 (1992)

[7] lcrudo F., Garcia Navaro P. (1994), "Computing two-dimensional flood propagation with a high resolution extension of Mc Cormack method." Modelling of flood propagation over initialy dry areas, pp. 3-17.

[8] Anderson J. *Modern compressible flow* Mc Graw Hill (1990)

[9] Brooks A. N. and Hughes J.R. *Streamline upwind/Petrov-Galerkin formulations for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations* Computer Methods in applied mechanics and engineering,32,pp.199-259 North-Holland Pubishing Company (1982)

[10] Casper J., *Finite volume implementation of high-order essentially non-oscillatory schemes in two-dimensions*, AIAA Journal,vol. 30, pp.2829-2835 (1992)

[11] Chandrasekhar S. *Hydrodynamic and Hydromagnetic Stability*, Dover Publications inc., New-York

[12] Colella P., Woodward P., (1984), *The piecewise- parabolic method (ppm) for gas dynamical simulations*, J. Comput. Phys., vol. 54, pp. 174-201.

[13] Cunge C.A. *Simulation des écoulements à surface libre dans les canaux et les rivières* Université de Grenoble (19

[14] hatt G. Touzot G. *Une présentation de la méthode des éléments finis* Maloine, Paris (1981)

[15] Fennema R.J., Chaudry M.H., *Explicit methods for 2-D transient free surface flows*, Journal of Hydraulics Division, ASCE, vol. 116, pp. 1013-1034, (1990)

[16] arcia Navarro P., Alcrudo F., (1992), *1-D open channel flow simulation using TVD Mc Cormack scheme*, Journal of Hydraulics Engineering, ASCE, vol. 118, No. 10, pp. 1359-1372, (1992)

[17] Garcia R. and Kahawita R. *Numerical Solution of the St-Venant Equations with the MacCormack Finite-Difference Scheme* International Journal for Numerical Methods in Fluids, vol. 6, 259-274 (1986)

[18] Gerald C.F., Weatly P.O., *Applied Numerical Analysis*,Addison-Wesley, (1990)

[19] Glaister P., *Approximate Riemann solutions of the Shallow Water equations*, Journal of Hydraulic Research, vol. 26 No.3, pp. 293-306, (1988)

[20] Goldstein H. *Classical Mechanics*. Addison–Wesley Co. (1980)

[21] Harten A., *On a class of high-resolution total-variation-stable finite difference schemes*, SIAM J. Numer. Anal., vol.21, pp. 1-23, (1984)

[22] Harten A., *High resolution scheme for hyperbolic conservation laws*, J. Comput. Phys., vol. 49, pp. 357-393, (1983)

[23] Harten A., Osher S., *Uniformly high-order accurate non-oscillatory schemes I*, SIAM Journal of Numerical Analysis, vol. 24, pp. 279-309, (1987)

[24] Harten A., Enquist B., Osher S., Chakravarthy S.R., *Uniformly high-order accurate non-oscillatory schemes III*, Journal of Computational Physics, vol. 71, pp. 231-303. (1987)

[25] Hirsch C. *Numerical computation of external and internal flows: vol. 1 Fundamental of numerical discretization* Wiley and Sons, (1990)

[26] Hirsch C. *Numerical computation of external and internal flows: vol. 2 Computation methods for inviscid and viscous flows* Wiley and Sons (1990)

[27] Mohamed N., Brugnot G., *Free surface flow modelling on a complex topography*" Modelling of Flood Propagation over Initially Dry Areas, pp. 298-308 (1994) (ÉLÉMENTS ET VOLUMES FINIS)

[28] Majda A., (1984), *Compressible fluid flow and systems of conservation laws in several space variables*, Springer-Verlag (1984) (A VERIFIER)

[29] Manzini M., *Applications des schémas de capture de choc à des problèmes de géophysique et d'astrophysique* Thèse de doctorat, CERFACS, France (?)

[30] Osher S., Chakravarthy S.R., *High resolution schemes and the entropy condition*, SIAM J. Numer. Anal., vol. 21, pp. 955-984 (1984)

[31] Paquier A., *New methods for modelling Dam- Break wave*, Modelling of flood propagation over Initially Dry Areas, pp. 229-239 (1994)

[32] Kahawita R., Tchamen G.W. *The Numerical Simulation of Wetting and Drying Areas using Rieman Solvers* Modelling of Flood Propagation Over Initially Dry Areas, Proceedings of the Specialty Conference co-sponsored by ASCE-CNR/GNDCI-ENEL held in Milan, Italy 29 june-1 july 1994

[33] Landau L.D. Lifshitz E.M. *Fluids Mechanics* Pergamon Press (1989)

[34] Landau L.D. Lifshitz E.M., *Physique Statistique, partie 2* Édition Mir.

[35] Leclerc M., Bellemare J.F, Dumas G., Dhatt G., *A finite element model of estuarian and river flows with moving boundaries*, Adv. Water Resources, Vol. 13, No.4, pp. 158-168 (1990)

[36] Leveque R.J. *Numerical Methods for conservation laws* Spinger-Verlag (1990)

[37] Manzini M. *Applications des schémas de capture de choc à des problèmes de géophysique et d'astrophysique*, Thèse de doctorat, CERFACS, France.

[38] Nujic M. *Efficient implementation of non-osccilatory schemes for the computation of the surface flows*, Journal of Hydraulic Research, Vol.33 No.1, pp. 101-111 (1995)

[39] Pearson J.M. *A theory of waves*, Allynand Bacon inc. (1966)

[40] Pedolsky J. *Geophysical Fluid Dynamics*, 2nd ed., Springer-Verlag. (1986)

[41] Pironneau O. *Méthodes des éléments finis pour les fluides* Masson (1988)

[42] Pinder G.F. Gray W.G.*Finite element simulation in surface and subsurface in hydrology* Academic Press (1977)

[43] Le Pourhiet A. *Résolution Numérique des équations aux dérivées partielles*, CEPADUES-édition (1987)

[44] Reddy J.N. *An introduction to the finite element method*,Mc Graw-Hill (1993)

[45] Shu C.W., Osher S., *Efficient implementation of essentially non-oscillatory shock capturing schemes*, Journal of Comput. Phys., vol. 77, pp. 439-471, (1988)

[46] Strickwerda J.C. *Finite difference schemes and Partial differential equations*, Chapman & Hall (1989)

[47] Tchamen G. Kahawita R.*The numerical simulation of wetting and drying areas using Riemann solvers*, Modelling of Flood Propagation Over Initially Dry Areas, pp. 127-139 (1994)

[48] Tritton D.J., *Physical Fluid Dynamics*, Oxford (1988)

[49] Van Leer B., *Towards the ultimate conservative finite difference scheme II. Monotonicity and conservation combined in asecond order scheme*, J. Comput. Phys., vol. 14, pp.361-376 (1974)

[50] Van Leer B., *Towards the ultimate conservative finite difference scheme IV. A new approach to numerical convection*, J. Comput. Phys., vol. 23, pp. 276-298 (1977)

[51] Vila J.P., *Simplified Godunov schemes for 2x2 systems of conservation laws*, SIAM Journal of Numerical Analysis, vol. 23, No.6, December 1986, pp. 1173-1192 (1994)

[52] Whitham G., *Linear and nonlinear waves*, Wiley-Interscience (1974)

[53] Woodward P., Collela P., *The Numerical simulation of two dimensional fluid flow with strong shock*, Journal of Comput. Phys., vol. 54, pp. 115-173 (1984)

[54] Yang J.Y., Hsu C.A., Chang S.H., *Computation of free surface flows, Part I: one dimensional dam-break flow*, Journal of Hydraulic Research, Vol. 31 No.1 pp. 19-34 (1993)

[55] Yang J.Y., *Uniformly second order essentially non-oscillatory schemes for the Euler equations*, AIAA Journal, Vol. 28, No. 12, pp. 2069-20? (1990)

[56] Yang J.Y., *Third-order non-oscilaltory schemes for the Euler equations*, AIAA Journal, Vol. 29, No. 10, pp. 1611-1618 (1991)

[57] Zhang H., Hassanzadeh Y., Nguyen Duc L., Kahawita R., *A 1-D numerical model applied to dam-break flows on dry beds* Journal of Hydraulic Research, vol. 30, No.2, pp. 211-224 (1992)

[58] Zhao O.H., Shen H.W., Tabios III G.Q., Lai J.S., Tan W.Y., *Finite volume two-dimensional unsteady flow model for river bassins*, Journal of Hydraulic Engineering, vol. 120, pp. 863-883 (1994)